

Companion Web site includes templates, custom properties, and macros to help you administer SolidWorks

Matt Lombard

SolidWorks[®] Administration

Install, configure, and
manage SolidWorks

Become your company's
CAD administrator

Establish best practices
and write standards



Bible

The book you need to succeed!

SolidWorks®

Administration Bible

SolidWorks® Administration Bible

Matt Lombard



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About the Author

Matt Lombard is an independent mechanical design consultant. He spent time as the Engineering Services Manager at a high tech manufacturer where his job included the work of a Documentation Manager as well as a mixed environment CAD Manager with both direct and indirect staff. Matt also has worked as a consultant and has performed dozens of implementations covering the setup and optimization of SolidWorks, Toolbox, and PDMWorks.

Because of the popularity of his SolidWorks blog Dezinstuff (<http://Dezinstuff.com/blog>), and sessions at SolidWorks World, he has built a reputation as the go-to resource for everything SolidWorks.

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Contents

| | |
|------------------------------|------------|
| Acknowledgments | xxv |
|------------------------------|------------|

| | |
|---------------------------|--------------|
| Introduction | xxvii |
|---------------------------|--------------|

| | |
|---|----------|
| Part I: Preparing for SolidWorks | 1 |
|---|----------|

| | |
|---|----------|
| Chapter 1: Defining CAD Administration | 3 |
|---|----------|

| | |
|---|----|
| Introducing CAD Administration | 4 |
| Defining CAD Administration | 5 |
| Comparing SolidWorks Administration to AutoCAD Administration | 7 |
| Comparing project physical scale | 7 |
| Managing file data | 7 |
| Process-based | 8 |
| Playing the Part: More CAD or Administration? | 8 |
| Distinguishing between CAD and design | 9 |
| Including Information Technology | 9 |
| Finding a balance | 10 |
| Defining Your Role | 11 |
| Establishing yourself as an expert user | 11 |
| Becoming a CAD mentor | 11 |
| Being a leader versus manager | 12 |
| Mentoring through leadership | 12 |
| Administering efficiently | 13 |
| Building a team | 15 |
| Summary | 16 |

| | |
|--|-----------|
| Chapter 2: Preparing Information Technology | 17 |
|--|-----------|

| | |
|--|----|
| Selecting Hardware | 18 |
| Building your own computers | 18 |
| Understanding the associated risks | 18 |
| Understanding CAD requirements | 19 |
| Selecting processors | 20 |
| Specifying memory and operating system | 20 |
| Selecting a video card | 21 |
| Picking a hard drive | 23 |

Contents

| | |
|--|----|
| Selecting monitors..... | 25 |
| Selecting a laptop | 25 |
| Selecting the processor | 25 |
| Deciding on a video card | 26 |
| Considering power | 26 |
| Choosing RAM | 27 |
| Picking your operating system | 27 |
| Selecting hard drives | 28 |
| Specifying a display | 28 |
| Touch and multi-touch screens | 28 |
| Understanding Network Requirements | 29 |
| Balancing network load and capacity | 29 |
| Understanding local and wide area networks | 30 |
| Building file storage..... | 31 |
| Creating a Backup Plan | 31 |
| Selecting a backup mechanism | 32 |
| Establishing a retrieval procedure..... | 32 |
| Securing data off-site | 33 |
| Imaging drives..... | 33 |
| Working with Antivirus Applications..... | 33 |
| Benefiting from antivirus software | 34 |
| Shortcoming in using antivirus software..... | 35 |
| Setting Up a CAD Workspace | 35 |
| Understanding ergonomics..... | 36 |
| Personalizing a workspace..... | 36 |
| Summary | 36 |

Chapter 3: Preparing Document Management.....37

| | |
|---|----|
| Naming Conventions | 38 |
| Choosing between intelligent and sequential conventions | 38 |
| Advantages of intelligent part numbering | 38 |
| Disadvantages of intelligent part numbering | 40 |
| Advantages and disadvantages of sequential part numbers | 42 |
| Comparing part numbers, document numbers, and filenames | 42 |
| Creating a hybrid intelligent-sequential system..... | 43 |
| Adding filenames to the mix..... | 44 |
| Using a descriptive naming convention | 45 |
| Using metadata..... | 45 |
| Metadata possibilities in PDM..... | 46 |
| Organizing Files..... | 47 |
| Creating libraries..... | 48 |
| Avoiding common mistakes | 49 |
| Suggesting options | 50 |
| Breaking references..... | 50 |

| | |
|---|----|
| Understanding SolidWorks document relationships..... | 51 |
| Organizing subassemblies | 52 |
| Using SolidWorks Explorer and SolidWorks or Windows Search | 53 |
| Controlling Revisions | 55 |
| Revisioning schemes..... | 56 |
| Revising models versus revising documents..... | 56 |
| Using different revisioning schemes for models and drawings | 57 |
| Storing revisions..... | 57 |
| Tracking revisions of models and documents | 58 |
| Changing revision levels for documents..... | 58 |
| Updating stored revisions..... | 59 |
| Putting revision levels in filenames or metadata | 59 |
| Revision levels in filenames | 60 |
| Revision levels in folder names | 60 |
| Mixing methods | 60 |
| Retrieving old revisions | 61 |
| Using Configurations to Store Revisions..... | 61 |
| Summary | 62 |

Chapter 4: Operating System Tools You Need to Know 63

| | |
|---|----|
| Managing Your Computer..... | 64 |
| Using the Task Scheduler..... | 64 |
| Using available triggers..... | 65 |
| Using available actions | 65 |
| Using the Create a Basic Task Wizard..... | 66 |
| Reading the Event Viewer..... | 69 |
| Sharing folders | 70 |
| Setting up Local Users and Groups..... | 70 |
| Monitoring Reliability and Performance | 71 |
| Performance | 72 |
| Reliability | 72 |
| Using the Device Manager..... | 74 |
| Managing storage | 74 |
| Services and Applications..... | 75 |
| Using the Windows Backup & Recovery tools..... | 76 |
| Backing up files | 76 |
| Back up computer | 80 |
| Using recovery tools | 80 |
| Using System protection and System Restore..... | 81 |
| Recommending Backup and recovery options | 83 |
| Understanding the BIOS..... | 83 |
| Working with the Windows Registry | 84 |
| Using Msconfig | 86 |
| Using the Startup tab..... | 87 |
| The Services tab | 87 |

Contents

| | |
|--|-----|
| The Boot tab..... | 88 |
| Using the Tools tab | 88 |
| About Windows | 89 |
| System Information | 90 |
| Remote Assistance | 91 |
| System restore | 91 |
| Programs | 91 |
| Windows Security Center | 91 |
| System Properties | 91 |
| Internet Options | 91 |
| Internet Protocol Configuration | 91 |
| Performance Monitor | 92 |
| Task Manager | 92 |
| Disable UAC | 92 |
| Enable UAC | 92 |
| Command Prompt | 92 |
| Registry Editor | 93 |
| Employing the 3 Gig Switch | 93 |
| Setting the 3 Gig Switch in XP | 93 |
| Setting the 3 Gig Switch in Vista | 94 |
| Performing System Maintenance | 94 |
| Doing and not doing | 94 |
| Maintaining drivers | 95 |
| Cleaning Windows | 96 |
| Using antivirus software | 97 |
| Running disk checking and defragmenting | 98 |
| Microsoft Update | 98 |
| Using the SolidWorks Network Monitor for Distributed Computing | 98 |
| Setting up the Network Monitor | 98 |
| Running tasks on the Network Monitor | 100 |
| Summary | 102 |

Chapter 5: Training Users. 103

| | |
|---|-----|
| Establishing Training and Evaluation Criteria | 104 |
| Using outside trainers | 105 |
| Training in-house | 106 |
| Using resellers for free in-house training | 106 |
| Using books as learning materials | 106 |
| Training on DVDs | 107 |
| Setting up CAD Administrator led training | 107 |
| Mentoring Power Users | 108 |
| Recognizing the type | 109 |
| Mentoring your protégé | 109 |

| | |
|--|-----|
| Identifying Sources of Learning..... | 109 |
| Using online forums..... | 110 |
| Getting the most out of the forums..... | 110 |
| Addressing confidentiality | 111 |
| Getting value from live user groups..... | 111 |
| Summary | 112 |

Part II: Installing SolidWorks 113

Chapter 6: Installing the Software 115

| | |
|--|-----|
| Selecting an Installation Method | 116 |
| Introducing the SolidWorks Installation Manager | 117 |
| Installing SolidWorks from downloaded data..... | 117 |
| Installing SolidWorks manually..... | 119 |
| Installing using the SWIM from downloaded data | 121 |
| Installing from the DVD | 128 |
| Installing from an Administrative Image..... | 129 |
| Welcome to SWIM | 130 |
| Using the Administrative Image Option Editor | 131 |
| Deploying and installing from the Administrative Image | 136 |
| Installing an Administrative Image from the command line | 138 |
| Using Microsoft Active Directory | 138 |
| Installing Multiple Versions | 139 |
| Installing the SolidNetWork License Manager..... | 140 |
| Uninstalling SolidWorks Software..... | 141 |
| Summary | 143 |

Chapter 7: Licensing..... 145

| | |
|---|-----|
| Selecting the Version of SolidWorks | 146 |
| Understanding your CAD needs..... | 147 |
| Matching needs to capabilities..... | 148 |
| Tracking Serial Numbers..... | 148 |
| Identifying types of serial numbers..... | 149 |
| Using the Customer Portal to manage assets..... | 149 |
| Activating and Transferring Licenses..... | 150 |
| Activating a license..... | 150 |
| Working in spite of an activation error | 152 |
| Identifying where licenses are used..... | 152 |
| Troubleshooting license problems | 153 |
| Administering floating licenses..... | 153 |
| Using license management software..... | 153 |
| Using dongles..... | 154 |
| Installing Floating Licenses | 154 |

Contents

| | |
|--|------------|
| Developing a Home Use License Policy | 155 |
| Weighing advantages and disadvantages of home use | 155 |
| Identifying who is eligible | 155 |
| Obtaining home use licenses for SNL users | 156 |
| Summary | 156 |
| Chapter 8: Standardizing Settings | 157 |
| Standardizing System Options | 159 |
| Identifying the need for standardization of settings | 162 |
| Selecting settings to standardize | 163 |
| Standardizing colors | 163 |
| Standardizing Sketch settings | 164 |
| Standardizing Performance settings | 165 |
| Controlling External References | 166 |
| Using Default Templates | 170 |
| Specifying file locations | 171 |
| Collaborating | 172 |
| Standardizing Modeling Practice | 173 |
| Standardizing for productivity | 173 |
| Establishing a best practice modeling standard | 173 |
| Relating the first feature to the part origin | 174 |
| Creating symmetry | 174 |
| Making relations to stable references | 174 |
| Putting fillets at the bottom of the tree | 175 |
| Using Verification on Rebuild | 175 |
| Fixing rebuild errors | 175 |
| Modeling detail | 175 |
| Creating simplified configurations | 175 |
| Standardizing assembly modeling | 176 |
| Using subassemblies | 176 |
| Limiting flexible subassemblies | 176 |
| Mating to in-context features | 176 |
| Limiting in-context modeling | 176 |
| Summary | 176 |
| Chapter 9: Maximizing Your Use of Templates. | 177 |
| Creating Part Templates | 180 |
| Identifying the most important settings | 180 |
| Drafting Standard | 181 |
| Selecting settings in the Grid/Snap page | 182 |
| Units | 183 |
| Detailing settings | 184 |
| Controlling document specific colors | 185 |
| Image quality controls display accuracy | 185 |
| Controlling plane display parameters | 186 |

| | |
|---|-----|
| Using custom properties..... | 187 |
| Finding hidden settings in part templates..... | 188 |
| Link values and global variables | 188 |
| Adding reference geometry..... | 189 |
| Including sketch settings..... | 189 |
| Set configuration properties in your templates..... | 190 |
| Creating Assembly Templates | 191 |
| Drafting standard | 191 |
| Setting detailing options for assembly templates..... | 191 |
| Establishing image quality settings for assembly templates | 192 |
| Establishing Drawing Templates | 192 |
| Understanding the difference between drawing templates and sheet formats..... | 192 |
| Identifying settings to control with drawing templates | 193 |
| Drafting standard | 193 |
| Setting units for drawing templates | 195 |
| Establishing line font settings for drawing templates..... | 196 |
| Establishing line style settings for drawing templates..... | 196 |
| Line thickness | 197 |
| Controlling image quality | 198 |
| Standardizing sheet metal settings for drawing templates | 198 |
| Putting views on the template | 198 |
| Using predefined views | 198 |
| Projecting predefined views..... | 199 |
| Polishing your templates and formats..... | 199 |
| Establishing Standardized Sheet Formats | 200 |
| Creating a new format | 200 |
| Placing table anchors in the format | 203 |
| Using multiple sheet formats..... | 203 |
| Changing the size or format of a drawing..... | 203 |
| Standardizing Other Types of Templates in SolidWorks | 204 |
| Summary | 204 |

Chapter 10: Implementing Toolbox 207

| | |
|---|-----|
| Understanding Toolbox | 208 |
| Understanding how a default Toolbox installation works..... | 209 |
| The Toolbox default install..... | 210 |
| The Toolbox database..... | 211 |
| The part library | 211 |
| Program files | 211 |
| Managing Toolbox data..... | 212 |
| Underlying basic flaw in Toolbox..... | 212 |
| Configuring a multi-user installation | 212 |
| Combining libraries..... | 213 |

Contents

| | |
|---|------------|
| Administering Toolbox | 214 |
| Installing Toolbox | 214 |
| Installing a shared Toolbox | 214 |
| Creating a shared Toolbox from an existing installation | 216 |
| Upgrading Toolbox | 218 |
| Dealing with version incompatibility in Toolbox | 219 |
| Upgrading and keeping both versions | 219 |
| Configuring Toolbox | 220 |
| Starting Toolbox | 220 |
| Setting up Toolbox | 221 |
| Selecting your hardware — Step 1 | 222 |
| Customizing hardware — Step 2 | 223 |
| Creating parts or configurations | 226 |
| User Settings — Step 3 | 227 |
| Permissions — Step 4 | 228 |
| Smart Fasteners — Step 5 | 229 |
| Adding user parts to the Toolbox library | 230 |
| File management and Toolbox | 231 |
| Alternatives to Toolbox | 232 |
| Summary | 232 |
| Chapter 11: Establishing Libraries. | 233 |
| Creating Library Parts | 234 |
| Adding parts to the Design Library | 234 |
| Using Mate References | 236 |
| Including metadata in libraries | 240 |
| Custom properties and tags | 240 |
| Using Library Features | 241 |
| Accessing SolidWorks Content and 3D Content Central | 244 |
| Examine a Routing library component | 245 |
| Adding to the Weldments library | 246 |
| Using Dissection Results | 247 |
| SolidWorks Search | 248 |
| Windows Desktop Search | 249 |
| Summary | 249 |
| Part III: Maintaining SolidWorks | 251 |
| Chapter 12: Writing a SolidWorks Standard | 253 |
| Defining a SolidWorks Standard | 254 |
| Putting together a 2D standard | 255 |
| Putting together a 3D standard | 256 |
| Enforcing Standards | 261 |
| Summary | 263 |

Chapter 13: Using Custom Properties 265

| | |
|---|-----|
| Establishing the Need for Custom Properties | 266 |
| Using custom properties in the BOM | 266 |
| Using custom properties in the title block | 269 |
| Establishing linked notes | 269 |
| Establishing the Title Block | 272 |
| Building Custom Property Tabs | 274 |
| Creating the Forms | 276 |
| Using Configurations with Custom Properties..... | 277 |
| Summary | 277 |

Chapter 14: Establishing Best Practice Rules. 279

| | |
|--|-----|
| Understanding the Concept of Best Practice | 280 |
| Defining what best practice is..... | 280 |
| Identifying what best practice is not | 281 |
| Adjusting your expectations to reality | 281 |
| Establishing Best Practice for Sketching | 282 |
| Handling sketch relations..... | 282 |
| Choosing between brevity and longevity | 283 |
| Using sketch Contours | 283 |
| Using under defined sketches..... | 284 |
| Choosing between features and sketches | 285 |
| Using layout sketches..... | 286 |
| Layout sketch | 286 |
| Understanding Horizontal Modeling | 286 |
| Determining Best Practice for Features..... | 288 |
| Selecting feature types wisely | 288 |
| Using Dome versus Shape versus Freeform | 289 |
| Selecting Rib feature versus Thin feature | 289 |
| Pattern/mirror features, faces or bodies | 290 |
| Sketch on a plane or on a face | 290 |
| Edit or delete..... | 290 |
| Selecting a draft technique | 290 |
| Deciding to use many or few features | 291 |
| Employing Best Practice in Assemblies..... | 291 |
| Using layout sketches in assemblies..... | 291 |
| Organizing assemblies..... | 292 |
| Using virtual components..... | 293 |
| Reorganizing assemblies with in-context references | 293 |
| Mating with best practice | 294 |
| Mating for position..... | 294 |
| Analyzing degrees of freedom..... | 294 |
| Mating for motion | 295 |
| General mating standards suggestions | 296 |

Contents

| | |
|---|------------|
| Standardizing Best Practice for In-Context Modeling | 297 |
| Understanding how in-context and external references work | 297 |
| Using file management techniques with in-context features | 299 |
| Summary | 300 |
| Chapter 15: Automating Tasks with the API | 301 |
| Recording Macros | 302 |
| Recording a rectangle sketching macro | 302 |
| Recording the macro | 303 |
| Taking advantage of journal files | 305 |
| Replaying the macro | 307 |
| Storing your macro | 308 |
| Connecting to a hotkey | 309 |
| Connecting to a custom toolbar button | 310 |
| Creating a Macro with a User Form | 310 |
| Building the Form | 311 |
| Finding Macro Help | 313 |
| Summary | 314 |
| Chapter 16: Supporting SolidWorks Users | 315 |
| Providing Support through an External Organization | 316 |
| Using the SolidWorks Forums | 322 |
| Using other social media | 323 |
| Going beyond technical support | 323 |
| Providing Your Own Tech Support | 324 |
| Troubleshooting hardware and software | 325 |
| Eliminating PEBCAK errors | 325 |
| Determining software bug or data corruption | 326 |
| Distinguishing between hardware failure and setting problem | 327 |
| Troubleshooting with SolidWorks Rx | 328 |
| Using Problem Capture | 329 |
| Diagnosing with Rx Diagnostics | 330 |
| Troubleshooting your system | 330 |
| Performing regular system maintenance | 332 |
| Understanding Files & Logs | 332 |
| Transferring data to SolidWorks | 333 |
| Looking past the obvious for solutions | 333 |
| Finding Answers to User Questions | 334 |
| Summary | 335 |
| Chapter 17: Collaboration | 337 |
| Sharing Data | 338 |
| Using the settings | 339 |
| Enable multi-user environment | 339 |
| Add shortcut menu items for multi-user environment | 339 |

| | |
|---|------------|
| Making It Work | 342 |
| Using Selective Open | 344 |
| Using Lightweight as a collaboration option..... | 344 |
| Summary | 346 |
| Chapter 18: Managing Change | 347 |
| Evaluating Your Situation Honestly..... | 348 |
| Acting on things you learn | 348 |
| Thinking like a manager..... | 349 |
| Choosing Your Battles | 349 |
| Encouraging Reluctant Users..... | 351 |
| Summary | 352 |
| Chapter 19: Rolling out New Service Packs | 353 |
| Deciding When or If to Update | 354 |
| Calculating return on investment | 355 |
| Making an intuitive decision | 355 |
| Considering Early Visibility testing..... | 356 |
| Installing a Service Pack..... | 356 |
| Applying a service pack to software installed | |
| from DVD or download..... | 357 |
| Applying a service pack to an Administrative Image | 363 |
| Applying service packs in SolidWorks 2010..... | 367 |
| Rolling Back a Service Pack..... | 368 |
| Summary | 369 |
| Chapter 20: Rolling out New Versions | 371 |
| Deciding When to Upgrade..... | 372 |
| Weighing software maintenance costs | 372 |
| Evaluating a new version..... | 374 |
| Upgrading as an option | 375 |
| Taking into account third-party issues..... | 376 |
| Implementing a new version | 376 |
| Going Cold Turkey..... | 376 |
| Phasing in by group..... | 376 |
| Phasing in by project | 377 |
| Failed implementations | 377 |
| Training | 378 |
| Installing | 378 |
| Working with upgraded installations where Toolbox already exists..... | 378 |
| Other libraries | 379 |
| Getting a Fresh Install | 379 |
| Taking care of activations | 379 |
| Uninstalling SolidWorks | 380 |

Contents

| | |
|--------------------------------------|-----|
| Wiping the SolidWorks registry | 381 |
| Using Ccleaner | 383 |
| Summary | 383 |

Part IV: SolidWorks Workgroup PDM **385**

Chapter 21: Introducing SolidWorks Workgroup PDM **387**

| | |
|---|-----|
| You Need Product Data Management..... | 388 |
| PDM is not the solution to everything | 388 |
| Discovering how PDM works | 390 |
| The vault..... | 390 |
| Organizing data into folders with PDM | 391 |
| Implementing Workgroup PDM | 392 |
| Implementing general PDM..... | 393 |
| Mapping the road ahead..... | 393 |
| File naming non-negotiable | 393 |
| Using installation best practice | 394 |
| Attempting unsupported cross-platform installations | 395 |
| Planning for the Workgroup PDM Viewer..... | 395 |
| Preparing for Vault and Client Maintenance..... | 396 |
| Utilizing your backup plan..... | 396 |
| Performing vault maintenance..... | 396 |
| Service pack updates | 396 |
| Major version upgrades | 397 |
| Moving the vault | 397 |
| Summary | 399 |

Chapter 22: Installing and Configuring the Vault **401**

| | |
|--|-----|
| Installing the Workgroup PDM Vault and VaultAdmin Tool | 402 |
| Configuring the Vault | 411 |
| Vault Settings | 412 |
| Vault information | 412 |
| Vault Lock..... | 412 |
| Global settings..... | 412 |
| Administrators..... | 414 |
| On vault startup | 414 |
| Viewing with eDrawings..... | 415 |
| Delete, Rollback, Archive documents..... | 415 |
| Vault Log..... | 415 |
| Search | 415 |
| Revision Scheme | 416 |
| Revision Table..... | 417 |
| Projects | 418 |
| Archive..... | 419 |
| Bulk Check In | 419 |

| | |
|--|------------|
| Lifecycle | 419 |
| Properties | 421 |
| Standard Libraries | 422 |
| Using vaulted libraries | 423 |
| Setting up Toolbox Multiuser Network Installation | 423 |
| IsFastener and IsToolbox properties | 424 |
| Triggers | 424 |
| Summary | 426 |
| Chapter 23: Establishing Permissions and Managing Toolbox | 427 |
| Assigning Users and Groups | 427 |
| Assigning users | 428 |
| Assigning groups | 429 |
| Establishing a Project Folder Scheme | 430 |
| Recognizing limits | 432 |
| Looking to future development | 433 |
| Using Lifecycle | 433 |
| Assigning Permissions | 435 |
| Using Toolbox with Workgroup PDM | 436 |
| Handling configurations in Workgroup PDM | 437 |
| Recognizing Toolbox parts | 437 |
| Using the IsFastener and IsToolbox properties | 439 |
| Establishing Libraries inside the Workgroup PDM Vault | 441 |
| Summary | 443 |
| Chapter 24: Establishing Client Best Practice | 445 |
| Accessing the Workgroup PDM User Interface | 445 |
| Using SolidWorks Explorer | 446 |
| Using the SolidWorks add-in client | 447 |
| Establishing Workgroup PDM Client Options | 449 |
| The Local tab | 449 |
| The References/Where Used tab | 450 |
| The Vault tab | 451 |
| The Command Defaults tab | 452 |
| The Folders tab | 454 |
| The Labels tab | 455 |
| Using the Right Mouse Button menu | 456 |
| Searching in Workgroup PDM | 456 |
| Running reports in Workgroup PDM | 458 |
| Checking in associated documents | 459 |
| Creating attachments | 459 |
| Collaborating with other users | 460 |
| Using Lifecycle settings | 460 |
| Distilling recommended SolidWorks settings | 461 |
| Getting the most from the software through general options | 462 |
| Summary | 463 |

Part V: Appendixes **465**

Appendix A: Online Help Resources 467

| | |
|---|-----|
| Accessing the SolidWorks Customer Portal | 467 |
| Getting Information from SolidWorks User Blogs | 470 |
| Accessing Help on Non-SolidWorks CAD Administration Blogs | 473 |
| Using Social Media Sites to Connect | 473 |
| Twitter | 473 |
| Facebook..... | 474 |
| LinkedIn | 475 |
| Design World..... | 475 |

Appendix B: Sources for Standards 477

| | |
|--|-----|
| General Services Administration | 478 |
| American Society of Mechanical Engineers..... | 478 |
| American National Standards Institute | 479 |
| American Society of Testing and Materials | 479 |
| Deutsches Institut für Normung (DIN) | 479 |
| International Organization for Standardization | 480 |
| Japanese Industrial Standards (JIS)..... | 480 |
| National Institute of Standards and Testing..... | 480 |
| Society of Automotive Engineers International | 480 |
| Concerning ASME or Other Standards..... | 480 |
| Establishing the Need for Standards..... | 481 |

Appendix C: Tools, Options Menu Settings 483

| | |
|---------------------------|-----|
| System Options..... | 483 |
| General Settings | 484 |
| Drawings..... | 489 |
| Display Style..... | 492 |
| Area Hatch/Fill | 492 |
| Colors | 494 |
| Sketch | 496 |
| Relations/Snaps | 499 |
| Display/Selection..... | 501 |
| Performance..... | 505 |
| Assemblies | 508 |
| External References | 509 |
| Default templates | 512 |
| File Locations..... | 513 |
| FeatureManager..... | 514 |
| Spin Box Increments | 517 |
| View..... | 518 |

| | |
|---|-----|
| Backup/Recover..... | 519 |
| Hole Wizard/Toolbox..... | 521 |
| File Explorer | 521 |
| Search | 522 |
| Indexing performance | 525 |
| Dissection..... | 525 |
| Collaboration | 525 |
| Advanced | 526 |
| Document Properties | 527 |
| Drafting standard | 527 |
| Annotations..... | 528 |
| Dimensions | 530 |
| Centerlines/Center marks | 530 |
| DimXpert | 532 |
| Tables..... | 532 |
| View Labels | 534 |
| Virtual Sharps..... | 534 |
| Detailing | 534 |
| Display filter..... | 536 |
| Text scale | 536 |
| Always display text at the same size..... | 536 |
| Display items only in the view in which they are created..... | 536 |
| Use assembly setting for all components..... | 536 |
| Hide dangling dimensions and annotations..... | 537 |
| Grid/Snap..... | 538 |
| Units | 539 |
| Fractions | 540 |
| Line font | 540 |
| Line style..... | 541 |
| Line thickness | 541 |
| Colors | 542 |
| Curvature | 542 |
| Apply same color to wireframe, HLR, and shaded | 542 |
| Ignore feature colors..... | 543 |
| Image quality | 543 |
| Deviation..... | 544 |
| Optimize edge length | 545 |
| Apply to all referenced part documents | 545 |
| Save tessellation with part document..... | 545 |
| Wireframe and high quality HLR/HLV resolution..... | 545 |
| Plane display..... | 545 |
| Sheet metal..... | 545 |

| | |
|--------------------|------------|
| Index | 547 |
|--------------------|------------|

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Introduction

Welcome to the SolidWorks Administration Bible. This book has been written as a desk reference for anyone charged with installing, implementing, and maintaining SolidWorks and Workgroup PDM for yourself or for other users in your company. Whether you are a professional CAD Administrator, a full-time engineer or designer pressed into service as a CAD Administrator, or just an individual user who needs to make the software work for yourself, you will find much to help you in these pages. IT professionals who are not familiar with the complexities of SolidWorks, but need to learn how to administer it will also find help here. The book is full of practical advice, recommendations, and the reasons behind the recommendations. In the spirit of the other SolidWorks Bibles I have written, much of the book comes from personal experience in industry, and not just an academic view of how things ideally *should* work.

About This Book

This book was written during the beta period of SolidWorks 2010, so some changes to the software before its official release may not be reflected here. This book does not touch on topics of direct use, but primarily on topics for administering the software. End user topics should be referred to the SolidWorks 2009 or 2010 Bible. An intermediate SolidWorks user, one who has been through the SolidWorks training for Essentials and Advanced Assemblies, should be able to understand the concepts in the book. You will also need to have some degree of understanding of IT and computer administration issues to be proficient with the topics contained here.

I have included an extensive appendix covering the Tools, Options settings. This offers an in-depth explanation of each option, including System Options and Document Properties. This is information that every CAD Administrator or go-to person should have available at his or her fingertips.

How This Book Is Organized

This book is divided into five parts.

Part I: Preparing for SolidWorks

Before you can be a SolidWorks administrator, you have to get an idea of the scope of the job. SolidWorks administration is difficult work because in addition to knowing your design or engineering job, you must also be a master modeler, a psychologist, mother, friend, disciplinarian, teacher, computer whiz, and financial analyst. The chapters of Part I help you along this path.

Part II: Installing SolidWorks

Installing SolidWorks is much more than just throwing a disk in the computer and clicking Next several times. You will have to deal with issues such as licensing, computer hardware and operating system questions, detail setup or optimization of SolidWorks including implementation of Toolbox and libraries. None of this is something you can get right the first time off the top of your head, and this book has been written from the experience of people who have both succeeded and failed, and you can learn from both experiences.

Part III: Maintaining SolidWorks

This part brings you information on topics such as support questions, installation of service packs, and new versions. Change management is a huge topic that CAD administrators come up against quite often, and here you find a chapter devoted to that topic, meant to help you get through some of the tough changes that you must drive as a CAD administrator.

Part IV: SolidWorks Workgroup PDM

Workgroup PDM simplifies all of the manual file management tasks that commonly trip up users and administrators. Part IV helps you understand what Workgroup PDM does, how to install it, and most importantly, how to configure it to do what you need it to do.

Part V: Appendixes

The appendixes in this book cover a range of ancillary data from where to find help, and how to implement the software to a detailed and complete list of the available options. The Tools@@>Options list in Appendix C is a comprehensive look at the available settings in SolidWorks, something every administrator needs to have handy.

Icons Used in This Book

This book uses a set of icons to point out certain details in the text. While they are relatively self-explanatory, here is what each of these icons indicates:

Best Practice

Best Practice icons point out recommended settings or techniques that are safe in most situations. ■

Caution

Caution icons warn you of potential problems before you make a mistake. ■

Cross-Ref

Cross-Ref icons point out where you can find additional information about a topic elsewhere in the book. ■

Note

Notes highlight useful information that you should take into consideration, or an important point that requires special attention. ■

Tip

Tips provide you with additional advice that makes the software quicker or easier to use. ■

My point of view while writing this book has been that of someone who is actually using the software, not of someone trying to sell ideas, nor of someone trying to make the software look good, or even that of an academic trying to make a beautiful argument. I try to approach the software objectively as a tool, recognizing that complex tools are good at some things and not so good at others. Both kinds of information (good and not-so-good) are useful to the reader. Pointing out negatives in this context should not be construed as criticizing the SolidWorks software, but rather as preparing the reader for real-world use of the software. Any tool this complex is going to have imperfections. I hope that some of my enthusiasm for the software also shows through and is to some extent contagious.

Terminology

CAD is a world full of TLAs (three-letter acronyms), and a book like this can't avoid something as inevitable as that. All of the acronyms have been carefully defined, usually at the first presentation to the reader, and it is assumed that you will know that acronym from there on out. Some are in common usage, such as IT meaning Information Technology (a euphemism for computer professional), or PDM (Product Data Management). I do branch out into some industry terms including ERP (Enterprise Resource Planning) and MRP (Manufacturing Resource Planning). You will come across a healthy dose of computer terminology such as RAM (Random Access Memory) and CPU (central processing unit) or GPU (graphics processing unit), and even some things that might be better left undefined, such as DXF (drawing exchange format) or IGES (international graphical exchange standard).

In one case, I had to invent definitions to make topics make sense. This case was to create a distinction between update and upgrade. Update I take to mean an incremental update, as in a SolidWorks service pack. Upgrade I take to mean the change of a major version, as in the difference between SolidWorks 2009 and 2010. SolidWorks documentation does not make this distinction, they tend to use the terms interchangeably, which causes confusion sometimes.

Contacting the Author

You might want to contact me for some reason. Maybe you found an error in the book, or you have a suggestion about something that you think would improve it. It is always good to hear what real users think about the material, whether you like it or thought it could be improved.

The best way to contact me is either through e-mail or through my blog. My e-mail address is matt@designstuff.com. You will find my blog at <http://designstuff.com/blog>. On the blog, you can leave comments and read other things I have written about the SolidWorks software, CAD, and engineering or computer topics in general.

If you want to contact me for commercial help with a SolidWorks implementation or modeling project, the e-mail address above is the best place to start that type of conversation.

Thank you very much for buying and reading this book. I hope the ideas and information within its pages help you accomplish your professional goals.

Part I

Preparing for SolidWorks

Before you can be a SolidWorks administrator, you have to get an idea of the scope of the job. SolidWorks administration is difficult work because in addition to knowing your design or engineering job, you must also be a master modeler, a psychologist, mother, friend, disciplinarian, teacher, computer whiz, and financial analyst. The chapters of Part I help you along this path.

IN THIS PART

Chapter 1
Defining CAD Administration

Chapter 2
Preparing Information
Technology

Chapter 3
Preparing Document
Management

Chapter 4
Operating System Tools You
Need to Know

Chapter 5
Training Users

Defining CAD Administration

This book is aimed primarily at helping you prioritize the necessary tasks to run SolidWorks efficiently, and set up multiple computers at your company. The intended audience is power users acting as CAD Administrators, professional CAD Administrators, and Information Technology (IT) professionals trying to perform the duties of a CAD Administrator. Even those of you who are AutoCAD Administrators and need to do a SolidWorks installation should be able to get something out of the book, although I assume you are already thoroughly familiar with SolidWorks from a user's point of view.

In the preparation for writing this book, I wrestled with the terms CAD Administration and CAD Management. I read other books and blogs on the topic, and talked with many people who actually do the work. This research helped me understand that from one company to another, vast differences exist in how those companies are organized. In particular, there are almost as many different ways to create engineering and design data, flesh it out, and document it in a form that can be manufactured, as there are different companies. Some of this is strictly driven by a company's size: Larger companies have a tendency to allow more specialization, and smaller companies require people to have a broad range of skills.

Some large companies have the equivalent of a traditional drafting department headed by a CAD Manager. In these departments, detailers take directions from engineers and add detail to the design concept all the way down through detailed 2D drawings for individual components and assemblies. These companies may have one group of people creating product documentation and another group of people creating manufacturing documentation. Companies like this are large enough to be able to afford departments that

IN THIS CHAPTER

Defining a CAD administration

Playing the CAD administration role

Defining yourself as an expert user and mentor

Part I: Preparing for SolidWorks

specialize in a particular function. In addition, a dedicated CAD Manager can only exist in these large companies, because the role is entirely a support role, counted as overhead, not a position that actually produces any revenue.

Not everyone is familiar with that kind of work situation. Far more common is the smaller engineering department lead by an Engineering Manager, where the engineers do most of their own CAD work, and coincidentally most of their own CAD Administration work. In these groups, everyone has to have a wide range of skills and may take a design from concept through to the shipping dock.

I am more familiar with the latter situation, but I will try to write topics relevant to all-sized companies. This book assumes that the CAD Administrator is not a people manager but a technology manager, is still doing billable project work, and that he or she is essentially a SolidWorks power user with extra duties that his or her management probably does not understand. While this scenario doesn't fit all potential readers, I believe it covers the majority.

If you are a full-time professional CAD Administrator, you will find all the technical details and suggestions that you need to keep SolidWorks running useful, along with best practice suggestions to ensure your users create safe data that can be reused easily by others.

CAD Administration is often a thankless job because you have all the responsibility and too often none of the authority. The only way you know you are doing a great job is when it looks like your job is not needed. This makes it tough sometimes for upper management to see the need for a CAD Administrator, although management does not usually need much prodding to remember that they do not understand CAD technology well enough to run it themselves, and that CAD is too complex and too critical to be trusted to run itself.

Often in writing this book, I might sound as though I am talking to CAD users in general; what I have to say might be applied and actually work. However, I'm talking to SolidWorks users specifically. Mixed CAD environments surely exist, but I assume SolidWorks is the predominant software, at least for the reader.

Introducing CAD Administration

CAD Administration means different things to different people. It can mean an entire range of potential tasks and responsibilities from creating documentation to replacing paper rolls in engineering plotters. It is often combined with other unrelated tasks, and the honorary title may simply be assigned to the employee with the most aptitude. It is definitely an overhead job (does not actually produce anything you can sell), and one that management tends to not understand very well.

The CAD Administrator can accumulate tasks that don't fall into anyone else's job description. For example, when your company starts to get into Finite Element Analysis (FEA), who is responsible for installing and maintaining that? What happens when your documentation system is overwhelmed by additional electronic data and the relationships between all the files, and you realize that the vault room with the drawers and cabinets is not going to work as a document control

system any more? Moreover, when the documentation data needs to be electronically linked to the MRP (Manufacturing Resource Planning) or ERP (Enterprise Resource Planning) system, you can be sure it is going to become an interdepartmental task. Product Data Management (PDM) or Product Lifecycle Management (PLM) is a huge responsibility on its own, and again, IT is not always the right choice to manage a system like that, so it often is dropped into the lap of the ad hoc CAD Administrator. By the time you accumulate the titles of CAD Administrator and Documentation Manager, the combined work can be a full-time position.

Typically, the people who find themselves in CAD Administration are those who show a combination of passion and aptitude for CAD along with a strong desire to help people. It seems odd to people who are not CAD users, but a certain percentage of people who use CAD professionally are completely caught up in it as if it were a hobby. Management, of course, loves this brand of over-achiever, because these people will fight to keep the job that overworks them, and management won't have to pay them more money. Of course, this is an exaggeration, but it is hauntingly descriptive of many scenarios I have seen. It's good to enjoy your job, and from management's point of view, it's great to have employees who love what they do.

CAD Administration is one of those positions in a company that always creates itself, even if management is completely oblivious to the need. CAD users are a fraternity of employees with specialized, advanced skills, entrusted with the company's design data, using expensive software and hardware. You like to help one another, and like challenges. The CAD Administrator is the one who rises to the top of that fraternity, and gets the position either by appointment or by default. Many SolidWorks users probably actually are CAD Administrators without even knowing it.

Defining CAD Administration

For the purposes of this book, I am defining CAD Administration as broadly as possible. You can be a CAD Administrator without having all the following responsibilities, but you should at least be familiar with all of them. This is how I define CAD Administration:

- Headed by an expert CAD user
- Standardizes design document deliverables
- Establishes best practice guidelines for CAD users
- Efficiently installs, configures, and maintains CAD/design software
- Evaluates updates to existing software and new design, analysis, or documentation software
- Implements new design or analysis software
- Trains users in new or improved software and any standards that exist
- Provides support to users in the use of design software and standards
- Acts as an intermediate between IT and CAD users as well as between management and CAD users
- Maintains electronic product documentation and revisions
- Very often involved directly in PDM

Part I: Preparing for SolidWorks

CAD software really is more complex than office applications. It is more demanding on your hardware, and the software is often very complicated and expensive. Support for CAD is difficult and expensive, more so than anything else except maybe programming. CAD users must be more computer savvy than the average spreadsheet user, and they must have more control over their local computers than people who check e-mail and surf the Web.

CAD software is also far more complex to install and set up, on par with complex accounting systems. This requires more from the local user's hardware, more from the network on which the users reside, and more from the CAD users themselves. When you have an IT department, the CAD Administrator has to be able to work in harmony with IT to fulfill the needs of all the CAD users.

Often IT departments try to group CAD users together with all other types of technical computer users in the company. To a CAD user, SolidWorks and Photoshop don't look anything alike, but to an IT person, where else would you put Photoshop? It's too graphical to fall under standard office applications. Where does it stop? The programmers use special software for version control called Source Safe. That is complex file management with special relationships between the files, which to an IT professional sounds a lot like the situation with CAD files. Should the CAD Administrator also be responsible for all the complex technical software at the company including Photoshop and Source Safe? Different companies answer that question differently. It might not make sense to have someone in the graphics department and someone in programming who has special IT access handle it, or does it?

Then add other gray areas to the mix, such as circuit board design software. It looks an awful lot like CAD, even to a CAD user. Now add a second CAD system on top of SolidWorks, such as AutoCAD. If your CAD Administrator comes from a 3D background and is asked to administer the 2D installation, that's one thing. Asking a 2D Administrator to cover a 3D CAD package often involves a hefty learning curve. If you find yourself in the latter situation, hold on tight: it may be an exhausting journey, but you will come through on the other side.

You may have other mixed CAD environments such as SolidWorks and Rhino or SolidWorks and Pro/ENGINEER. In the case of mixed environments, the CAD Administrator should be most proficient in the software with the most usefulness in the future. A SolidWorks Administrator may need assistance establishing best practice settings for a package like Pro/ENGINEER. Mixed CAD environments are a fact of life, and one that is not likely to change any time soon.

For all these reasons, the CAD Administrator needs to be both a CAD expert, and have enough access to IT to make necessary changes, as well as have the trust of management. He or she also has to be flexible enough to handle mixed CAD environments and other similar challenges, not just strictly CAD applications. CAD Administration as a separate, stand-alone job is a difficult position to maintain. If you are not using the CAD software on a daily or weekly basis, it is impossible for you to be in touch with the needs of the users. You cannot maintain the credibility and influence of the expert if you are not maintaining your skills as much as those around you are.

Comparing SolidWorks Administration to AutoCAD Administration

To be fair, SolidWorks and AutoCAD are often found in the same offices. Sometimes it is because the organization has not fully made the 2D to 3D transition; other times, it is because there is still a need for both kinds of tools. On the other hand, using both programs may be to maintain legacy 2D drawings that don't make sense to update to 3D, or because the company has some drawings that are better done in 2D.

I won't get involved in the religious argument of 2D versus 3D, but I do believe not all types of design documentation can be done in 2D. In the same way, not all types of products require 2D. If you are creating electrical schematics, 2D is the only way to go; there is no role for 3D. If you are designing complex plastic parts, 3D is the only way to go, with limited roles for anything but 3D.

Comparing project physical scale

As a design package, SolidWorks is typically used for projects around product manufacturing. The products could be built for commercial, industrial, or consumer use. This means that SolidWorks is typically used to design things that fit in your hand, fit on your desk, or might even fit in a construction site. In fact, SolidWorks parts have a hard-coded limitation that you cannot model anything outside of a 1000-meter cube centered on the part origin. This doesn't seem to be much of a limitation, but if you start to model civil engineering projects or architectural projects in SolidWorks, you could easily run up against this size limitation.

One of the most widely used applications of AutoCAD is architecture. In architecture, you are typically designing something much larger than what you design in SolidWorks. So, one of the differences between SolidWorks and AutoCAD might be immense size difference. If you are designing a building, it is likely that you have a team of people for a significant period of time. If you are designing a toaster, it will take fewer people less time to complete.

When you talk about standards in AutoCAD, you are talking about things such as line weights, layer colors and names, the drawing border used, and the use of hidden or frozen layers. When you talk about standards in SolidWorks, what does it mean? Certainly you could have a similar discussion about SolidWorks drawings, with line weights, and possibly layers (although SolidWorks does not rely on layers to the same extent that AutoCAD does), but what kind of a standard would you write for a 3D document?

I answer questions such as this one in later chapters of this book. Standardization for 3D data may seem like a difficult thing to define at this point, but you will understand a good definition or at least a good range of possible definitions before you are through.

Managing file data

Further comparison between SolidWorks and AutoCAD can be made in the area of file management. AutoCAD typically has a one file per paper drawing ratio. SolidWorks typically has at least

Part I: Preparing for SolidWorks

two files per paper drawing (if paper or even 2D electronic drawings are used at all), and relationships between these documents must be maintained through revisions, name changes, folder changes, and all other types of changes. Therefore, SolidWorks adds a significant burden in terms of file management that you may not be accustomed to if you have previously only managed AutoCAD or other 2D data.

Is that something you should find frightening? No, not as long as you know what you are doing before trying to manage the interrelated 3D data. The point is you have to follow rules to manage the SolidWorks file data, but the rules do exist, and you can do this kind of work successfully.

I have heard people use the complexity of 3D documentation associatively as a reason to stay with 2D documentation processes. Unfortunately, CAD Administrator, you will encounter many emotions from users when you make changes to the way they work. Fear of the unknown tends to be one of them, and it usually turns out to be a fully irrational fear. Yes, handling data from a parametric modeler like SolidWorks is complex, and yes, you can seriously screw it up if you don't know what you are doing, but people learn how to handle this type of data effectively every day, and there is no reason why you can't learn to do it too.

Process-based

One of the most important differences between a parametric modeler like SolidWorks and a 2D drafting program like AutoCAD is that modeling work in SolidWorks tends to be very process-based. It matters what the order of operations is. This difference is the learning curve that drafters who move from AutoCAD to SolidWorks must deal with, and certainly, what the CAD Administrator must take into account when training new users, and especially when establishing new standards for SolidWorks use.

Playing the Part: More CAD or Administration?

The role of the CAD Administrator is a two-part role. As the name suggests, the CAD Administrator must be both a CAD power user and an administrator of the necessary housekeeping to keep the technology and the people running.

CAD is the common geometrical language for manufacturing. Administration is the necessary work that has to be done for CAD to give consistent results. Administering a CAD system, in particular SolidWorks, cannot be done by someone who does not have a thorough understanding of how the CAD system works, and in particular the standards used at your company. This is why the CAD Administrator is almost always someone who has worked his or her way up through the ranks of being a CAD user into a leadership role.

Which skill is most important, CAD or administration? To be most successful, you need more CAD knowledge than administrative knowledge, but as I mentioned earlier, in this role you must have both.

Distinguishing between CAD and design

In most of this book, I talk about CAD as if it is a stand-alone function within your organization, and it might be. However, it probably isn't. Larger companies with a traditional organizational strategy are more likely to have a specific CAD department, where drafters or modelers are responsible for taking designs from other sources such as designers or engineers and adding detail, but most companies who use SolidWorks software are smaller organizations where that kind of specialization is not possible.

Many times, the CAD users at a company are also engineers or designers who are responsible for the overall concept and then must add their own detail, without the luxury of having a department of CAD specialists to make it all happen.

Both types of users (concept designers/engineers and detail drafters/modelers) work most efficiently when their CAD environment is installed and set up properly, they are trained on all the tools they need to do their jobs, and CAD standards are established and followed. These different types of users are going to have different needs in system setup.

I'm a firm believer in the idea that design usually happens long before you get to CAD. Design happens in your head, and everything else — napkins, envelopes, white boards and even CAD systems — is used to add detail, and communicate and document the design. SolidWorks is a fantastic tool for adding engineering detail and documenting designs but is a little cumbersome for truly conceptual design.

You may often hear words such as “modeling” and “designing” used interchangeably, but they have distinctly separate meanings. *Design* is the process of creating a new idea, and working with the raw concept of a product, mechanism, or assembly technique. *Modeling* is putting flesh on the bones of the concept, adding enough detail to manufacture individual pieces. Both concept and detail stages are necessary. Sometimes different groups handle the different tasks, and sometimes the same individual handles both. I discuss this again later in this chapter, but the distance separating concept and detail comes down to the size of the organization.

So while I may write generically about CAD as if it is a function separated from other duties, please recognize that I'm only doing that because writing a single book that covers all possible scenarios would otherwise be impossible. You are going to need to superimpose the organizational chart from your company onto the ideas presented here.

Including Information Technology

Information Technology (IT) is an equally important part of the CAD Administrator's job. Often, you will become an honorary member of the IT department. At the very least, you need a good working relationship with the management and members of the IT department because you need access to administrator rights on computers, have access to special areas of the network, perform hardware upgrades, and have the authority to make changes to users computers.

Part I: Preparing for SolidWorks

Every company has a different IT policy, ranging from situations where IT is really just there to offer expert advice and assistance, to situations where IT has absolute control over all hardware and software. The closer your company is to the latter of these two options, the more methodical you will need to be in order to accomplish what needs to be done. At times, it can feel like IT is working against you. Remember that they have responsibilities that are different from yours, and you need their help.

It is a common situation to find IT departments that are highly controlling and resistant to allowing outside forces to have any influence over their schedule, plan for software, or access. If you assume IT will cooperate easily just because you were very polite when asking, you may be in for a rude surprise. Depending on your company's office politics situation, you can usually resolve this by either going to the IT Manager directly or by having your chain of command do it for you. Once management gets involved, and IT is told it must comply to serve the needs of the company, your experience will usually improve. Regardless of how things "should" work, the IT department can make it miserable to get your job done, so when possible, it's a good idea to try to be cooperative and considerate.

Finding a balance

The CAD Administrator is often a power user who looks beyond his own workstation and sees how CAD affects the business as a whole, or at least the business outside of his own department. However, if as CAD Administrator you spend too much time in the bigger picture, the real-world CAD skills may become rusty. If your CAD skills diminish too much, your effectiveness as the lead CAD user can decrease.

The way you become the expert is through using the software. The best way to maintain your skills is to keep using the software to do project work. Since your role as expert depends on your technical knowledge, it makes sense that if your knowledge fades over time, so does your influence as CAD Administrator.

For this reason, along with several others, I believe that the best CAD Administrator is a hands-on, practicing CAD Administrator. In small companies, this is a necessity, but in larger companies CAD Administrators can be prone to lapsing in their skills. In this case, it is best if the CAD Administrator can keep in practice by answering as many questions as possible instead of subcontracting the support to an outside organization. The same can be said for training. CAD Administrators that have outside groups handle all support and training and aren't involved in project work anymore are prone to becoming figureheads, decreasing in effectiveness with each passing release of the software.

The main point is to remember that to maintain your skills you have to get your hands dirty from time to time. Many CAD Administrators I know find it enjoyable to get back into the software again. In addition to keeping your skills up, it can also be a safeguard against burnout.

Defining Your Role

Many different scenarios can exist for CAD Administrators. Some CAD Administrators have CAD operators who report to them, some do project work themselves. Your company's needs and your personal strengths will likely play the major roles in defining what specific tasks you will take on.

If you are establishing the CAD Administrator position in a company where there has previously been no such thing, you have the unique position of being able to define your role. Remember that other people are going to try to offload some of their work onto you, and may try to write a little bit into your job description that wouldn't otherwise be there. If the position is new, you should establish the boundaries as quickly as you can so that other people do not establish them for you.

If you are stepping into a role that existed before you arrived, your work is both easier and more difficult. It is easier because the role is defined for you. It is more difficult because if you feel that the role should be adjusted in some way, that adjustment may be far more difficult to accomplish.

Establishing yourself as an expert user

As a CAD Administrator, you need to be an expert user for a couple of reasons. First, if you are going to be writing standards and establishing best practice, you can't just be someone with an abundance of self confidence; you must actually have solid experience with both the finished product (meaning the drawings and other documentation your company produces) and the tools used to create that documentation (SolidWorks and other design-related software). I have often seen companies push new users, right out of college, into CAD Administrator positions. When they do this, it is usually because management is only considering the tedious software management side of the duties. I believe they are not considering the standards, the enforcement, the training, and other roles that require an experienced and established CAD Administrator. New users are not typically skilled in all these areas but may acquire the skills over time.

Something needs to be said here that may be difficult to hear. Just like being a professional athlete, not everyone is cut out for the job. The most successful CAD Administrators are those who were born with the natural talent of curiosity to keep learning; the desire to continue supporting and training users; the software interface gene that somehow lets you know instinctively how to use software that you have never seen before; and the geometry gene, which enables you to visualize 2D and 3D geometry instinctively as the result of any process you could name.

You could establish yourself as an expert just by sheer hard work and concentration. But when it is instinctive for you to rise to the top, others will take notice immediately, and there won't be any question about it.

Becoming a CAD mentor

There is at least one big difference between a leader and a manager. The difference is a little philosophical, and usually the difference in pay does not reflect which kind of person is more valuable to the company. Think of a manager as "push" and a leader as "pull."

Part I: Preparing for SolidWorks

Being a leader versus manager

Managers are typically less passionate about their work, and they may appear to have more generalized skills. They may be more oriented toward organization, or people skills, they may seem to embrace the corporate entity, and may seem gifted at coercing people to do something whether the person wants to do it or not. The manager *pushes* from behind.

An analogy for this might be a pioneer of the old west, going west with a couple of oxen and a wagon. The pioneer sits on the wagon and cracks the whip over the head of the oxen. The pioneer himself might be weak and frail, but the ability to crack the whip makes the heavy cart move forward. Managers lack CAD skills, but they can make those who have them produce.

Leaders, on the other hand, tend to be passionate or even obsessive about their field and are thrust forward by those around them. Employees can resist a manager, but a real leader is someone chosen by peers. A leader cannot be appointed. Leaders within engineering departments are typically very skilled and are selected for the leadership role by people who bring problems in exchange for great answers and thereby establish this person as an authority. The leader *pulls* from out in front.

You can sometimes find people who are false leaders. This is a trap to avoid for sure. Think of a long line of cars on the thruway. Is the person in that car at the front of the line really a leader just because his car is first? No. This is a false leader. The person might look like a leader because of his position, but the truth is he is going so slow, he holds everyone else back. Do not confuse someone's position at a company with his status as a leader.

A true leader is the soldier who runs forward to attack first, then looks back and signals for everyone else to charge. They see that he has done it, and it makes it easier for them to follow. This is why a true leader of CAD users is always an enthusiastic hands-on technical leader, not a sedate non-combatant.

Mentoring through leadership

As an expert or power user, you are already a leader, whether you have the title or are being paid for it or not. The best use of your skills may not actually be doing project work (but the irony here is that if you don't keep doing some actual project work, you may lose your status as an expert), and it is probably not sitting in front of a spreadsheet or schedule being a manager. Your skills may best be used in leading and motivating other users. If you could duplicate yourself, your knowledge of your company's practices, and your skills in the software, your company would be much better off than just having you pounding out models and drawings day after day.

Becoming a CAD mentor is an important part of being a CAD Administrator. Developing the skills of other CAD users, and in particular the potential power users, can be one of your most important contributions to the future of your department.

The first step to becoming a CAD mentor is to find someone to mentor. The most suitable subjects are usually young, ambitious, and already gifted in CAD, process, geometry, or visualization. Sometimes you can find people who find new purpose mid-career, who haven't been sufficiently encouraged by other leaders.

If you lead or attend training sessions with the rest of the CAD users, future power users are easy to spot. They are the ones in the front row who are bored because they went through the whole exercise before the rest of the class is half done. You don't want to waste this kind of training opportunity. If someone works through the exercise first, assign a special exercise that requires some problem solving. Maybe draw a figure on the white board and ask her to find different ways to model the same thing. I used to keep several sample parts in my desk, and would hand them out to people like this, and simply ask them to model the part. This kind of a challenge is a great learning experience for people who are well beyond the rest of the class.

Natural CAD power users often have an instinct with computers, and usually have an above average understanding of manufacturing processes. The best power users have the uncontrollable urge to help people by answering questions. They are also curious, and never stop learning. If you take the time to educate them one on one, they will be your allies and become assets for you.

Developing power users from your existing crop of users requires one-on-one attention. If you give a future power user challenges, they will rise to the challenge. Sometimes it is useful to pair a power user in training with another user who needs some help with basic skills.

Most of all, avoid suppressing enthusiasm or talent. I have seen some CAD Administrators try to cool off users who were too excited about learning. This may be because they were afraid of losing their jobs to younger, brighter, rising stars. On the other hand, if you are seen as responsible for helping someone with gusto develop his potential, you may be training your own replacement so you can be promoted. Don't suppress talent when you find it.

Administering efficiently

As I mentioned earlier, the term CAD Administrator has two words, and one of them is administrator. Administering anything requires a bit of organization, some people skills, and some planning. If you're not a full-time CAD Manager, then most of the actual "management" tasks will be taken care of for you either by your Engineering Manager or by the IT Manager, but there are some things that no one will keep track of for you:

- Subscription renewal dates
- Lists of serial numbers, software levels, and subscription costs
- Rollout dates for service packs and new versions
- Reseller training dates
- Training classes completed by each CAD user
- SolidWorks World dates
- Ink and paper levels in the engineering plotters
- Key VAR contact
- Key SolidWorks contact
- Submitting and tracking all ERs and SPRs
- Chair in-house user group(s)
- Attend and/or determine attendees for local user group(s)

Part I: Preparing for SolidWorks

Make sure to keep a calendar for the dates, and a regular schedule for checking things that IT may have left to you like the engineering plotters. Make sure also to keep management aware of the important dates.

Spending subscription money wisely

Although you may not have direct control of a budget, the software you work with requires a significant budget for the subscription and training. Your managers are counting on you for input as to whether that money is well spent or not. If you have ten licenses of SolidWorks Standard (base package), the annual cost of subscription is \$12,900. The subscription covers technical support from the reseller, all service packs, and new versions that come available during the 12 months of the subscription period.

If your users have been on subscription for several years, and you have a good base of knowledge, the technical support may not be benefitting you at all. Unfortunately, you cannot separate the cost of the reseller technical support from the cost of the service packs and new versions. If you do not believe that the service packs and new versions are benefiting you much, it may be time to support only a few licenses or eliminate subscription altogether.

If you decide to stop paying subscription and then later decide to get back on, SolidWorks charges a \$500 penalty to get you back on. Therefore, if you get off primarily to save money, you may need to stay off for a couple of years for it to add up financially.

I will revisit the topic of licensing and subscription in a later chapter. Here, I have begun to get you thinking about things that you are responsible for in the course of administering the CAD technology at your company.

Cross-Ref

Chapters 19 and 20 discuss the value of subscription services in more detail. ■

Training and efficiency

Efficiency always includes balancing competing interests. Whenever businesses talk about efficiency, they are usually balancing against time or money. I have already talked about balancing software subscription money against the benefits you get from that money. Training is another commodity that you can look at through the efficiency lens. Are you getting your money's worth for the training you are giving your users?

Reseller training has the reputation of being consistent. The materials used have existed for a long time and work well for general SolidWorks training. The main variable is the instructor. Before you spend money on reseller training, it is a good idea to meet the instructor that will lead the class for which you are actually paying. If you don't feel he or she has enough experience, don't be shy

about asking for a trainer you know or one with which you are more comfortable. Simply being a certified trainer or having given training courses for ten years do not guarantee that the trainer will communicate well with your CAD users. Reseller training is expensive, and you deserve to get value for the money you spend.

Another option is to purchase your own training materials. SolidWorks will not sell you the training materials used in the reseller classes, but other training materials are available commercially. Several printed books are available, including the *SolidWorks 2009 Bible* (Wiley Publishing, 2009). Training materials are also available online, which your employees can use at their own convenience without the need for travel arrangements or time off from work.

Be aware as well that no off-the-shelf training program is going to cater to your specific needs. It may contain a lot of information you don't need, and may skip specific practices developed for your company. At some point, you should consider creating company-specific training for special techniques and standards that your users need to know on a daily basis.

Custom training classes to cover these specialty needs can be developed by consultants, by your reseller, or by yourself. Custom training, while more expensive than standard training, may turn out to be a better value for the money, because you know that your users actually need all the material in the class.

Building a team

If you want to have success as a CAD Administrator, one of the first things you need to realize is that you are probably going to need some help. When working with groups, people often resent individual efforts. Therefore, even if you actually can do everything by yourself, it may be better for the entire group if you build a team anyway. A feeling of inclusion often helps people cooperate.

Team building can be a part of mentoring that I mentioned earlier. Teams can be a great way to find or create consensus if you need direction or support. You might need a team for one of several things, such as for a standards review committee, for software evaluation, or as part of a report on the state of CAD at your company to management.

A team that is made of CAD users from multiple departments can have other beneficial effects. If you are building a team to help select a PDM product, getting input from a wide range of people is key. No one person can anticipate all the electronic document control needs for several departments. A team is also an ideal way to help you gage the effectiveness of a SolidWorks implementation.

Another way to use a team is in the form of a user group. SolidWorks user groups have been very successful in drawing in and educating users. If the team is big enough, you might consider it the user group. If it is small enough, it might be the leadership core for the user group.

Summary

Becoming an effective CAD Administrator requires a combination of technical leadership, organizational, and people skills. The most effective CAD Administrators are also top-notch users with a deep understanding of documentation and manufacturing processes. As the CAD Administrator, it is very important for you to maintain your skills in the tools and processes that you are administering. There is no replacement for hands-on experience. If you can provide your own training and support, that will be of great value to your company.

To accomplish your job, you must be willing and able to work with other people in a team setting to help build a feeling of inclusion. Mentoring other users and recognizing power user talent when you see it are both important aspects of building a team of expert go-to users. CAD Administration is a big job that is often under-recognized by management.

Preparing Information Technology

Information Technology (IT) is the secondary force behind successful CAD Administration, the primary being CAD knowledge. If you are not IT savvy yourself, you should have people available to help you understand or make decisions who are. Some companies try to put the IT department in charge of CAD Administration. While this might work, I don't believe it is the best way, because CAD users need more direct control over their systems, especially when changes are made. SolidWorks Administration is not always a precise science. Changes to the software come frequently, and often with the requirement for testing to avoid newly introduced bugs.

Hardware options change so rapidly that it is difficult for most people to stay on top of what is available now and in the near future. Researching hardware for optimal CAD implementation could be a full-time job just by itself. Beyond that, I cannot get very specific in this book about which hardware to buy, because this book may have a life of several years, but the recommendations may not be valid for more than a few weeks.

This book will arm you with the priorities you need to have going into hardware selection and some general standards that have been true for a long time. With this information, you should be able to make an informed decision with either a little help from a hardware-oriented IT professional or some diligent research on your part into the most recent trends in hardware. Don't be afraid to look for hardware reviews from non-CAD users to educate yourself about the hardware that is available when you start to do your search.

IN THIS CHAPTER

Selecting hardware for installation

Assessing network requirements

Putting together a backup plan for data

Working around your antivirus applications

Creating a workable CAD workspace

Selecting Hardware

The hardware that you select for a SolidWorks installation can be one of the most critical choices you make. If you purchase incorrect or incorrectly configured hardware, your users could complain of slow operation or continual crashes for a long time. Lost productivity is not worth saving a few dollars, or even a few hundred dollars on cheap computers or cheap computer components.

Building your own computers

Frequently I run into people who assume that you get a better computer for less money if you just build your own. People who do this successfully have many different approaches to selecting the computer's components. Some of these methods are successful, and some are not. If you are a serious computer hobbyist and have had good success building your own computers, this may be an option for you.

Understanding the associated risks

On the other hand, building your own computer for professional activities carries with it some risks. Aside from the mechanical and electronic aptitude required for assembling delicate components, other factors also come in to play in ordering components that work together. Some of the conflicts I have seen in home-built computers have included Interrupt ReQuest (IRQ) conflicts that were not resolvable; combinations where a motherboard and sound and network cards were not compatible; incorrect type of Random Access Memory (RAM); insufficiently tested RAM; power supplies that did not supply enough power when power-hungry video cards or DVD writers were used; bios versions that were incompatible with particular hardware; cooling or over clockingscenarios that do not work the way you expect them, and so on.

If you are ready and willing (and have the skills, time, and patience) to take on this kind of troubleshooting, you are a good candidate for building your own computer. Be aware that not everyone who thinks she is up to this task is really up to the task. If you have any doubts, it may be best to work with someone who has completed a build successfully already.

However, if your workplace expects new machines to work on delivery, and wants to hold someone responsible for the entire system rather than individual components, you need to buy completed systems from a manufacturer that provides a warranty, along with performing a system burn-in to make sure that components will not give up after a short life span.

Troubleshooting hardware combinations can take a lot of time and resources, especially if it is not something you do all day, every day. Once you get a system built and deployed, you can run into problems where SolidWorks crashes continually, and the first reaction is to blame it on the software, when it might not be the software's fault at all.

Understanding CAD requirements

Building CAD systems as a hobby and building them professionally are two different things. Even if you choose to have the local computer store put one together for you, you might still run into problems. These folks are more likely versed in building gaming rigs or multimedia consumer type computers rather than heavier-duty systems typically required for CAD. CAD, and SolidWorks in particular, puts a heavy load on your computer. The video cards used for SolidWorks tend to use a lot of power. You have much higher throughput on your RAM and hard drives than with typical office applications or Web surfing. You have to have a machine to handle it. Typically, anything that is classified “workstation” class is something that can handle SolidWorks requirements.

I recommend brands such as Dell, HP, Lenovo, Xi, and Boxx for purpose-specific CAD machines. Xi and Boxx, in particular, focus on the CAD market for workstations that have professional grade components and functionality. Some of these brands enable you to get very detailed in your selections, down to the brand of motherboard used, and the manufacturer is responsible for putting it together, testing it, and guaranteeing that it will work.

Using SolidWorks on a Mac computer

I do not recommend using Apple computers for SolidWorks. While some people are successful in making them work through boot options or virtualization, inherently, each of these options is a workaround. Depending on a workaround for mission critical functionality is not prudent. Those who use SolidWorks on Macs are typically Mac enthusiasts who use Macs for everything else but are able to boot up the Macs in Windows to run SolidWorks. The main reasons for recommending against this is that if you use an option like Parallels, you are essentially running two operating systems at the same time, and using twice as many resources; unless you have unlimited resources, your functionality will be limited. The RAM is probably the most notable limiting resource. Video card functionality is also suspect in this operation-operating mode. Processors are probably not affected by this problem as much because Macs tend to overemphasize the need for multicore processors. Typically the higher the number of cores, the lower the clock speed any of the cores is, and SolidWorks typically prefers the opposite: higher clock speed in preference over the number of cores. This is to say that Mac tends to force you into quad-core or dual quad-core processors in their better machines. This is a poor choice for SolidWorks, which runs better on faster dual core processors, which are also less expensive.

Running with an option such as Boot Camp means that you have to boot Windows and OS X separately if you are going to use all the functionality of the computer, which is time-consuming. Also, as of this writing and for a long time leading up to now, good OpenGL video drivers do not exist for professional level graphics cards running under Windows/Boot Camp.

Additionally, SolidWorks Corporation does not support SolidWorks software when it is run on a Mac under Boot Camp or Parallels. If you choose this arrangement for a corporate setting, be aware that it is not the recommended option. If you choose it for yourself as an individual, it is, of course, up to you.

As for SolidWorks ever creating a version of SolidWorks that runs natively on a Mac Operating system, it all comes down to market share. Not just Apple market share in general, but the market share that Apple has within the engineering community. In short, SolidWorks on a Mac is something you are not likely to see in the next several years.

Selecting processors

Whether in the end you choose to build your own machine, custom specify a machine through a reputable builder, or pick a machine off the shelf from a large volume dealer, you still have to make sure that whatever you choose has the correct components to work for your application. The priority that you should give to your component selections should be in approximately this order:

- Processor
- RAM
- Operating system
- Video card
- Hard drive

At the time of this writing, most users selecting processors are faced with the main decision of how many cores to buy — two or four. Other options are available, but these are the most common. Single core processors are not common in new computers at this time. Dual quad cores (eight altogether) are available, but are considered overkill in cores and underpowered in clock speed for any applications other than rendering or Finite Element Analysis (FEA).

SolidWorks does use multiple cores, but not to the extent that you might think. Typical SolidWorks parts are made from a list of instructions that must be executed in the order in which they were made to build the part. This means that you can't separate the instructions between different threads of computations. If the file is a multi-body part, an assembly, or a drawing, those types of data make better use of multicore processors.

Other types of software also make use of multi-core processors, in particular FEA software such as SolidWorks Simulation (formerly Cosmos), or rendering software, such as PhotoView 360.

Generally, the processor recommendation is to get the fastest processor available for standard SolidWorks. If you are doing analysis or rendering, get the most cores available. The fastest clock speed for a dual is faster than the fastest speed for a quad. This is what makes the choice difficult. If you do renderings or analysis only 25 percent of the time, do the extra cores make enough of a difference to be a benefit over the 10 percent faster dual cores?

As you can see, there is a range of activities where the difference probably isn't that big, and it comes down to intuition about what you will do in the future or personal preference. Still, the processor choice is the one that is likely to make the biggest difference in computer speed for most users.

Specifying memory and operating system

Again, at the time of this writing, the options are different from what they will be at the end of the lifespan of this book. The end of the XP operating system's life is near, but many people still prefer it for SolidWorks as well as for other types of computing. XP has come to the point where it is stable, there are plenty of applications available for it, and above all, people are familiar with it.

Also at the time of this writing, Windows 7 is a few months from release to the public. This new operating system has received primarily good reviews, especially in comparison to Windows Vista. Windows 7 uses fewer system resources than Vista, but more than XP. Even before release, several SolidWorks users have made the switch to Windows 7 release candidate software. This combination is stable and about 10 percent faster than SolidWorks on XP, according to my tests.

Note

SolidWorks Corporation announced in the summer of 2009 that SolidWorks 2010 would be the last version of SolidWorks to support Windows XP and the first version to support Windows 7. ■

In comparison to XP, Vista is a resource hog. It takes an additional gigabyte of RAM to run Vista, which can bring you dangerously close to the 3GB limitation of 32-bit operating systems. If you are using 3GB of memory in XP, when upgrading to Vista, you should add an additional gigabyte of memory, which would put you at 4GB. To handle the 4GB fully, you will need to move to a 64-bit operating system. For this set of reasons, 32-bit Vista does not seem to make any sense for use with SolidWorks, being that it is caught between the resource hungry Vista and the 32-bit limitation.

In past years, SolidWorks users allowed their systems to make extensive use of the page file. Page files come into use when the system does not have enough RAM; it uses hard drive space as if it were RAM. The problem is that hard drives are much slower than RAM. Recent years have seen the cost of RAM fall dramatically, so now it is much more reasonable to have enough RAM that your system doesn't have to page constantly to the hard drive to run SolidWorks. In fact, if your computer uses the page file more than the minimum necessary, you should install more RAM. Keep an eye on your system resources. When the amount of memory used approaches the amount of RAM installed, you should get more.

Regardless of what the SolidWorks Web site says about recommendations, I recommend that you run SolidWorks with a minimum of 2GB of RAM, and an average user will probably want 4GB of RAM. If you are doing large jobs in FEA, rendering, or working with large digitized data sets, you may want 8GB to 16GB.

If you run a 32-bit system, you will start to see crashes when more than 2.6GB of memory are in use. This is a hard limitation with all 32-bit systems.

Whatever you do, now, 32-bit Vista doesn't make much sense anymore; if you move to Vista or Windows 7 for that matter, it should be 64-bit. Drivers and applications for both Vista and Windows 7 64 bit exist for most new hardware, and there is fair coverage even for some older hardware. Either way, the future of computing is 64-bit, so if you have a choice, it is best to head in that direction.

Selecting a video card

The video card has been the most controversial selection when setting up a SolidWorks workstation. The one thing that has remained consistent is that for best performance, you want to get a good OpenGL supporting card. In recent years, this has meant cards with nVidia chipsets and, in

Part I: Preparing for SolidWorks

particular, the Quadro line of cards. (Quadro NVS are 2D cards not suitable for SolidWorks use). The one exception to that is the ATI line of FireGL cards.

The price you pay for not getting a supported card is slow operation, a limited number of OpenGL accelerated windows that you can have open at any one time, and the potential for video-related crashes.

In addition to the brand and model of the card, it is also important to get the right driver version. For this reason, SolidWorks has put together a nice Web page that shows which versions of drivers for which cards you can expect to work with a given version of the SolidWorks software. This gets very complicated if you are trying to make an old card last with another version of SolidWorks. The system works well if you buy new hardware every year, but few companies can justify doing that.

On top of that, you cannot fully trust the SolidWorks recommended driver versions to work flawlessly. In the last couple versions of SolidWorks (2008 and 2009), it seems that regardless of how safe you try to play it with your hardware and driver selection, you are going to have to live with video glitches such as black screens, partial screen updates, graphics area freezes, and other similar intermittent video display problems. I have yet to see a fully glitch-free hardware arrangement using recent releases of SolidWorks.

If you cannot use an approved graphics card, you may be forced to use Software OpenGL Mode. This means that instead of the video card hardware accelerating the graphics, all the graphics calculation is done by software using the central processing unit (CPU) rather than graphics processing unit (GPU). This mode is far slower than hardware acceleration and still does not allow for 100 percent correct operation.

Part of the problem with buying video cards is trying to anticipate what is going to happen in the CAD graphics market in the next couple of years. Right now, the CAD graphics market is split. Many CAD applications have chosen to go the Direct3D route. Direct3D is a subset of DirectX, which is the Microsoft graphics standard. It is the primary competitor to OpenGL. Most computer games on the market are using Direct 3D. OpenGL is an open source graphics language that has seen comparatively little development in recent years compared to Direct 3D.

Products such as Inventor have gone with Direct 3D, but Rhino is staying with OpenGL. This is where the earlier discussion about Apple hardware comes back into play. Most Macintosh-using SolidWorks users would like SolidWorks to write a new version of SolidWorks especially for the Mac. Fortunately, the graphics engine used by Mac is OpenGL (DirectX is again a Microsoft product, archenemy of Apple). Therefore, CAD software that is using Direct 3D will probably not be ported to Mac. Anything using OpenGL has one less hurdle.

So what is the short answer? There is no short answer. With the need to look forward, it is hard to say if SolidWorks will go to Direct3D for all the benefits of being mainstream with more modern development, or if they will stay with the huge investment they have made in OpenGL and leave the door open to a Mac version at some point.

Add to all this the Hoops graphics layer, which allows graphic application developers to simply work with the Hoops API, and Hoops deals with OpenGL and Direct3D. SolidWorks uses Hoops, but it is unclear to what extent they have implemented the Hoops toolset. It is hard to know what, if anything, is going to change in the future implementation of graphics in SolidWorks. The size of the investment in RealView tends to make me believe that SolidWorks is going to stick with OpenGL for some time to come.

For the present (meaning late in 2009), the nVidia Quadro cards and the ATI FireGL cards are the safest choices. Expect to pay between \$300 and \$3,000 for cards that are usable for SolidWorks.

You can make two kinds of mistakes when buying graphics cards for SolidWorks — buying cheap junk and buying expensive junk. Remember that the graphics card only has an effect when you are actively rotating your model on the screen with your mouse. Any rendering that is not real time is done with the CPU (this means PhotoWorks and PhotoView 360 are CPU dependent, not GPU dependent). My recommendations for video cards are usually on the inexpensive end, but not the bottom of the line. I believe that the graphics card is fairly overrated in most users' PC selections. Dual SLI arrangements are definitely overkill for SolidWorks. I would even say that if you pay more than \$500 to \$700 for your video card, you are not likely to get much of a productivity return for all that money.

When many people look at video cards, they compare the card's specs, such as frame rate or memory bandwidth, and these are valid. However, don't be lured into assigning too much importance to these numbers. If one card appears to be 20 percent faster than another on the spec list, but you only can apply that 20 percent speed improvement for 10 percent of the total time you spend using SolidWorks, then the more expensive video card only improves your overall speed with SolidWorks by 2 percent. If that 2 percent costs you \$50, then it's not so bad, but if it costs you \$1500, you may want to rethink your video card choice.

Picking a hard drive

When you select a hard drive, you will need to consider a few factors. First, and most important, is capacity. Next is speed. As of this writing, you can get hard drives for desktop units up to 15,000 revolutions per minutes (RPMs). For most laptops, you are looking at a maximum of 7,200 RPM. Solid-state drives are available for laptops, where the main advantages are they are lightweight, have no moving parts, and are relatively fast. They are still prohibitively expensive, up to ten times the cost of similar capacity rotating disks.

Another very important factor is reliability. If a drive fails, you can lose important data. I won't make any brand recommendations, but you can read reports in *Consumer Reports* or search out other Web sources for reliability information. Drives are often given a warranty period, which you can use as a reliability index. Some drives are guaranteed for up to seven years. Drives with longer warranties are generally more expensive, but when compared against the cost of recovering data from a hard drive, the little bit of extra money spent on a better made drive is well worth it.

Part I: Preparing for SolidWorks

On the other hand, if you do not plan to keep a computer for more than three years, then the more reliable drives may be more than you need to pay for. If you are using hard drives for your backup system, you definitely want the extra reliability. There is no such thing as 100 percent reliability. Even data backed up onto DVD can be lost by scratching or mutilating the disk in an accident or a fire. Solid state drives are highly reliable because they have no moving parts.

I like to have two hard drives available inside a computer. The first drive is typically small, between 60GB to 80GB, and is used for the operating system. The second drive is typically much larger, from 500GB to 2TB, and is used for local working data. I keep legacy data stored on an external hard drive and a backup. You will read more on backup scenarios later in this chapter.

Another decision you will need to make is whether to use a RAID (Redundant Array of Independent Disks) configuration. It means that you can have several disk drives set up in a computer that work together as a system in one of several configurations:

- **RAID 0 (striped disks).** Distributes data across several disks to improve speed with no lost capacity, but all data on all disks will be lost if any one disk fails. Although such an array has no actual redundancy, it is customary to call it RAID 0.
- **RAID 1 (mirrored settings/disks).** Duplicates data across every disk in the array, providing full redundancy. Two (or more) disks each store exactly the same data, at the same time, and at all times. Data is not lost as long as one disk survives. Total capacity of the array equals the capacity of the smallest disk in the array. At any given instant, the contents of each disk in the array are identical to that of every other disk in the array.
- **RAID 3.** Data gets striped across multiple disks at the byte level. A single disk is used to store parity information, and the failure of any one disk, including the parity disk, will not cause failure of the array. The major bottleneck in this solution is the single parity disk, which must be accessed every time information is written to the array. RAID 3 requires at least three hard disks: two for storing data and one parity drive. The size of the array is equal to the size of the smallest drive (total number of drives minus 1).
- **RAID 4.** The same as RAID 3, but the data is striped in blocks and not bytes. This improves the performance of reading data.
- **RAID 5 (striped disks with parity).** Combines three or more disks in a way that protects data against loss of any one disk; the storage capacity of the array is reduced by one disk.
- **RAID 6 (striped disks with dual parity; less common).** Can recover from the loss of two disks.
- **RAID 10 (or 1+0).** Uses both striping and mirroring. “01” or “0+1” is sometimes distinguished from “10” or “1+0”: a striped set of mirrored subsets and a mirrored set of striped subsets are both valid, but distinct, configurations.

The most popular arrangement in workstations is RAID 0 for disk read and write speed. Unfortunately, this arrangement is more than twice as likely to fail as a single hard drive and has no redundancy or backup protection. If any of the drives in a RAID 0 configuration fails, you lose all the data; because the data exists in small stripes that alternate drives, both drives must be functioning to have all the data.

In servers, RAID 5 is the most popular configuration because it helps with speed (due to striping alternate drives), but it is also mirrored, so the data is more secure than the RAID 0 option, but it also requires four drives to set up, two to stripe and then two to mirror the striped drives.

Selecting monitors

Monitor setup is a very important part of the working environment and interface, not just for SolidWorks, but also for all the other productivity tools you use on your computer to do your job. Monitors are coming down in price, so having a large monitor or having multiple large monitors is a real possibility. Starting with the 2009 version, SolidWorks has implemented some tools that begin to make working with multiple monitors more effective. These tools are the detachable PropertyManager and the detachable Task Pane. You can move both of these to a second monitor.

Windows Vista and Windows 7 both allow you to make better use of multiple monitors than Windows XP did. For example, with Vista, you can place the main monitor (the one with the Start menu and Task Bar) on either the left or the right. This can help you get more screen real estate where you need it.

In addition to this, LCD monitors are getting larger and wider than ever before. You can get high-resolution monitors in excess of 30 inches with maximum resolutions of 2560 x 1600 pixels.

Most workstation grade video cards will allow you to connect two monitors. Large widescreen monitors side by side may be bigger than you can actually use. You could opt for an option where you use two screens that are not the same size. For example, I use a wide aspect 24-inch monitor next to a normal aspect 20-inch. They are about the same height, so transitioning open windows from one to the other is not a problem. SolidWorks runs well on a wide screen, and the Internet and e-mail run on the normal aspect screen.

In addition to traditional flat panel displays, you can also use a pen tablet with SolidWorks. These can range in size from a 3x5-inch input-only pen tablet to a 21-inch display/pen tablet combination. SolidWorks is already well suited to pen input for certain types of work, but SolidWorks 2010 has included a couple new features for touch and multi-touch systems. I have included more information on this topic in the next section on laptops, but remember that pen tablets are also available for desktop machines as well.

Selecting a laptop

Selecting a laptop for SolidWorks use can be a difficult task, and depending on your requirements, the result can get pricy. The range can go from models with rock bottom requirements used to display simply CAD models to full-on desktop replacements in a portable chassis including a display. This decision definitely requires a compromise between price, power, and portability.

Selecting the processor

With a laptop, the decisions you have to make are the same as with the desktops, but the processor choice is more polarizing. You can get laptops in dual or quad-core arrangements, but the processor

Part I: Preparing for SolidWorks

speeds are lower than the desktop processors, which mean you are less likely to select a quad-core laptop, unless you spend even more of your time rendering or with FEA.

The reason for laptops to have slower processors generally has to do with power consumption and heat dissipation. Faster processors usually require more power, which generates more heat. Heat is a problem with laptops. It makes users holding the laptop uncomfortable, either from the temperature of the chassis or the hot air blowing out of it. Because one of the requirements for most laptops is that they should be small, you can't build a big chassis that enables you to blow a lot of air through it to cool it off more quickly. Of course, this is even truer with performance laptops with heavy power-consuming components like CPUs and video cards.

Deciding on a video card

Choosing a video card is probably the biggest decision in a laptop computer. It is a bigger decision than in a desktop because the choices for laptop video cards that will adequately run SolidWorks are very limited. You want to avoid the computers with the Intel GMA (Graphics Media Accelerator) graphics. These will run in Software OpenGL mode, are prone to not showing sections of the interface, and certainly will not run Real View graphics. Transparent displays often turn to "screen door" quality, and the detail decimation routines regularly make small curved objects into boxes.

For anything other than casual use, you also want to avoid the Radeon and GeForce cards. As with the desktop units, Quadro and FireGL cards are the ones you want to focus on. This alone will severely limit the choices available to you. You will probably not be able to walk into an electronics or computer store and just buy something like this off the shelf. You will likely have to order it.

Considering power

Power can mean several things in a computer context, but in this case, I'm referring to electrical power: battery power and power from the cord. Both of these sources play an important part in the usability and portability of the system. The electrical power has a direct effect on the processing power, which you need to consider as you use the laptop in various situations.

Processors tend to slow down when the cord is unplugged because they are trying to conserve limited battery power. One of the most important accessories that you can get with a laptop for SolidWorks use is a spare or larger battery. CAD video cards and the typically larger displays burn through battery charge very quickly.

For this reason, if you are leaning heavily on the compute power of the laptop, you want to run it plugged in whenever possible. You may want to have a spare power cord or an adapter for use in the car or airplane. You can run some computers using the cigarette lighters in cars and special outlets in planes. The main limitation with this is wattage. Some of the extremely powerful laptops that can run SolidWorks can have a power requirement of around 90 Watts. Car cigarette lighter outlets might not be able to handle this kind of load.

Along with this large power requirement is an AC adapter brick that can be very large and heavy. You don't normally think of the adapter when figuring the weight of the computer, but it is a necessary component when you travel with your laptop, so it must be counted.

Be aware that if you have a laptop with, for example, a 17-inch display with a CAD quality graphics card, it is going to require a lot of power, and the extra batteries and power adapter for it are going to be heavy to the point that you may not want to be toting them around regularly. Figure 2.1 shows a 17-inch laptop next to a 12-inch laptop, with their respective power supplies.

FIGURE 2.1

The size comparison between a 17-inch laptop and a 12-inch laptop



Choosing RAM

Most laptops at this writing max out at about 4GB of RAM, although that is changing with the increased use of 64-bit systems. The same type of rule applies to laptops with memory as with desktops: the more the better. Memory for a laptop may be more expensive because it may only have room for two slots, meaning that you have to buy two 2GB modules rather than four 1GB modules. The larger modules are disproportionately more expensive.

Picking your operating system

At this time, any new computer that you buy is likely to have Vista on it. In less than a year that will change to Windows 7. Windows 7 is reputed at this time to have better performance than Vista on the same hardware. Laptop users are sensitive to performance issues because that is always what is sacrificed when you put all that hardware into a small and light package.

Part I: Preparing for SolidWorks

Selecting hard drives

Hard drives for laptops are also typically slower than desktop hard drives to conserve power and reduce heat. You may be lucky to get a 10,000 rpm (revolutions per minute) hard drive, but laptop drives are typically 5,400 or 7,200 rpm.

One option for laptops is a solid-state drive. As I mentioned earlier, at this writing, solid-state drives tend to run about ten times the cost for the same capacity spinning disk. The advantage, however, is that they are far lighter, less prone to damage, and faster. As the price of this technology comes down, it will become a more viable option, enabling even CAD laptops to be smaller and lighter.

In some rare laptops (or should I just call them “portables” because they are larger than your lap?), you can actually find computers with RAID options. Computers like this can cost in excess of \$5,000.

Specifying a display

Laptop displays can vary a lot between models. It is probably a good idea to make sure you have seen the particular type of display you are planning on buying. The viewing angle, maximum resolution, glare, and brightness are all factors that you need to consider in addition, of course, to the size of the display.

In many cases, a laptop will be hooked up to an external display such as a projector or an external LCD. If this is the case, the display quality or size may not make much difference. However, if you spend much time working directly on the laptop display, it will make a lot of difference.

For the display connections, you will want to have at least one DVI (Digital Visual Interface) connector. Having a pair is obviously better for dual LCD displays. A VGA (Video Graphics Array) connector is also a good thing if you plan to use the laptop with projectors frequently.

Touch and multi-touch screens

Starting with SolidWorks 2010, you can now make better use of touch and multi-touch screens (tablets). The interface now includes functions for mouse gestures, as shown in Figure 2.2.

FIGURE 2.2

The mouse gesture wheel can be configured to recognize four or eight directional gestures.



This interface is enabled when you drag the RMB slightly. Flicking the cursor (either with the mouse or with the tablet stylus) over a segment of the wheel activates whatever tool is shown in that area. The tool is probably best used when the user is already familiar with the positions of the tools around the wheel and is less effective when you have to pause to look at the icons. The wheel can be customized for four environments: part, assembly, drawing, and sketch.

Note

To customize the commands on the wheel for four or eight segments, choose **Tools ⇄ Customize ⇄ Mouse Gestures** from the menu. ■

Understanding Network Requirements

Most computer programs are written and installed with the assumption that the user will use data on the same computer where the software is installed. CAD software in general and SolidWorks in particular are often collaborative, so users on multiple computers may share files stored on a network location somewhere.

When you work in a multi-user environment, one way or another someone will have to access documents over the network. SolidWorks files can be large. For those of you who are not direct SolidWorks users, perhaps you are an IT person studying up to become a SolidWorks Administrator, SolidWorks file sizes for single part documents can be on average 5MB, and in special cases can grow to a hundreds of megabytes. Assembly documents can be the same, and depending on settings and options used, have the potential to be much bigger. Assemblies may average in the 10MB to 40MB range. Drawings can also be large. It is common for a drawing to be two to four times the size of the parent document.

If you are working on a medium-sized project with, say, 200 parts, you can do the math to figure out how much data may have to be transferred across the network. The entire data set is large. Be aware that all of this data has to travel across your network. This is one of the reasons why Virtual Private Networks (VPNs), which are typically low bandwidth systems, are not generally recommended for serious or frequent SolidWorks work.

Balancing network load and capacity

CAD users place more demand on hardware and software. You may need to show IT that 3D parametric design is not just a graphical editor. It's not Photoshop, Excel, or PowerPoint. You may need to show IT the sizes of sample files. Of course, your production data is going to be far more complex than the sample files, but this demonstrates how much data the applications must manage.

How much data can one user make? How many users will you have? How will this data be stored? How many other people also need to access this data? How will you handle revisions of the data? These are some of the questions IT people will probably ask, and ones to address if they don't. All those questions lead up to the one big question: How big a network investment is this group of users going to require?

Part I: Preparing for SolidWorks

I hope that your IT people will be able to see that CAD data users should be using a server and fast-as-possible network all to themselves. This is both to ensure that the CAD users have plenty of bandwidth but also so that other critical network traffic within the company is not interrupted or delayed by heavy CAD traffic.

Some workplaces are not separated neatly into CAD users and non-CAD users, so your IT people may not be able to separate out a special network in that way. In this case, IT will need to employ network-monitoring utilities to keep network traffic flow balanced. This is probably something they do anyway, but with such high-traffic users sprinkled around the company, it will become even more important to watch for high traffic and bottlenecks.

Understanding local and wide area networks

The LAN is a local area network, typically within a single building that is relatively fast. LAN switches usually operate in the 100 Mbps (megabits per second) or 1 Gbps (gigabits per second) range. You are looking at the LAN when you connect to the server in your office. This type of networking is relatively easy to create and relatively inexpensive.

The WAN is a wide area network and can be much larger. The Internet itself is a large public WAN, but WANs are also used for larger companies. WANs are much slower, and far more expensive.

The VPN (Virtual Private Network) mentioned earlier is a technique for creating a virtual WAN by encrypting data that uses the Internet to connect to a WAN or LAN, usually between an individual user at home or traveling and the central office. This type of connection can be comparatively very slow, but it may also be necessary if you need to access WAN/LAN data but you are not at a location the WAN/LAN services directly.

As a CAD Administrator, you don't want to have to move data through a VPN regularly, but it may be acceptable on a temporary or emergency basis. WANs are frequently used to mirror databases or folder structures, but are still not usually acceptable for working with large sets of files directly across them. In fact, WANs usually try to synchronize folders in different LANs so that users on both sides of the WAN can work with local LAN files.

You should be prepared at this point to talk with your IT people a little about PDM (Product Data Management). One of the goals of PDM is to allow files to be stored in a central location, but when they are used, to move these files to the local computer — not just moving files to the LAN, but the actual computer of the user. By having all the files stored centrally, and a database with software managing the revisions, this complex choreographed dance with large amounts of data works far better than manual file management; multiple individuals either are copying files in and out of a central location or are all working on the files directly from the network folders.

Working on network folders presents immense risks — overwriting data does not have an undo option. In addition, the delete function does not have a recycle bin. When you manage mission critical company design data on a network without these safety net mechanisms, you are taking unnecessary risks. A PDM system eliminates these risks and automates many other error-prone file

management tasks, including revision management, document ownership, access to design documentation for the entire company, and security/traceability. The last section of this book is dedicated to SolidWorks Workgroup PDM.

Building file storage

If you, for whatever reason, must use a manual file storage technique, you will actually wind up using some of the same techniques to make your data searchable as you might use if you switched to a PDM system. Whether you are storing CAD files in folders or in a PDM system, the organization structure can be similar. When you create the storage structure, the structure should make files easier to find. Unfortunately, many storage systems re-create other data that is easy to get another way and adds nothing to organization. For example, if you use part numbers as filenames, I have seen companies organize a folder structure by part number. The part number is redundant information. If you organize the storage structure by product or project, that adds something. Dividing data by the type of document (part, assembly, drawing) doesn't add anything, because that type of information is obvious from the name of the file. Something that isn't in the filename such as manufacturing drawing, inspection drawing, or reference purchasing drawing would add information you might use in a search for the document.

Another mistake IT people often make is to set aside too little space for the data. If you go with a RAID 5 setup, adding more storage is easier than if you have no RAID at all. Say a single drive on the engineering server is mapped on everyone's machine as E:\. If this is a single drive, there is no way to expand it from, for example, 120GB to 500GB. A RAID 5 arrangement can be expanded (if the RAID controller supports that) in size. For this reason, I would consider RAID as a way to create a large, fast storage area and to create data redundancy for backup. For these reasons, it is much better than a single drive with a tape backup. I discuss backup more completely in the next section.

If you are the CAD Administrator for a group of users that includes people who travel frequently and have to take CAD capabilities with them on the road, you may also have to consider the possibility of external, removable drives. When users swap back and forth frequently between laptop and desktop computers, they will often keep data on a USB (Universal Serial Bus) flash drive or external hard drive. Due to the amount of handling, removal, and insertion, these devices are extremely vulnerable to mechanical failure, being lost, or otherwise damaged. While the media for this kind of storage is highly convenient, it is also the most susceptible to failure.

Creating a Backup Plan

Regardless of whom you are and what type of data you create, if you use a computer and you have data that you don't want to lose, you need to have a backup plan. Any IT department earning its paycheck has this in place already or is working on it. However, if you are an independent consultant and you are accumulating many data and need to manage this yourself, you may need to know a few things, or at least be aware of what questions you should be asking.

Selecting a backup mechanism

When backing up data, you are safeguarding it. To do this effectively, you have to identify what sort of a threat you are defending it against. Threats can include anything from a disk drive that just stops spinning to the building burning down or floods — some sort of physical disaster. Some of these failures just require the data to be on a different drive, and some would require that the data be in a physically different location, like a different building or even a different city.

You may also be guarding against a file management failure such as saving an unwanted change, or overwriting or deleting a file. To guard against this type of failure, you have to have several versions of the data backed up. This could be a scheme where data is backed up every other night, and weekly backups are retained in long-term storage.

Another type of threat you have to safeguard against is malicious software such as viruses. Some viruses delete, rename, or overwrite files. Backups and antivirus efforts can be contradictory, because what can happen is that you are actually backing up virus files. Data security is serious business, and you should approach it as if your job depends on it, because it does. The likelihood of such an attack seems low until it actually happens.

Data backup comes in many forms. The simplest is a couple of drives in a computer with some backup software that periodically copies one drive to the other. Depending on what you are doing, this may or may not be sufficient for you. If you go this route, you need to make sure that the two drives are at least physically different drives. Copying data from one partition on a drive to another partition on the same drive doesn't accomplish anything.

You can look at different levels of backup as having different goals. If you need a backup that can retrieve an older version of an existing file (protecting against incorrect versions), the backup mechanism will be more sophisticated than if you simply need to keep copies of static data. Static data backups can be as simple as a stack of recorded DVDs or entire hard drives in a fireproof box.

Establishing a retrieval procedure

A backup plan is only as good as your ability to get data back from it. Unfortunately, the more secure backed up data is, the less convenient it is to retrieve. Data that is copied from one drive to another is obviously easy to retrieve. If the data is stored on physical media such as a tape, DVD, or removable hard drive that has been moved to off-site storage, retrieval is more complex. Retrieving from a cloud storage facility may be the most convenient, although that might depend on the format in which the data is stored and your method of accessing it.

Note

Cloud storage refers to storage on a server accessed through the Internet. Storage of this sort is getting cheaper all the time, and this type of solution may turn out to be a key element in your backup plan because it is secure, inexpensive, and accessible. ■

Data stored inside an archive file such as *.zip or *.rar accessed by FTP (file transfer protocol) requires you to download the entire archive before looking for the file. If you access through VPN, you can look inside the archive without downloading all the data.

When you first establish your backup policy, make sure it includes a retrieval procedure and that you test both the backup and retrieval for all types of data using all of the backup mechanisms you have decided to employ.

Securing data off-site

A complete backup plan will include some off-site storage of long-term storage data. This can get expensive, but your company's data is not replaceable if it is lost. Companies exist that specialize in long-term storage of physical media holding company data.

Off-site storage might suggest cloud storage. This seems to have many advantages, because it satisfies all the requirements of isolating backed up data, without the inconvenience of physically transporting the data to a physical storage unit. The major drawback with this scheme is the bandwidth it takes to transmit gigabytes and gigabytes of backed up data. This bandwidth costs both money and time. Software exists that can help you plan the transmission of backed up data to off-hours, late at night when employees are not using the network bandwidth directly. Existing companies charge for this sort of security service along with the physical maintenance and security of the actual server on which the actual data resides.

Imaging drives

In many situations in which you have to recover lost data, you may also have to reinstall software. If you take an image of a drive, that will include the operating system and all installed software. Common practice with this tool is getting a computer set up with all of the software you need to use on it, and then taking an image of the drive. Imaging software such as Symantec Ghost is easy to set up and use.

When I worked at SolidWorks resellers, we ghosted all training machines. There is no telling what sorts of things a bored student might do to a training box when you aren't watching. After the training class, we reapplied the ghost image, and the computer was back to its original state in about ten minutes, rather than the hours to reinstall an operating system and programs.

Working with Antivirus Applications

If you have been around the SolidWorks program for a number of years, you are probably aware of the recommendations to turn off your antivirus applications before installing SolidWorks. The story goes that the antivirus software cannot tell the difference between legitimately installing software and a virus trying to overwrite system files. Anyone who has been responsible for SolidWorks

Part I: Preparing for SolidWorks

installations has had it beat into them that if you are crashing randomly, it is probably because you installed with your antivirus turned on, and there is something wrong with your SolidWorks installation.

It turns out that some of this paranoia has been well founded, and some of it hasn't. The story evolved over time. It started out that all antivirus software blocked the SolidWorks installation from doing things it needed to do in the course of installing the software. Then it changed to only a couple of the antivirus vendors being at fault. As of this writing, mid- 2009, the recommendation on the SolidWorks system requirements Web site shows that most of the antivirus programs are indeed okay. Figure 2.3 shows the results of the SolidWorks Web site antivirus testing.

FIGURE 2.3
SolidWorks antivirus test results at www.solidworks.com/sw/support/1260_ENU_HTML.htm

| Application | Version | Passed | Notes |
|-----------------------------|---------|--------|----------------|
| Avast! Professional Edition | 4 | ✓ | - |
| AVG Anti-Virus | 7 | ✓ | - |
| CA eTrust AntiVirus | 8.1 | ✓ | - |
| eScan AntiVirus | 9.0.x | ✓ | - |
| Ipswitch AntiVirus | 2008 | ✓ | - |
| Kaspersky AV | 7.0.0.x | ✗ | see SPR 397619 |
| McAfee VirusScan Plus 7 (1) | 7 | ✓ | - |
| Norton AntiVirus 2008 | 2008 | ✓ | - |
| Panda AntiVirus 2008 | 2008 | ✓ | - |
| Sophos AV | 2.0.x | ✗ | Not supported. |
| TrendMicro OfficeScan 8.0 | 8.0 | ✓ | - |
| Windows Live OneCare | 1.5 | ✓ | - |

(1) Only support use of "Normal mode". "Extreme mode" will not allow SolidWorks to be installed.

Benefiting from antivirus software

If you have ever lived through a virus attack on your computer or network, you know the benefit of using antivirus software, especially if you were responsible for keeping the computer or network clean in the first place and cleaning up the mess afterward. Cleaning up from a virus attack can mean reformatting hard drives and reinstalling the entire operating system and all your software.

Some features of antivirus software may be annoying, such as requiring you to confirm when software writes to the registry, or how slow your computer runs when it is scanning files, but that inconvenience pales against the inconvenience of files disappearing or losing control of your own computer. Unless you work in a very sterile environment where you don't have access to the Internet or e-mail, you need to keep the antivirus software running and updated.

Shortcoming in using antivirus software

You should be comfortable with the need for an antivirus, but you also need to know exactly what it costs you so you can make better decisions about how you configure it. If you allow an antivirus to scan all files as you touch them, it is going to scan each and every SolidWorks file. This can cause access problems such as locking down the write access to a SolidWorks file while it is being scanned. You sometimes hear people complain that they are not able to write to files, and overactive antivirus software is a probable culprit. This is one possible source of the “Server Busy” file access error.

Technically, SolidWorks files do allow macro features, which cause an executable file to run when the SolidWorks file is opened. Therefore, it is conceivable that a SolidWorks file could contain a harmful virus. I personally have never seen or heard of an infected SolidWorks file. If you have created all your own SolidWorks data, there is a very slim chance that one could become infected accidentally. If you receive files from an outside source, though, you certainly should scan them. One thing you can do to decrease the amount of time spent scanning files is to add *.sld* files to the ignore list for your antivirus software. Files that you get in from DVD disk, FTP, Internet, or e-mail should be scanned before being mixed with your other data.

From time to time, you do hear about AutoCAD files with an infection of one type or another. Therefore, the concept of malware inside CAD data is not beyond the realm of the possible. If you consider how long these types of files have been available, and how many infections have been reported, the number is very low.

Setting Up a CAD Workspace

Some CAD users also have other responsibilities, such as assembling or troubleshooting physical products. I think they are the lucky ones. To design something using your head and then work with it using your hands is very rewarding. Some CAD users, however, sit in front of the monitor for eight hours a day just doing CAD. When you do one task all the time, you get very good at it, but it also requires that you have a high-quality office environment. If you spend that much time working on a computer, you have to make sure that your workspace doesn't hurt you. Hiring an ergonomics consultant can be expensive, but if you are setting up many workstations, it could be worth it.

A workspace consists of a desk, chair, monitor(s), keyboard, mouse, lighting, and drawer space. Some employees may require extra desk space to lay out large paper drawings, or a bench area for mechanical or electrical assembly work.

Employees need to be provided with adjustable chairs, desks that are functional and sturdy, and options such as keyboard trays, adjustable monitor arms, and lights. The most expensive equipment isn't always the best, but you can guarantee that the least expensive is rarely the best for the employee.

Part I: Preparing for SolidWorks

Typically, an office furniture reseller will have someone on staff to help you with setting up workstations for different purposes. Given the amount of time employees spend in their chairs at their desks, an investment in making them want to be there might be worthwhile.

Understanding ergonomics

Workspaces need to be set up for the inhabitant to be comfortable. If someone is going to sit in a chair and do a job that requires concentration for 40+ hours a week, it should at least be comfortable. Everyone has a different idea of what “comfortable” means. To some it is a fully articulated, professionally designed piece of art, and to others it might be an overstuffed bench. Some people prefer to sit upright, some prefer a slightly laid-back position, and some prefer to change position every few minutes. It’s best to provide a flexible environment because like software interfaces, people interact with office furniture differently.

Other ergonomic concerns might include reducing sharp corners on desks, and providing adjustable footrests and adjustable lighting. CAD users are often particular about lighting. Because they are looking at the lighted screen all day, they often prefer the workspace around them to be somewhat dimly lit, with smaller lights for desktop use. If you have ever been at a company where they have dedicated CAD users, you have probably walked into a room that was dark with many computer screens.

Personalizing a workspace

Employees can get finicky, and it may become difficult to draw the line between accommodating and coddling. Giving each employee a workstation budget and allowing them to spend it as they see fit within certain guidelines, such as selecting approved items from a catalog, is a reasonable approach that enables the employee to feel a certain pride of ownership and personalization without stepping too far out of the corporate design scheme. Recognizing that everyone has different needs and making a gesture at accommodating them can go a long way toward happier and more productive employees.

Summary

Hardware selection is more important than software settings because hardware is more difficult and more expensive to change. You have to make good choices right from the beginning, so if you don’t have the knowledge yourself, you need to make alliances with those who do.

IT people are necessary to make the whole CAD scheme work, but they often have their own agenda. You may need to work through management to gain the cooperation of IT. Still, you need to be careful not to alienate these folks, because they will play a part in your success.

As with anything in business, it’s not just about the technology; it’s also about the people. It’s important to make your people happy and comfortable; and a professional workspace that they feel they have had some hand in can be one of the keys to happier people and higher productivity.

Preparing Document Management

The interrelationships between part, assembly, and drawing documents in SolidWorks need to be managed more carefully than a system of flat files. SolidWorks and all 3D parametric modelers have some special needs when it comes to file management. If you follow some simple rules, you can manage the files successfully.

However, taking many shortcuts with file management may leave you with a set of disassociated documents if you do not understand and follow the document management rules and handle files according to those rules. The better you understand the rules, the more you can bend and even break them, but trying to break rules without a complete understanding of the rules and consequences is risky.

If you or your company is new to SolidWorks, it is important to get your document management process figured out as early as you can. How painful making changes to an existing system will be depends on how entrenched the system is, and how many changes you have to make. You have many important choices before you, and it is not easy to change your decisions later on if you discover a flaw in your reasoning or execution. You have to understand the issues completely right up front and make good decisions right away that you will live with for the life of your document management system. While that statement may be somewhat overblown, and you can actually make changes or adjustments, changes to a system in use are something you should avoid if possible.

IN THIS CHAPTER

Choosing file naming conventions

Organizing SolidWorks documents

Managing files by controlling revisions

Storing revisions by using configurations

Naming Conventions

File naming conventions is a topic that has spawned many fervent arguments. You find strong opinions on all sides of this issue, with the main ones for or against an intelligent part numbering system where portions of the part number signify options in the product.

I won't declare a winner in this feud, but, rather, present the positive and negative arguments for each side. The issue is complex, and there certainly is no clear correct choice. When it comes right down to it, the choice that documentation people face here is how much do you trust your ability to create a system that anticipates all your needs, including needs that don't exist right now?

People who like automatic systems tend to go with the intelligent schemes. People who like to be positive that nothing bad will happen go for the sequential system.

Choosing between intelligent and sequential conventions

Regardless of which system you choose, it is probably prudent to have at least a portion of the part number to be sequential (nonintelligent, random), although I will give you an excellent example of a situation in which that piece of advice was not followed, yet the system is highly successful. The reason for requiring a section of a number to be sequential is to avoid duplication in a system that could possibly create duplicate part numbers for different parts. I do not dispute that there may be some value in intelligent part numbering, but I also think that in any intelligent system, unless things are extraordinarily well defined, there is the possibility for duplicate part numbers.

Advantages of intelligent part numbering

The main advantages of intelligent part numbering are that you can always tell exactly which part you are talking about right from the part number, and that parts that are similar will have part numbers that are similar. Intelligent part numbering appeals to our need for order and our desire to classify things. It's a very organized and thoroughly planned way of doing things.

Intelligent part numbers are often divided into sections, possibly using prefixes or suffixes that are set off from the main number by dashes or dots (see Figure 3.1).

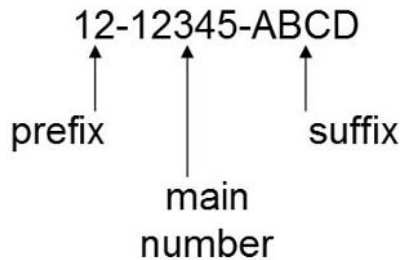
The prefix in this case could stand for a document type such as a manufacturing drawing, an inspection drawing, or assembly instructions, or if the part number signifies a part rather than a drawing, it could stand for some classification of the actual part, such as casting, machined part, circuit board, and so on.

Note

There is an important distinction between using part numbers for parts or documents. With intelligent part numbers, you can tell the difference between numbers intended for drawings of the same part that have different functions and the part itself. ■

FIGURE 3.1

An example of an intelligent part number



The main number can also have significance, and you usually need a key to decipher its meaning. For example, within the prefix 08, which signifies circuit boards, the first two numbers (shown as 12) mean different types of circuit boards, 01 for single layer, 02 for double layer, 03 for ceramic, 04 for FR-4, 06 for flex circuits, and so on. The next three numbers might apply to the project number.

Suffixes can be used or not used as needed. In the case of the circuit board example (refer to Figure 3.1), the suffixes might signify its input or output voltage, types of connectors, or other characteristics of significance where options exist.

Examples of significant part numbers are easy to find. Part numbers for many catalog parts are laid out as intelligent part numbers, with some sort of an identification key for ordering. Catalogs might have many different intelligent schemes, because a single scheme would not be enough to apply to every part in the catalog inventory.

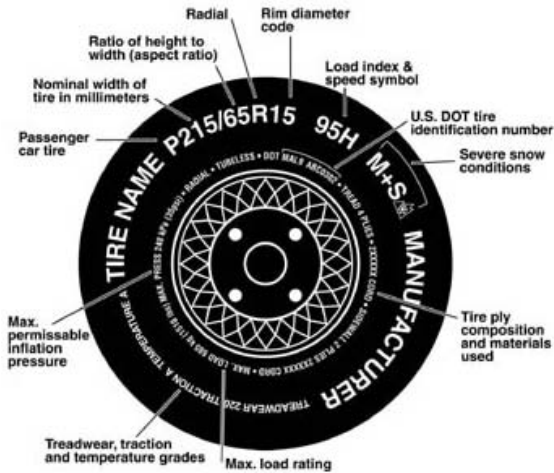
An example of a highly successful, intelligent part numbering system is the one used for automobile tires. In fact, tires have a long-established intelligent part numbering system that works across many vendors around the world. You can go to different tire vendors, give them the part number from a different vendor, and they know exactly what you are talking about. This system has held up for years and works seemingly flawlessly. Each number or letter has significance.

Figure 3.2 shows the classification system used on tires, which serves as part numbers. The number P215/65R15 is recognizable around the world regardless of vendor. Notice also that as successful as it is, this part numbering scheme has at least one glaring error that engineers would never tolerate — mixed units (inches for rim diameter and millimeters for width).

Part I: Preparing for SolidWorks

FIGURE 3.2

Intelligent part numbering system on tires



This image is from the Wikipedia.org entry on Tire Code.

So, intelligent part numbering is really trying to classify the part with the part number. When the system works, it makes identifying parts very easy.

Disadvantages of intelligent part numbering

Because intelligent part numbering is really just building a classification system, the success of an intelligent part numbering system depends heavily on the number of items being classified remaining relatively static and very easy to identify. Unfortunately, changes happen often, and parts are not always easy to identify. Frequent changes sometimes require changes to the part numbering system after an existing system has already been in use for some time. This can cause a lot of confusion, not only in creating the part numbers, but also in interpreting them. Secondly, differences between parts are not always easy to identify, or identify consistently. One person might look at a car and say, “it is red,” while another person could look at it and say “it’s a V-8,” and a third might look at it and say, “It’s a gas guzzler.” Each of these people is correct. If you were to create an intelligent part numbering system for automobiles, you would need variables for each option, which would get out of hand very quickly.

The Dewey Decimal System

Another example of an even older intelligent part numbering system is the Dewey Decimal System (DDS) used to classify books in libraries. In this system, there are ten top-level classes, each broken down into dozens of subdivisions. This number is then followed by a Cutter number, which is composed of a composite of the author’s name and a number from a chart of names, along with

potential entries for year of publication, edition number, copy number, and a number of other things. The Cutter numbers are mainly to avoid duplicates in the classification system.

It is no wonder students find the Dewey Decimal System such a chore to learn, and industry has not been lining up to copy it. While the Dewey Decimal System is still in use, it is a good example of what can go wrong with an intelligent numbering system. It leaves a lot open to interpretation such that a book may have two different DDS numbers from different libraries. The rules are so complex that it's a difficult system to follow.

The system has worked for more than 125 years, but it has gone through 20-plus revisions in that time and retains many faults. However, it does work. This is a common comment with intelligent numbering systems. The systems have many flaws, but they do work. It is up to you to determine if what it buys you (a flawed classification system) is worth the price you pay for it (it makes duplicate part numbers, but doesn't guarantee that two people would assign the same part number to the same item). You must also determine if it is better than the alternative, sequential part numbers, where there is no classification at all other than the order in which numbers have been assigned to the items.

The circuit board example

Going back to the circuit board example, take a look at the problems with that intelligent system. Figure 3.1 shows the part numbering system at use in this example. As a reminder, the prefix 08 signifies circuit boards, the first two numbers (shown as 12) mean different types of circuit boards, 01 for single layer, 02 for double layer, 03 for ceramic, 04 for FR-4, 06 for flex circuits, and so on. The next three numbers might apply to the project number. In this case, the suffixes might signify its input or output voltage, types of connectors, or other characteristics of significance where options exist.

First, what happens if the company designs a new radiator cap for an automobile that has circuitry built into the injection-molded cap. How do you classify this new part? Is it a circuit board, because it uses that design and manufacturing process, or is it an injection molded part, because it also uses that process? On the other hand, what if your company had a classification for inseparable subassembly? It is also that.

The big problem here is that depending on what kinds of parts your company has to deal with, you may be in a situation where change is not predictable; and be clear that the one thing that begins to bring this intelligent part numbering system to its knees is change, especially when parts start to cross classifications.

Another disadvantage is you may have different classification systems for different types of parts. This means that you do not have only one numbering system, but many.

If you make tires, you may be in luck, but other types of manufacturers are not so lucky. The example of the tires shows that even flawed classification systems can be highly successful, if successful equates to being unambiguous and widely used. However, the circuit board example shows that not all products are easy to classify, and not all products will see the same sort of success as tires.

Advantages and disadvantages of sequential part numbers

On the surface, sequential part numbers do not seem to have any advantages whatsoever because they do not give you any information unless you happen to remember a particular part number. Sequential part numbers are usually just the number next in line in a database, or in days gone by, when they would come from a logbook, the next number in the log in the list of entries.

However, sequential numbers do have several advantages. The main one is that there is no way to get duplicate numbers. The idea that each item should have a unique identifier is one of the cardinal rules of document management, and it turns out to be one of the rules that fully intelligent part numbering can easily transgress.

When people make arguments in favor of sequential part numbers (also called *non-intelligent numbers*), another advantage they cite is that there is no data associated with the part number. This is advantageous because you are not duplicating the function of the part number. Is the part number meant to classify the part or simply identify the part? The data, or significance that would be conveyed through the intelligent number, must be attached to the part in some other way. Proponents of sequential numbering claim that the part number is only for identification, and that classification comes through the metadata (non-geometrical text data) attached to the part through a database or, in SolidWorks, through the custom properties functionality.

If you use sequential part numbers, the bottom line is that you have to have some mechanism for relating the description of the article to the part number. This usually means a Product Data Management (PDM), Manufacturing Resource Planning (MRP), or Enterprise Resource Planning (ERP) system. These database-driven systems always have a way to connect metadata to the part number. This enables text searches to find actual human readable text of the metadata rather than cryptic number strings of the sequential part number.

Obviously, this system does not depend on the types of products a company has to document. You can apply sequential numbers to absolutely anything without restrictions. Intelligent part numbers only work for rather simplified classes of products.

A real-life example of a sequential numbering system is the social security number system. It turns out that the number is not completely sequential; it actually has some significance to it, but very few people know the significance of any part of it, so it may as well be simply sequential. This type of number was needed to identify people because names are too easily duplicated. Your name is the equivalent of a descriptive name assigned to a part or document. I discuss descriptive names later in this chapter.

Comparing part numbers, document numbers, and filenames

Part numbers, document numbers, and filenames are sometimes all used in the same way. In this chapter, I'm defining part numbers as numbers that attach to a specific physical part. This is to identify it on the shelf. The same number is used to identify the part electronically. The part number is different from a document or drawing number in that it identifies a document, either paper

or electronic. That document usually has as its subject a physical part with a part number, and some companies use the part and drawing numbers interchangeably. Part and document numbers are often used to identify parts and drawings inside business systems much as MRP or ERP.

A filename is the name given to an electronic file. The electronic file may be a drawing, or a 3D part model. A filename can be either the document or part number, or neither. These three concepts need to be understood separately, even if they are sometimes all used interchangeably. The filename is the single identifier of an electronic file and is used to identify the file inside of a system such as PDM or even just Windows operating system.

In the discussion on part numbers, I didn't really talk about filenames, although I alluded to the idea that numbers that identify documents may or may not be the same as numbers that identify actual physical parts. This may get a little confusing, so stay with me here.

Let's say you have a Bill of Materials (BOM), and a warehouse employee has to put together the list of items from the BOM. The first item on the list is 12345, and it has a description of Transmission Housing Casting. In the warehouse, there is no such part, but when he looks it up on MRP, he finds that 12345 is a drawing of the casting, and there is also a drawing for the casting die and a drawing for the secondary machining operations for the casting, as well as a machining fixture, machining fixture drawing, shipping crate, a shipping crate drawing, a transmission assembly instructions document, and a transmission assembly drawing. Not only does that number bring up all those items, but it also lists part numbers for physical parts, electronic files for the drawings, and the models the drawings were made from.

If you are new to document management, this is something you will need to confront at one point or another. When creating part numbers, you have to be explicit about what that number stands for: a physical part, paper drawing, or electronic document, which may be a drawing, a model, or any of a number of other types of design documentation. What is the warehouse employee supposed to collect for his BOM list? An aluminum casting? A paper drawing? An electronic drawing? A SolidWorks model file?

Creating a hybrid intelligent-sequential system

Instead of adhering to only one method for part numbering, such as either an intelligent part number or a sequential part number, it may be useful to recognize, as I have shown, that each system has advantages. So why not combine advantages, and use a selection of both systems to create a hybrid? If the part number 12345 stands for the Transmission Housing Casting, you may or may not want all documents related to that part to have the same number. A prefix accomplished this task in the circuit board example, but it can also be done with a suffix, as shown here.

- 12345-01. Physical part ready for assembly
- 12345-02. Part specification or drawing
- 12345-03. Secondary operations drawing
- 12345-04. Raw material part (as cast) in need of secondary operations
- 12345-05. 3D CAD model electronic file

Part I: Preparing for SolidWorks

This system is a hybrid between a sequential main number and an intelligent suffix. It has broken down the intelligence to a level where it is controllable and useful. You could use a strictly sequential scheme where it might look like this:

- 12345. Physical part ready for assembly
- 12346. Part specification or drawing
- 12347. Secondary operations drawing
- 12348. Raw material part (as cast) in need of secondary operations
- 12349. 3D CAD model electronic file

This system is also perfectly acceptable, but it does not immediately convey what type of item each part number is. It is easy to see why the hybrid system might be attractive. Taken in smaller doses, the intelligence seems like a good thing to have, but it doesn't seem to be able to handle the top level of part number organization unless you have a very controlled system, such as the one for tires.

You may think this is starting to get rather complicated, because for this one Transmission Housing Casting, you have several types of items to keep track of. You have the physical part in a couple of different states of manufacture, several different 2D electronic drawings, and the 3D models. Is it necessary to keep track of each of these? Yes, it is. You have a manufacturing process, and need to keep track of Work In Progress parts, along with finished parts and completed subassemblies, tested, untested, and so on. You need to think in part numbers and document numbers, and where the parts are in the manufacturing process.

Adding filenames to the mix

In SolidWorks, when you put a part or an assembly onto a drawing, the drawing inherits the name of the model. This doesn't necessarily carry over to document management, but it can if you want it to. In SolidWorks, you can have a part named 12345.SLDPRT along with a drawing called 12345.SLDDRW and an assembly named 12345.SLDASM.

The one cardinal rule in SolidWorks document management is that you can't have duplicate filenames. Your document control process, whether manual through folders in a network drive or automated through a PDM system, needs to reflect this. Windows does not enable you to put two files with the same name into the same folder. PDM systems that allow multiple files with the same names do it because they identify the files through database identifiers, not necessarily because it's a good idea.

The need for unique filenames is the same as the need for unique part or document numbers. If you get away from the idea of intelligent part numbers, then all these names and numbers are used for the exclusive purpose of identifying the file, part, or document. It makes sense that the part or document number should be used as the filename because the filename has the same purpose as the part number — just to identify the file. It doesn't remove the need to distinguish electronic files from physical parts, but it does guarantee that you do not get things confused. I also recommend using some sort of a prefix or suffix to be able to identify quickly different types of numbers.

Using a descriptive naming convention

The discussion about descriptive names versus the more cryptic numerical names for SolidWorks filenames often brings up as heated a debate as the one on sequential versus intelligent part numbers. Your friends don't identify you by your social security number, they use your given name, and it is natural for us to try to do the same sort of thing with part and document numbering or naming systems. By *descriptive names*, I mean giving a human-readable name in plain language to a SolidWorks file. This means assigning a name like "Cover.SLDPRT." This could also apply to more specifically descriptive names such as Motorola Transformer Cast Cover.SLDPRT. You probably see many people with this kind of name for their SolidWorks parts, but I want to discourage you from adopting a convention like this for anything but concept design work. If it has to be documented and managed, you should assign a proper name to it.

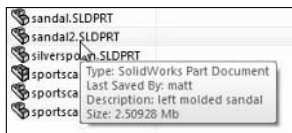
Some people reason that the filename should be human readable text to help you identify the part just by sorting alphabetically and looking for it in Windows Explorer. This seems like a reasonable assertion until you look into the issue a little deeper. The requirement isn't only to be able to identify the part with text, but to be able to identify the part *uniquely* with text. This issue will keep resurfacing as long as you try to use the filename to add information other than a unique identifier for the document.

Using metadata

By now, you probably see the need for unique filenames. The case is not yet complete, however. If you can't put significance in the part number, can't use descriptive plain language names, and are stuck with cryptic sequential numbers or hybrid sequential semi-intelligent numbers for filenames, how will you identify parts aside from memorizing the numbers? The obvious answer is to use metadata. Metadata includes any sort of descriptive text in any category you might choose that is either kept inside the file as custom property data or associated with the file through a database. So if you have a description for the part or document, put it in a field called Description. In SolidWorks documents, this is best done as a custom property. You can search custom property data in Windows Explorer, even if you are only doing manual document management, but it is not as convenient or as powerful as if you are using a true PDM system. Figure 3.3 shows SolidWorks custom property for Description showing up in the tool tip balloon.

FIGURE 3.3

The SolidWorks custom property is shown in Windows Explorer tool tip balloon.



Part I: Preparing for SolidWorks

Metadata possibilities in PDM

The ultimate answer that this line of questioning is leading to is that for almost any SolidWorks user, documents should be managed in a PDM system. PDM does not have to be expensive or difficult to install or maintain, but it can satisfactorily answer all the questions raised in this chapter.

The way PDM addresses potential difficulties is by enforcing unique document names used to identify unique documents, and using metadata to describe or categorize the document. Typical metadata (custom properties) include description, material, finish, supplier, color, customer, voltage, drawn by, approved by, date approved, and so on. Any way you want to classify the part or document where a property has a value can be used to identify the document through search functionality. It is more than just convenient, more than just a reasonable replacement for Windows Explorer searches, and you will find it far more useful as well. In addition to simple searches, you can also create BOM type data from PDM, as well as Where Used reports, or a number of other uses.

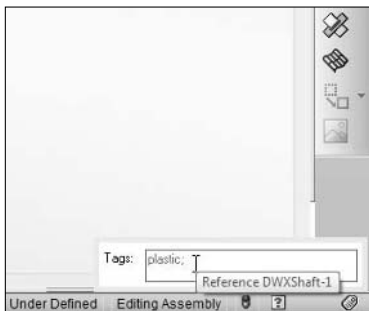
Cross-Ref

Chapters 21 through 24 are dedicated to Workgroup PDM, formerly known as PDMWorks, which is the simple PDM tool included with all but the base level of SolidWorks. Workgroup PDM is easy to install and set up, and handles your SolidWorks and other types of data. ■

Another way to add metadata to SolidWorks parts, assemblies, and drawings is to use the Tags function. You will find the Tags function in the lower-right corner of the SolidWorks window, with the small, yellow icon that looks like a tag, shown in Figure 3.4.

FIGURE 3.4

Applying a SolidWorks tag



Tags can be used to tag specific parts in an assembly or features within a part. If Windows Search has indexed your files, then you can use SolidWorks Explorer and SolidWorks Search to search for

properties as well as tags. You can think of tags as being like properties, except the tag only has the value without having the property. You could also think of tags as keywords that are associated with a part or feature.

One piece of good advice that you can count on is if you already have a file-naming scheme, and it is not causing problems, stick with it. Changing part numbering schemes will probably cause more problems, even with a flawed scheme. It is likely that any system you come up with is going to have a flaw of some sort in it, and replacing flaws you know with potential flaws is not a move forward.

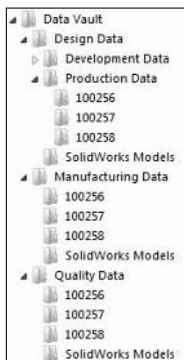
Organizing Files

You can organize SolidWorks documents similarly to the way you would for data from other 3D modelers such as Inventor, Solid Edge, or Pro/ENGINEER. It is widely different from how you would organize 2D design data, however. 2D design data is typically flat files without relationships or hierarchy, much like image or music files. Typically, these types of files are simply put into folders that can be browsed easily or searched with Windows search functionality.

Folder structures for flat files can take a number of forms. I've seen folders named by part number range, by the department that uses or creates the data, by customer, by file type, and other sorting methods. If you have an effective search tool, organization doesn't matter much, although some people prefer to browse to files rather than use search. Figure 3.5 shows a potential organization scheme with the top-level organization by department.

FIGURE 3.5

Organizing data by department



Part I: Preparing for SolidWorks

How you organize SolidWorks files is very dependent upon what type of data you are creating and how it is used. Many valid organization schemes exist; your main task is to identify one that works best for your business practices and minimizes the drawbacks.

An example of widely different needs for organizing files would be a company that makes a product with all custom parts, where all or most parts in the assembly are unique to that product and are not shared with other products. You might organize all files for that product in a single folder. Contrast the way you would store SolidWorks files for this type of company as compared with a company that makes every product from standard purchased items, parts, or subassemblies that are shared with other products the company makes. You would probably organize those files in folders by the type of part, standard hardware, springs, wheels, frame, cover, motor, and so on. I cannot make a recommendation that will fit every situation.

Some companies do only the design, and send the design out to be manufactured. Some companies do only manufacturing, without designing finished products. Some companies do both design and manufacturing. Data for products that are shipped to customers should probably be separated from data for tooling, fixturing, and manufacturing reference. Some documentation may even be sent directly to the customer, such as installation instructions, exploded view parts lists, or troubleshooting guides.

It might be considered a best practice to use the classifications of different types of documentation that you create as the top-level divisions. This would mean the top-level folders would be Design, Manufacturing, Quality, and Service. You would not typically include personnel, sales, facilities, or financial records with your design data, but I have seen companies include all official documentation such as Human Resources manuals, policy documents, forms, and so on in the documentation system. These documents are or should be revision controlled, with controlled access, and this is exactly what a documentation system provides. As a CAD Administrator, you may not control those documents, but you would control the system that enables access.

Creating libraries

Managing libraries and reused components are cases that bring up several interesting questions. Some companies like to use project or product names/numbers in their folder organization scheme and/or part numbers, but this scheme does not work well with library parts. Often libraries contain parts that have multiple sizes of similar parts; this arrangement works well with a semi-intelligent numbering scheme, where the main part number is a sequential number, and then an intelligent suffix identifies the size or other options for a part. You need to make tabulated drawings for this type of data, showing a breakdown of the intelligence in the scheme, such as shown in Figure 3.6.

The scheme shown in Figure 3.6 shows the sequential part number in the front, 54186 selected sequentially from a database to signify Pan Head Machine Screw; the next number, the dash number -0-, signifying the library type part; and a suffix, ABCD, that describes the characteristics of the screw.

FIGURE 3.6

Breaking down a semi-intelligent part numbering scheme



This system works well for this particular part, but what about when you start using flat head screws? You have 82- and 100-degree heads, and so you will need another classification for that.

When you establish the format for numbers like this, it is critical to get the numbers in the correct order. The main issue you need to deal with here is alphabetization. When you sort the numbers, you want them to show up in a reasonable order. In this case, the numbers will show up in the order of the sequential number. That doesn't make good sense, because the sequential number might as well be random, so sorting by that number doesn't give you any additional information. Possibly moving the "dash" number to the front would group all hardware (-0-) together, making it the prefix.

One of the characteristics of a library is that it can be used across departments, and in different types of documentation. Engineering can use the screws to put together a product that is shipped to the customer, and manufacturing can use screws to design fixturing to assemble the product.

In short, libraries are a great way to reuse data. Toolbox, an automated fastener configurator library, creates a bit of an administration headache, but you can get it to work. How well it will work for you depends on what you want to get out of it and how you plan to use it.

Cross-Ref

Chapter 10 covers Toolbox implementation methods in depth. ■

Avoiding common mistakes

The biggest mistake that companies make when setting up an organization scheme is to make it too complicated. Browsing is out. Searching is in. When you perform a search, the only reason you would want to limit the search would be that the complete search takes too long or produces too many results. Therefore, unless you have enough data to slow down your search mechanism

Part I: Preparing for SolidWorks

significantly, you have little reason to organize your folders at all. It may seem foreign to you to throw all the data into one big folder, but when you use search rather than browsing, that actually becomes an option.

If you feel that you have to organize for browsing, try to keep the hierarchy to a minimum. Needing to browse more than four levels deep is probably unnecessary. Three levels are sufficient for most purposes. If you have to think about the options for more than a few seconds, your scheme is probably too complicated. Anyone browsing to data is going to rely on quick decisions.

Suggesting options

Some options for top-level organization would be the By Department example laid out earlier. Remember that this included Design, Manufacturing, Quality, and Service. Within each of these top-level folders, you might consider each product or project to have its own folder, with a folder for items that are common to all (for example, hardware libraries), and possibly a folder that belongs to no project (such as standard forms or service repair logs). Within the product/project folders, it may or may not make sense to break it down any further than that.

Of course, the entire organization structure could also go exactly the opposite way, where the product/project folders are at the top level, and then the departmental breakdowns would be inside those folders. One of the determining factors to help you choose between the available options might be security. Does your company policy require that you establish permissions for different levels of access? Are these permissions easier to assign and maintain if they are done by department or by project?

Breaking references

I have seen some beginning SolidWorks users try to organize folders in Windows Explorer by SolidWorks document type. After completing a design, they took the files, just like they did their AutoCAD files, and moved the SolidWorks parts to one folder, organized by part number, assemblies to a similar area, and drawings to another area altogether. There was a little surprise when assemblies and drawings were always broken after the official release of the documents.

Moving files by using Windows Explorer breaks relationships between files because although you moved the files, you never told SolidWorks that the files were moved or where they were moved. The proper way to do this is either to use the Save As functionality within SolidWorks itself or to use SolidWorks Explorer. I describe the various types of relationships in the next section of this chapter.

To avoid the common mistake of breaking relationships by moving files around, it may be best to keep groups of files together when they are together naturally. If you create a set of files together,

they are together naturally. If you make all the parts to be put together into an assembly, those parts should probably remain in the same folder. There are some exceptions, such as parts that will be reused in other assemblies, including library parts.

Understanding SolidWorks document relationships

Many people get overwhelmed by managing the different types of relationships that can exist between SolidWorks documents. In reality, the relationships are not all that complex. If you are not already familiar with them, here is a bit of a primer.

Part files are the bottom of the food chain in SolidWorks. This is where it all begins. Parts have the *.SLDPRT extension. It may be capitalized or lowercase, depending on the template the file was created from, but the standard templates from SolidWorks use uppercase.

Part files may be put into assemblies or drawings. When that happens, the assembly or drawing references the part and must be able to find the part at all times.

Assembly documents may also be put into assembly documents, such that you would have a sub-assembly and upper-level assembly. Each assembly file has its own name, and the assembly file extension is *.SLDASM. Assembly files may be put onto a drawing.

Drawing documents are the top of the SolidWorks document food chain. Nothing references a drawing, but a drawing always (or usually) references something else. Parts and assemblies may be referenced by the drawing, and the drawing has to know where the part and assembly files are at any given time. There are small exceptions to all this with functionality called detached drawings, lightweight, or Speed Pak, but ignore those for now.

When you open an assembly that references a part, SolidWorks looks for the part in specific locations. If you assume it first looks for the part in the last location where the part was saved when the assembly was open, you would be wrong. That is actually the second or third place where it looks, depending on settings.

Note

Okay, this should scare you a little bit. SolidWorks looks for referenced documents in one or two other places before it looks in the last place you left them. This means that there is sometimes the potential that SolidWorks will find a part with the same name as the one you are looking for, but in the wrong location. This is one of the reasons why it is so important to have unique filenames, and to avoid multiple versions of the same part lying around in various folders. I hope that got your attention, because it is one of the most important pieces of SolidWorks file management information that you will learn in this or any book. ■

Part I: Preparing for SolidWorks

The fact is that SolidWorks can look for any referenced document in a total of 13 different places. This is why it is so imperative that you pay attention to the file management rules in SolidWorks and do not move SolidWorks files without telling the referencing file where the referenced file has gone.

Tip

For the list of all 13 locations where SolidWorks looks for referenced files, look in the Help menu under Search file locations for external references. ■

As a hint, the very first place SolidWorks looks is to see if the referencing document is already open. If it is, you get that document, whether it is the one you are looking for or not. Read that again. This needs to sink in because it is very important.

If you have an assembly open, and that assembly has a part called Cover.SLDPRT in it, then you open another assembly, which also has a part called Cover.SLDPRT, SolidWorks is going to put the Cover.SLDPRT from the first assembly into the second assembly. It will also give you a warning that the part doesn't belong in the second assembly.

This is why unique filenames are so important, in fact mandatory. It is also why descriptive names and intelligent names are discouraged. Neither descriptive nor intelligent names can guarantee uniqueness unless they also contain a portion of the name that is sequential and taken from a single list of numbers (a database).

Organizing subassemblies

In SolidWorks, many reasons exist why someone might want to make a subassembly. Subassemblies can be created for performance (speed of software operation) reasons, or maybe when the product is assembled, one set of parts is put together as a group. You might create a subassembly to facilitate patterning or motion. You might make a subassembly to assist with an exploded view on a drawing, or possibly to create a detailed view of a purchased component that has parts that must be removed in order for the component to be installed.

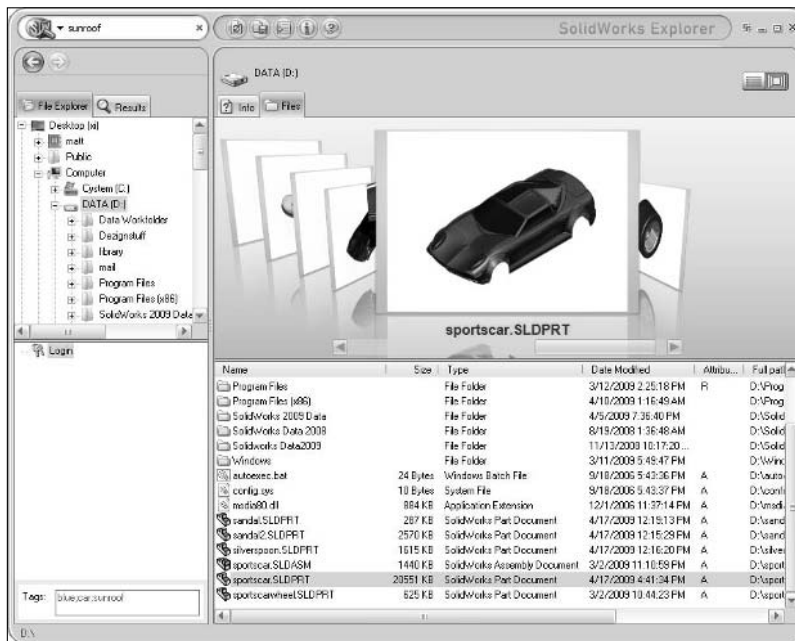
Of the many reasons that exist for you to make subassemblies in SolidWorks, few of them might actually result in a drawing document. Many assemblies are made for modeling reasons rather than documentation reasons. This practice is something you need to be aware of at your company. This concept comes up later in this chapter when I discuss making revisions or changing part numbers of subassemblies and how that affects other documentation.

Using SolidWorks Explorer and SolidWorks or Windows Search

SolidWorks Explorer doubles as the Workgroup PDM stand-alone client interface as well as the no-cost simple file management tool for SolidWorks data. Even if you have another full-featured PDM solution, you can still use SolidWorks Explorer to copy, rename, or replace files outside of the vault. You can access SolidWorks Explorer by choosing Start ⇨ Programs ⇨ SolidWorks. This may initially bring up only a small search box, but you can also expand this to show a File Explorer interface for browsing and a larger display interface. Figure 3.7 shows the SolidWorks Explorer interface.

FIGURE 3.7

SolidWorks Explorer expanded interface



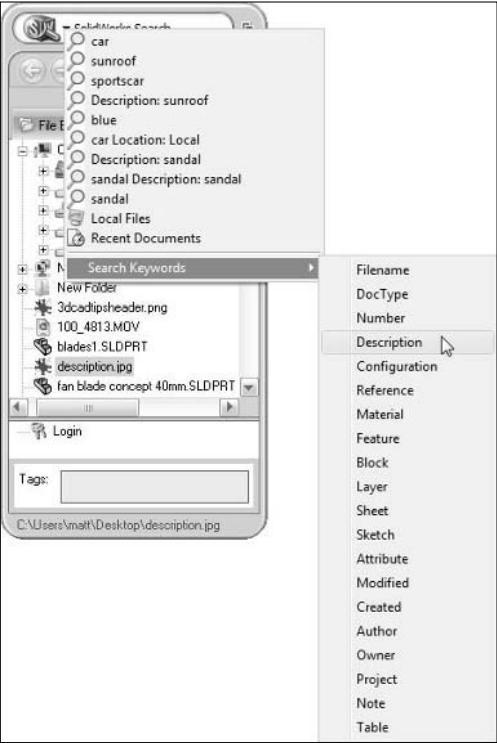
You should only expect SolidWorks Explorer to work well for searching if you have used Windows Search to index all the drives or folders you want to search. Otherwise, it will miss some items, such as tags and custom properties. To use SolidWorks Explorer to search for files, enter the name

Part I: Preparing for SolidWorks

of the file in the box at the upper left of the expanded interface. This brings up a Results panel immediately under the search box.

Figure 3.8 shows the partially expanded state of SolidWorks Explorer, along with some additional search narrowing options. The drop-down arrow in the search box enables you to select from recent searches or a set of commonly used keywords. When you select a document to view, the box grows larger and enables you to view graphics and other types of data.

FIGURE 3.8
SolidWorks Explorer partially expanded interface with search narrowing options

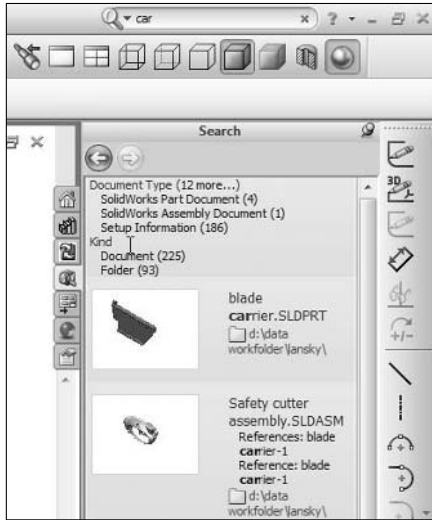


You may also recognize some of the functionality in SolidWorks Explorer as regular SolidWorks functionality. The SolidWorks Search box also exists in the upper right of the SolidWorks window, along with the File Explorer interface; so much of this functionality is also available from within the SolidWorks interface itself (see Figure 3.9).

In the chapters in Part 4 of this book, I return to SolidWorks Explorer in the guise of the Workgroup PDM stand-alone client interface.

FIGURE 3.9

SolidWorks Search and File Explorer functionality in the SolidWorks window



SolidWorks Explorer was originally meant as a stand-alone interface to do some of the basic file management tasks that you would otherwise need SolidWorks to do. You can use it to rename, replace, move, copy, Pack and Go, view, add custom properties and tags, and so on.

Controlling Revisions

Product development and engineering work require making changes to documents. Controlling which specific revisions of those documents are given to people who use them for assembly or manufacturing is one of the main purposes of file management. If you are new to document control, and setting up a new document control system, you will need to make many decisions. Even in light of the controversial topics I have already discussed in this chapter, revision control can be one of the most contentious of them all. If you are a novice in this area, listen to all the arguments carefully before taking sides, and be aware that most veterans of this field have strongly held opinions, probably related to a specific incident they lived through at some point in their careers.

First, you need to be able to articulate what is the difference between a revision of a part and a change that requires an entirely new part number. The simple answer is that if you can replace the old part with the new part, and it doesn't change the fit, form, or function of the product, you've got yourself a revision. If it is not a drop-in replacement, you should assign a new part number. This distinction is hammered religiously into new documentation recruits, and believe me, you will get it wrong as often as you get it right — even experienced professionals will disagree about

Part I: Preparing for SolidWorks

which route to take from time to time. You really need to be careful with the distinction between new part numbers and new revisions, and always have a reason for your choice that you can articulate clearly.

Beyond that is the distinction between revisions and versions. This definition may vary from company to company, but versions are usually defined as incrementally changed iterations between revisions. So if you have Revision A and Revision B, you might also have Revision A.1, where the *.1 is the first version between A and B. With those definitions out of the way, I'll move on to revision schemes.

Revisioning schemes

You should not expect any straight answers to your questions on revision schemes. As many revision schemes exist as companies that have set them up. Every company has different needs, and everyone who works at these companies has an opinion about how the revision scheme should work.

The interesting part comes when you ask people at a company that already has a revision scheme set up to describe that scheme; it will be rare to get any two people to agree on what it is that they are currently doing. If you are starting fresh, you are lucky, because you can train people right from the start.

An example of a revision scheme is that development revisions go from 1 to 100 or as necessary. Production revisions go from A to Z, leaving out O, I, Q, S, X and Z. These letters are left out because they may resemble numbers (S can resemble 5, O can resemble zero), resemble one another (O and Q look alike), or are used for other purposes (X may simply be used to cross something out). If you run out of letters, revisions go AA -ZZ with the same exceptions. Intermediate versions are for engineering and quality use only, not for production, and will be designated as A.1, A.2, A.3, and so on.

In the chapters on Workgroup PDM, I revisit revision schemes with a slightly different spin. I cover making revision schemes work within the limitations of the Workgroup PDM software. PDM software automates revisioning functionality, and the first casualty of automation is usually flexibility, so you may not be able to match a company's existing scheme 100 percent; you may have to make some compromises here and there.

In the end, your revision scheme should be simple and work for your business. You don't get extra points for creating something complex or clever. It needs to work for everyone all the time.

Revising models versus revising documents

One of the issues you must come to grips with is the difference between documents and models. Beyond that, the difference between models and documents in terms of revision handling is important. Many new CAD administrators ask the question of whether the revisions of SolidWorks models (meaning part and assembly files) need to match the revisions of the drawings created from those models. To take the question further, you might also ask whether SolidWorks model files need to be revision controlled at all.

Contemplate this situation, for example: You have a SolidWorks part and a drawing of the SolidWorks part. The first question you must answer is if these two files should always have the same revision level. Think about that one for a minute before you answer. Can you ever make changes to a SolidWorks part that would not affect the SolidWorks drawing? Might it ever make sense for a model to be at Revision B while the drawing for that part is at Revision A?

If you avoid assigning revision levels to models, the drawings become more difficult to manage. It may be difficult with a manual file management system to assign model revisions effectively, but with a PDM system, it is standard practice. I have seen companies try to make model and drawing revisions match, and I have never seen a successful implementation of this concept. Either you avoid making changes that need to be made or you wind up revising the drawing frequently when it doesn't need to be done. Model and document revision levels should not be locked into identical revisions.

As an example of why a model and a drawing do not need to have the same revision, first consider a model that goes through multiple changes before the drawing is ever made. The part might be at Revision C before the drawing is created, and the drawing must be released at Revision A. Another example is that some non-geometrical change happens to the model, which requires that it be updated, but does not require a change to the drawing. The change might be a color, modeling changes to remove a sketch error, or a change to the configurations used in the part that do not affect the drawing. You might choose to change custom property information that affects the assembly BOM, but not the part drawing.

Most people start by thinking that intuitively it is best to have the model and the drawing reflect one another completely, but by the time they have reasoned out a few scenarios, most are convinced they should be separate.

Using different revisioning schemes for models and drawings

You may decide that it is best to use different revision schemes for models and drawings, just to make the distinction clear and avoid confusion. For example, you might choose to use a numerical revision scheme for models and an alphabetical scheme for drawings. You could then call out the model revision on the drawing, such that drawing revision B has model revision 4.

Will models such as SolidWorks 3D parts and assemblies ever actually have a “released” status in your company’s documentation scheme? Issues like this are why the Documentation Manager and the CAD Administrator are often the same people, and, if not, need to work closely together. For your company to put together a definitive answer to this question may require some open-minded discussion between multiple parties. It is not the kind of question that has a single correct answer.

Storing revisions

If you are manually managing revisions of SolidWorks documents, this is where your nightmare actually begins. Revision management is why most parametric 3D modelers like SolidWorks push automating document management through PDM applications so much. Storing and retrieving revisions with a PDM application is reasonably simple. The actual work of manually storing and

Part I: Preparing for SolidWorks

managing revisions in SolidWorks can be painfully difficult and is very error prone. That said, it is actually possible to do things correctly, but the work is tedious enough that you will wish for a simple file manager by the time you understand how it all actually works.

When it comes to storing revisions, you will need to break several best practice suggestions. There isn't really a way around that if you want to store old revisions, and you don't want to use a PDM system.

Tracking revisions of models and documents

If you plan to keep old revisions of drawings, you also should plan to keep old revisions of the models that go with the drawings. Again, this is painfully difficult if done manually to the point of being completely impractical, but a snap if you do it through a PDM application. To do this correctly, you have to know which version of the model goes with which version of the drawing. PDM tracks this automatically, but if you were to do this yourself, it would require a lot of work to first store, and then more work to retrieve.

I don't think anyone would argue about the necessity of tracking revisions to drawings, but sometimes companies debate the issue of whether it also makes sense to track revisions of the models (parts and assemblies). It is certainly a recommended "best practice" for all companies to manage all data, but it also adds significantly to the workload if you are trying to manage revisions manually.

If your business uses primarily paper documents or electronic DWG/DXF (2D drawing formats) or PDF (Portable Document File) documents to have products manufactured, then your SolidWorks drawing documents are the primary document type, and model data is just reference. I don't actually recommend not managing model data, because it is incomplete and a sloppy way to work. However, if you are managing files manually anyway, it may be the difference between absolutely nothing and something. You might think this is a valuable shortcut, but I warn against it. Consider as an alternative that you could save out a DWG file at the end, and just manage the DWG file. This is also poor practice, but one that is better than trying to just manage the SolidWorks drawings without the models. Managing translated DWG files is considered bad practice, because all the associativity of the SolidWorks drawing is simply discarded when you move to the DWG format. In addition, translating out to DWG is a method often employed in failed SolidWorks implementations, where users are too busy leaning on legacy tools to focus on moving forward. Mixed 2D-3D CAD environments are certainly valid, but typically, data created in one system stays in that system.

Most companies will decide that they need to manage model revisions as well as drawings. Part of the reason for this is that the drawings without the models aren't worth much. In addition, if a model changes, your drawing changes, so the only complete way to control a drawing in SolidWorks is to also control the associated model.

Changing revision levels for documents

Let's say you have an assembly at Revision A, all the parts are initially at Revision A, and the assembly drawing and all part drawings are also at Revision A. Then you make a change to one of the parts that requires it to go to Revision B. Does your assembly also go to Revision B at this point? What about your part drawing and assembly drawing?

Your assembly should not change revision level if the only change is the revision level of a part. An assembly revision-worthy change to a SolidWorks assembly document would be removing or adding a part, changing an assembly feature, or possibly changing mates.

Also, recall the “fit, form, or function” rule to determine if a change to an assembly would require the assembly to get an entirely new part number altogether. Of course, that depends on if your assembly scheme actually lines up with any assembly documents.

Updating stored revisions

When moving from one version of SolidWorks to another, say from SolidWorks 2009 to SolidWorks 2010, SolidWorks technical people often recommend CAD Administrators use the SolidWorks Task Scheduler to save all your files in the new version format. Part of the reason for this is so that large assemblies will work faster because they do not have to update all the files on the fly and write them to memory. On the surface of things, this is a good idea. However, is it really a good idea? I think not.

If SolidWorks moved from one version to another without introducing problems with files, this would be a great idea. However, the truth is that often SolidWorks introduces small changes in how mates are evaluated, how a spline is calculated, or the conditions used in a Boundary surface affect the final shape. Because of things like this, moving from one version of SolidWorks to another is rarely an uneventful task, especially for assemblies and complex parts.

SolidWorks offers tools that will even go into the Workgroup and Enterprise PDM vaults and change files directly inside the vault without changing revisions. However, it does not provide these tools for other PDM/PLM applications like SmarTeam, even though SmarTeam is a Dassault product. To me, this is a cardinal sin from a document control point of view. If a file is changed in any way, you should change the revision. Updating a file between SolidWorks versions has the potential to change geometry in unpredictable ways, and it should be done on a case-by-case basis so that you can catch the problems as they occur.

You will need to weigh for yourself how you want to manage this issue. If you have large assemblies of mostly simple parts, the effects on your models may not be that great and the time-savings may be the driving factor, but if you have drawings with a lot of annotations or various view types or complex shapes with lofted splines, fit and boundary surfaces, and so on, your data may be critical enough to check it on a per document basis.

Putting revision levels in filenames or metadata

It is common practice for editors of flat files (files without any external links to other files) to put revision levels in the filename. This works well enough for Word, Excel, or even AutoCAD files, but it doesn't work well for SolidWorks, especially for SolidWorks part and assembly files. When you think about how to handle revision levels in SolidWorks documents when manually managing files, you come to two conclusions, both of which are considered poor practice.

Part I: Preparing for SolidWorks

If you have ever worked with AutoCAD or another 2D package, and you were responsible in any way for controlling revisions in that package, you probably are used to putting the revision level into the filename by appending “Revision A” onto the filename just before the extension.

In SolidWorks, doing this can work, but it may also cause files to become disassociated. When you use a PDM product with SolidWorks, the revision levels are kept in a custom property inside the SolidWorks file and/or inside the PDM application’s database. Remember that changing a filename can break the link between a document and its referencing document.

If you place a revision label inside a part (in a custom property), you cannot see it outside the part except with SolidWorks Explorer or a PDM application. If you are manually managing files, that’s inconvenient. So how do you manually manage files and not put revision levels in filenames?

Revision levels in filenames

If you place the revision levels in filenames, you can break links between parts. If you have large amounts of data, managing this manually becomes very tedious, and you will have to use a program like SolidWorks Explorer (to mass replace “Revision A” with “Revision B”). Of course, something like this is also error prone. If you have relatively few documents to work on, this might be relatively painless. I have seen people put dates in filenames and used that as a revision level when no formal revisioning scheme existed. I have used dates in names when my project consisted of a single part. When dealing with a single part, there are no references to break.

Revision levels in folder names

An alternative to revision levels in filenames might be revision levels in folder names. Again, if the files inside the folder are referenced by files outside the folder, you may also break associativity. Another problem with this system is that it is not a requirement for all the parts and subassemblies to have the same revision level, so parts could be scattered all over the place. On the other hand, you might just put the assembly revision level in the folder name and copy the entire folder to make a new revision level, though this does nothing for managing part revisions.

The main problem with this way of handling revisions is that it creates multiple copies of files with the same name. Remember the 13 paths SolidWorks looks through to find referenced files. Revision levels in filenames are bad because you can break links to assemblies and drawings, but at least the errors are obvious. Revision levels in folder names are even worse because the error of picking a part from one of the obsolete folders may look right, but it may not be immediately obvious that it is a mistake.

Mixing methods

Ironically, the best solution to two different bad ways of doing something might be to combine the bad methods. This might also be complicated and not immediately obvious, but it will help with retrieving old revisions, the next topic in this chapter, and it will solve the problems I mentioned previously.

If you leave the current revision files without revisions in their names, and only add revisions to the filenames after a revision becomes an “old” revision, it is the most reliable method for manually managing files. It breaks the links when revisions become outdated and maintains the links for latest revisions. Inside of SolidWorks, the way you would save out the old file and go forward with the latest revision is to choose File ➤ Save As with the Save As Copy option, renaming the copy to include the old revision level. This method is not completely intuitive, because if the latest revision is C, you are saving the file with a revision of B.

For example, the latest revision files would be 12345.SLDASM, which contains the parts 12346.SLDPRT and 12347.SLDPRT. These files might be Revision A, Revision B, and Revision C, respectively. The old revisions for the files could exist in the same folder or a different folder, and would be named 12346 Revision A.SLDPRT, 12347 Revision A.SLDPRT, and 12347 Revision B.SLDPRT.

Any manual method is going to rely on a group of people remembering to do the right thing every time, and, thus, will be prone to many errors. However, this is the only method I am aware of that actively leaves the valuable SolidWorks functionality intact and minimizes potential for human error.

Retrieving old revisions

In any document control process, you have to be able to retrieve old versions of files and documents. If your company has a service division that repairs old product, accessing old versions of documentation is essential. If piecing back together an old SolidWorks assembly is part of that process, the “mixing methods” technique enables you to do that easily. When various revisions of parts and assemblies must be maintained for repair and maintenance business or when end items must be reproduced for compatibility reasons, this is typically handled using As Built Control Record (ABCR). If this is a requirement for you, you must seriously consider an advanced PDM application.

The first thing you would have to do would be to find a BOM that shows the revision levels for all the parts at a given Revision of the assembly. Then you pull all those documents and remove the “Revision X” part from the filename.

To retrieve an assembly along with parts from the “revision levels in folder names” system, you pull together the correct files from the correct folders. No renaming is necessary. To retrieve data using the “revision levels in filenames” method, you would need to find the correct assembly, and then the assembly should automatically find the correct parts. You should also be aware that the likelihood of an error with any of these methods is high. Any technique for manually managing electronic files is sure to be error prone.

Using Configurations to Store Revisions

In an effort to get around the need for a PDM system, I have seen individuals and companies invent some very creative ways to store revisions. One way is by using configurations. SolidWorks configurations is powerful, but it is not meant for storing revision data within the part file.

Part I: Preparing for SolidWorks

Another issue that may initially seem intuitively correct is to use configurations to store revisions. I have seen people do this and even claim that it works, but I do not believe that there is any way that using configurations can offer the same type of certainty that you will be able to go back to an old revision and get the data you want.

The problem is that in real revision level changes, items are deleted, and things are changed in such a way that you cannot account for the change with a configuration. Many changes to features are configurable, but some are not, and many non-dimension changes in sketches are not configurable.

For this scheme to work, you would have to limit your changes to dimensional changes that can be configured (remember that some dimensions used as parameters in some PropertyManagers cannot be accessed by configurations, such as the values to drive a helix). Hole Wizard holes have been only recently configurable (2009).

Beyond those arguments against this method, if you ever wanted to use configurations for any of the functions for which they were intended, you wouldn't be able to because the configurations would already be commandeered by the revision method. Again, this is an idea I don't believe has any real application in real usage.

Summary

It is important to consider carefully how you will name and organize your SolidWorks data. Naming conventions or part numbering systems will be with you and your company for a long time, so it is important to come up with a reliable system that does not need to change as your company and data sets grow.

Document management in SolidWorks is more critical and more difficult than with file types that do not build relationships and links between files. Revision control is also more critical. Documents can be managed manually using Windows Explorer, but it is difficult, requires a lot of discipline, and is very error prone as projects and the workgroup grow in size. Even a simple PDM package will automate most of what you need to do for document management, revision control, and document sharing between multiple SolidWorks users.

Operating System Tools You Need to Know

In some cases, as a SolidWorks Administrator, you have to act as your own IT department. This adds many extra duties, and often requires a lot of extra research to learn about topics you might not know much about. The tools listed in this chapter help you simplify many tasks that otherwise might become very tedious or time-consuming. The goal of this chapter is not to make you into an IT professional, but to make sure you have at least the baseline level of skills that you need to troubleshoot and manage a Windows computer to run SolidWorks.

All the screen shots in this chapter were taken in Windows Vista, service pack 1. Shortly after the time that this book is printed, Windows 7 will be available, and will likely quickly become the operating system of choice for many types of users, including SolidWorks users. Most of the Windows 7 settings will be similar to the Vista settings. Vista is being used in this case for two reasons: first because it is the only officially supported operating system available at the time of this writing, and second because it represents a middle ground between Windows XP and Windows 7. I expect all three operating systems to be in use and overlapping while this book is available.

SolidWorks has also announced that SolidWorks 2010 will be the last version of SolidWorks that is supported on Windows XP, and that Windows 7 support is tentatively planned for SolidWorks 2010 sp1. Several users have reported that using SolidWorks 2009 on Windows 7 RC1 (release candidate), SolidWorks is already functional and stable.

IN THIS CHAPTER

Running your computer

Respecting the Windows Registry

Controlling your booting options using Msconfig

Accessing more memory with the 3 Gig Switch

Completing system maintenance

Using the Network Monitor for distributed computing

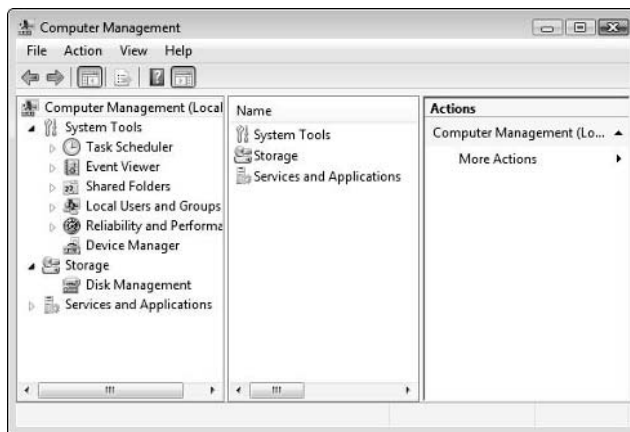
Managing Your Computer

Managing a workstation for SolidWorks use is not difficult, and does not require a lot of specialized knowledge. The biggest requirements are discipline and judgment. Keeping a computer in shape to run SolidWorks is mainly an exercise in computer hygiene. Troubleshooting tools become necessary if the user of the computer does not keep it tidy and clean.

The set of tools found in the Computer Management dialog box is a great place to start if you are troubleshooting a system or are simply looking for specs. You can access the Computer Management dialog box, shown in Figure 4.1, by right-clicking on the shortcut to Computer (in Vista and Windows 7) or My Computer (in Windows XP) from either your desktop or Windows Explorer.

FIGURE 4.1

The Computer Management dialog box



This screen shot was taken in Vista; therefore, this area may look different in XP or Windows 7.

Using the Task Scheduler

The Task Scheduler enables you to set up actions that occur when something else happens. For example, you might want to start a specific program when a user logs in or run a script that converts all files in a folder to PDF files at a certain time of day. Establishing tasks through the Task Scheduler is a good way to automate repetitive tasks that you might otherwise have to remember or remind a user to do. Automation saves time and increases accuracy, but it also can limit options, so be aware that it may not be the answer to every need.

Getting useful results from the Task Scheduler often involves using script writing. Scripting can be relatively simple or complex. Batch files are essentially DOS (Disk Operating System) commands that are strung together and can be used to achieve certain types of results. More complex scripting is often done using VBS (Visual Basic Scripting), and you can find many examples of finished VBS scripts on the Web. While I do not cover scripting in this book, it can be a very useful tool for any kind of CAD Administration.

Using available triggers

Automated tasks need to meet some criteria in order to start. This is done using triggers to set a task in motion. Triggers can occur as follows:

- On a schedule
- At logon
- At startup
- On idle
- On an event (from the system event log)
- At task creation/modification
- On connection to a user session
- On disconnect from a user session
- On workstation lock
- On workstation unlock

Triggers can also have conditions under which the action may or may not run. For example, conditions could be if the computer is on AC power (as opposed to running on batteries), or if a particular network connection is available.

Using available actions

The Task Scheduler can perform different types of action when the trigger event is detected. The actions available in the Task Scheduler are:

- Start a program/run a script
- Send an e-mail
- Show a message

This might seem like a limited list compared to the available triggers, but the scripting capability opens up the possibilities significantly.

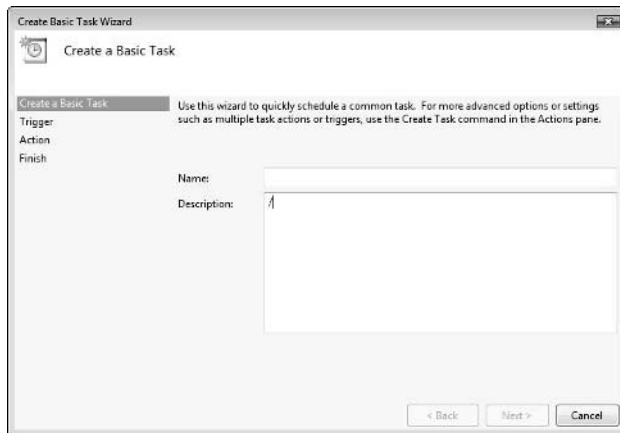
Part I: Preparing for SolidWorks

Using the Create a Basic Task Wizard

The steps for using the wizard are the same as those you would take without the wizard, but with the wizard, the steps are separated on different screens, and may be less intimidating for the user who is unfamiliar with the process. The wizard steps are shown starting with Figure 4.2.

FIGURE 4.2

Starting the Create Basic Task Wizard



You must start by naming the task. This helps you understand what the task does at a glance when you come back to edit, copy, or stop the task later on. The description box enables you to be more detailed, and explain how the task works, spelling out the arguments used or a more detailed description of why the task exists.

To navigate through the wizard, you click the Next and Back buttons at the bottom of the dialog box. It may be tempting to use the steps displayed at the left, but that does not work. Figure 4.3 shows the trigger selection options. The time-based options such as Daily, Weekly, Monthly, and One time have further options available.

In this case, let us say you want to send an e-mail notification every day that this particular computer is turned on. Obviously, the task will not execute if the computer is not turned on when its time trigger passes. Figure 4.4 shows how the wizard accomplishes this.

FIGURE 4.3

Selecting a daily trigger and setting the Daily options

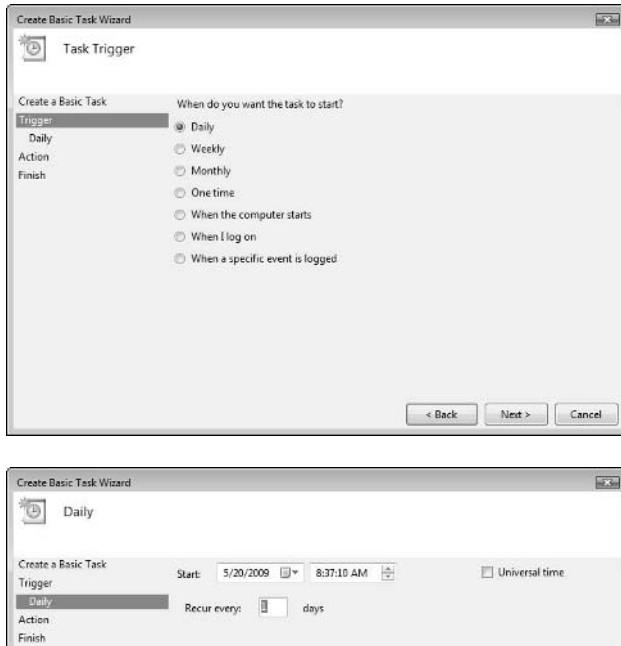
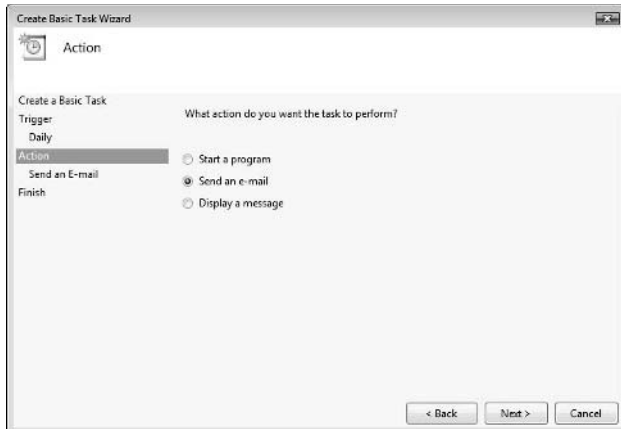


FIGURE 4.4

Automating the sending of a daily e-mail notification



Part I: Preparing for SolidWorks

Figure 4.5 shows how to set up an e-mail to be sent. Don't forget the SMTP server (outgoing mail server).

FIGURE 4.5

Filling out the e-mail form

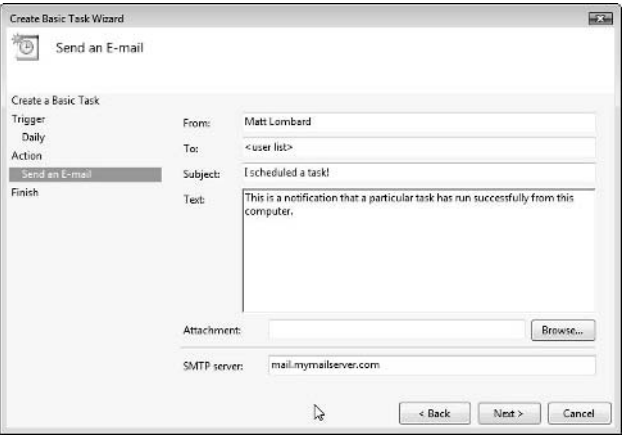
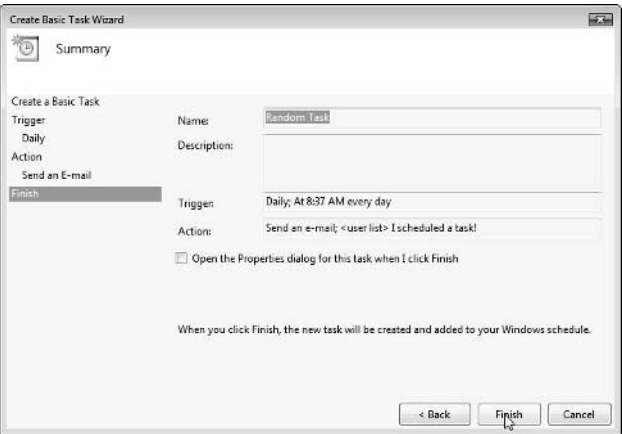


Figure 4.6 shows the final page of the wizard. It summarizes what you have done into a single location.

FIGURE 4.6

A summary of the task you have created

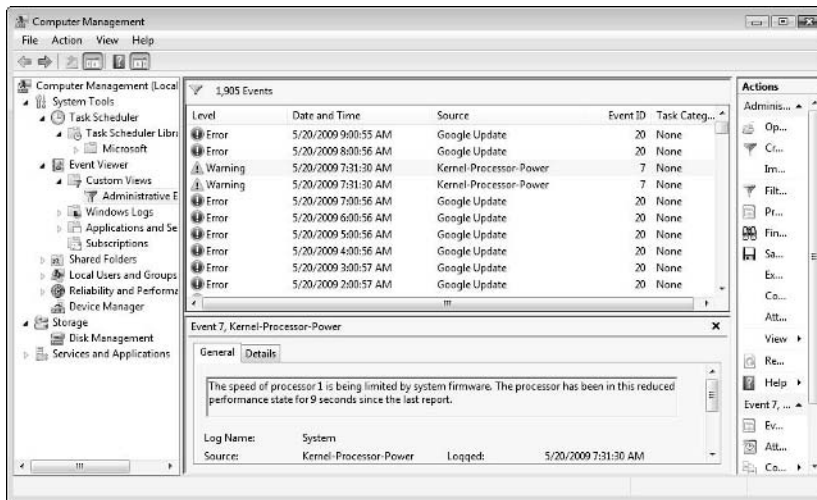


Reading the Event Viewer

The system Event Viewer is also found in the Computer Management dialog box. The Event Log is important when you are troubleshooting a problem that may be difficult to identify or track. If an error happens on your system, it is likely that the Event Log has recorded it. You may want to look at the Event Log periodically even if there does not seem to be any trouble; it will show errors you did not know you had, which may reveal a hidden problem or indicate what is hurting system performance. The Event Viewer in Figure 4.7 is shown as part of the Computer Management dialog box.

FIGURE 4.7

A sample selection from the Event Log



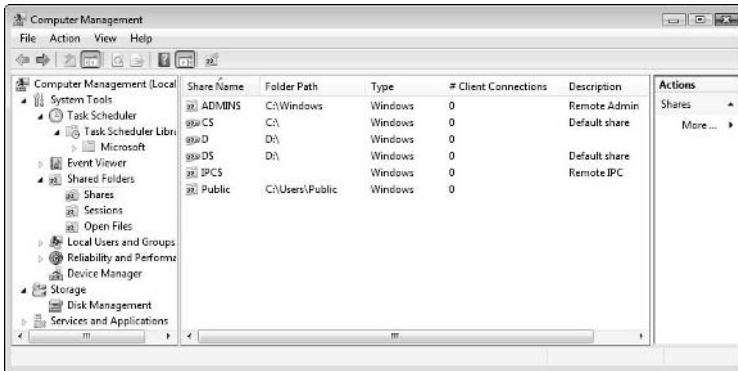
The situation described by the Event Log often looks far more dire than it actually is. Google Updater is a non-essential utility that does not show up in the uninstall list. The fact that it shows up so frequently in the Event Log is cause for concern, but not panic, although it has a red exclamation mark. The warning that is displayed as selected in Figure 4.7 is one that you may not be familiar with. Links in the lower box connect you with the Microsoft Web site for a more detailed explanation of the problem. Researching things you find in the Event Log can be interesting and educational, but it may not prove to be the most efficient use of your time unless you are looking for something specific.

Sharing folders

In the Computer Management dialog box, the Shared Folders section shows the existing administrative and user level shares that have been created on this machine. At companies with enough IT infrastructure to have file servers, sharing local drives is probably unnecessary except for possibly administrative shares for automated maintenance. Figure 4.8 shows the sharing section of the Computer Management interface.

FIGURE 4.8

Viewing shares through the Computer Management interface



If you are setting up a server, it is not advisable to share drives with operating system files on them. File servers should only share data drives. These are best set up as a single partition on a single physical drive unless you go for one of the RAID (Redundant Array of Independent Drives) options. RAID is used for both speed and data redundancy (for backup purposes). File servers commonly use RAID 5 and 10 (0 + 1). For more on RAID, see Chapter 2.

Setting up Local Users and Groups

The Local Users and Groups section of the Computer Management interface may or may not be of much use on the local computer, since they offer the biggest benefits for domain setup. Generally, a single machine is only set up with a few user accounts, the primary user, possibly a guest account, and an administrative account. The bulk of user accounts are usually set up on the domain server for the network.

When administering SolidWorks, user accounts can come into play in troubleshooting certain problems, particularly registry or permissions problems. When simpler troubleshooting fails, and you appear to have a problem that looks like a bug, but the functionality works properly on other

machines or for other users on the same machine, the problem can turn out to be a corrupt user profile. In cases like this, it is possible to troubleshoot and find the exact problem, but it is often more time effective to simply create a new user profile and continue with your work. Copying the old registry settings to the new profile is not advisable because you may bring the corruption along with the copied data.

Many IT departments limit the user rights of corporate computer users. It is understandable why they do this, because if they didn't they would spend more time repairing computers and reinstalling operating systems and software than anything else. Uneducated or undereducated users can wreck havoc on a computer that is used for business purposes by installing or uninstalling indiscriminately, accessing untrusted Internet sites, and bringing in infected media from home, among multiple other ways. You will never win the battle to secure your systems to the point that no one can harm them. For that reason, your best defense is a well-educated and well-disciplined set of users. You are better off educating users on how they should treat the company computers they have been entrusted with and then allowing them to police themselves than trying to prevent them from making mistakes by limiting what they can do. By locking down users into limited accounts, you make more work for yourself. This one particular issue makes evident why it is important to have or establish a good working relationship with your IT Department.

This balance between empowering employees and the myth of complete system security falls into the realm of best practice, although best practice from a different discipline — IT. There is always the balance between trusting users and giving them some control on one hand and locking things down to the point where the IT staff are the only ones who can make a change (but security is still not guaranteed). I personally prefer to educate and then trust users. If you cannot trust them, why did you hire them? You do many things to increase efficiency, but then if you bind users' hands by not allowing them to install service packs or other software, you may be working against yourself.

On the other hand, in some work environments you may indeed have very good reasons for making sure users do not upgrade independently (such as when you have deployed an administrative install, as described in Chapter 6). Many valid reasons for allowing users to install software exist, including testing new versions, service packs, or additional software. Arguably, these uses may be best reserved for you, the CAD Administrator, but there may also be times when you need to delegate some responsibilities.

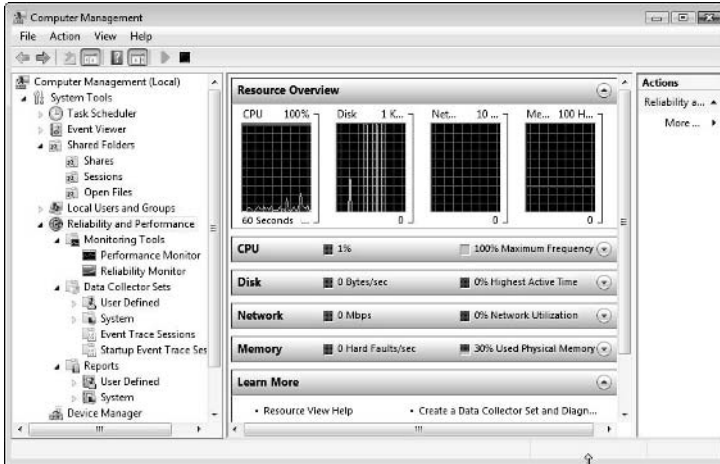
Monitoring Reliability and Performance

The Reliability and Performance tools listed in the Computer Management interface include tools that help you troubleshoot and assess the state of various metrics on your computer. Figure 4.9 shows the Reliability and Performance section of the Computer Management dialog box. This image is showing the resources in use for the CPU, hard drive activity, network, and system memory.

Part I: Preparing for SolidWorks

FIGURE 4.9

The Reliability and Performance section in the Computer Management dialog box



Performance

The Task Manager Performance tab enables you to see how the CPU is being used. For example, if your computer is a dual core or quad core, you see two or four windows, respectively, in the CPU Usage History area. Using Hyperthreading also increases the number of processor windows shown in this area. This information is similar, although not identical, to the information presented by the Task Manager (press Ctrl+Alt+Del or right-click on the Windows taskbar).

If you watch the core windows closely while SolidWorks is processing data, you can develop an idea for what types of documents are multi-threaded (able to use independently multiple processor cores). Assemblies and drawings both use multiple cores, but multi-body parts do as well. You can also access the Resource Monitor from the Performance tab of the Task Manager, as shown in Figure 4.10.

Monitoring numbers and usage on a machine becomes useful in several situations. If you need to troubleshoot a crash, particularly a memory crash, it is important to know how much memory is being used at any one time. It also becomes important if you are contemplating new hardware to know how much memory you use with your largest data sets loaded.

Reliability

The Reliability side of these monitors enables you to examine failures of various types over a period of time. Figure 4.11 shows the Reliability Monitor with a few failures.

FIGURE 4.10

The Task Manager showing CPU and Memory usage

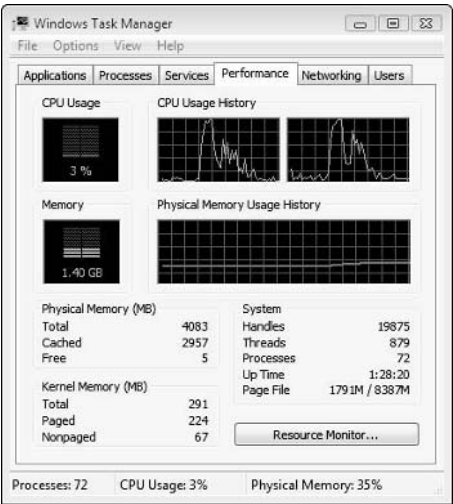
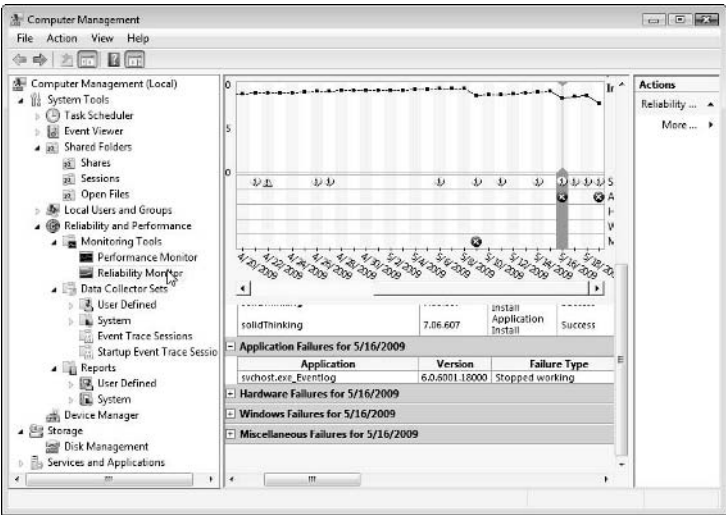


FIGURE 4.11

The Reliability Monitor graphs system reliability as a function of various failures.



Part I: Preparing for SolidWorks

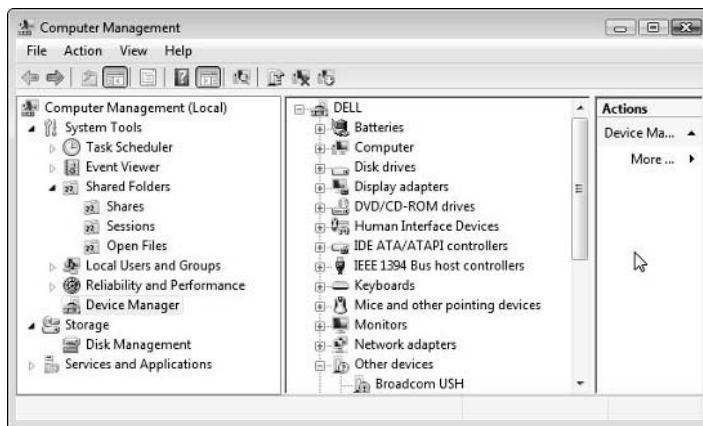
Tools such as this help you quickly pinpoint a problem in chronically failing software or hardware. The reliability index that appears at the right column of the chart can also help you quantitatively evaluate one machine or configuration against another.

Using the Device Manager

The Device Manager, shown in Figure 4.12, provides one location where all your installed hardware is listed. Devices that are failing or not installed properly are marked visually in the list to make them easy to spot.

FIGURE 4.12

The Device Manager listing hardware on the computer



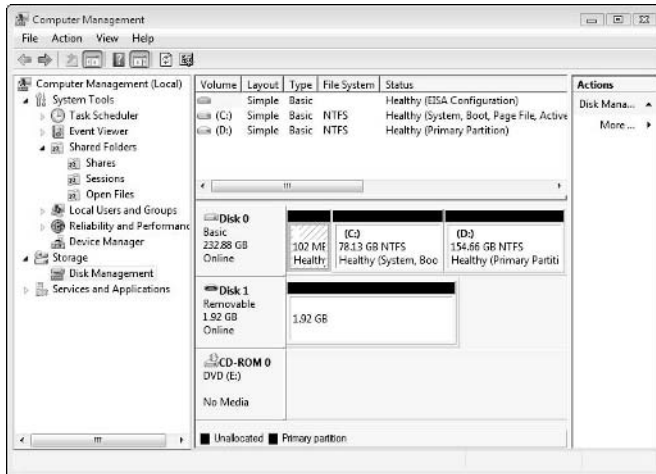
The device manager is a good place to check and update driver versions, hardware brands, and models, as well as conflicts with other devices (on the Properties ➔ Resources tab). You can also prompt the computer to search for newly installed hardware here. When troubleshooting hardware issues, you will return to this resource often.

Managing storage

The Disk Management section enables you to reformat, partition, change the drive letter, or check the file system type or size of any data drive. Figure 4.13 shows the Disk Management area of the Computer Management dialog box. Each row represents a physically separate drive. Notice that Disk 0 is partitioned into multiple drives. If Disk 0 had a physical failure, all the data on both C: and D: drives would fail. Disk 1 is actually an SD (secure digital) card reader with 2GB card in it.

FIGURE 4.13

The Disk Management area of the Computer Management dialog box



Services and Applications

The Services and Applications area lists services that are installed to run on your computer. Some of these run all the time, and some run rarely. At times, you may want to access the list of services to turn off something that might be getting in the way. Services generally run in the background, and you may not even know most of them are running, but if you have installed a lot of software and forgotten about it, extra services may be running that you don't need.

SolidWorks installs and runs some services that you never need until you update a service pack or use the SolidWorks Task Scheduler. It also installs some services that automatically do things when it thinks you aren't watching. Windows does the same thing, particularly with the Windows Search service. If you tend to do many searches for files on your local computer, then this may be something you need. SolidWorks Search depends on Windows Search and may be eating up computer resources without providing much benefit for you.

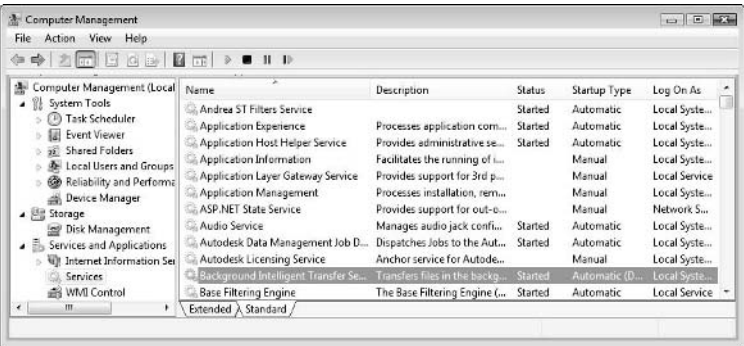
In the Services section, you can set the Startup Type for each service to Automatic, Manual, or Disabled. If the service is truly something you never want to use, you can turn it to Disabled, and it won't run. If you set it to Automatic, it starts when your computer boots up. Manual starts only when you start it. If you stop a service and notice your computer working strangely, you need to consider that it might have something to do with the service that you stopped.

Figure 4.14 shows the Services section, with the IIS (Internet Information Service) also showing. As a CAD Administrator, you do not need to be familiar with the IIS unless you get involved in Web-based PDM (Product Data Management) applications. IIS enables your computer to serve as a Web server. This is most frequently used on internal company networks.

Part I: Preparing for SolidWorks

FIGURE 4.14

The Services area of the Computer Management dialog box



Using the Windows Backup & Recovery tools

If you choose to use the standard Windows Backup & Recovery tools instead of other third-party software or devices, you can go to the Control Panel to get started. Backing up files looks like simply copying files from one place to another, but the Windows Backup & Recovery tools automate the process somewhat by continuing to copy files that meet criteria you set, and the tools enable you to retrieve specific files or groups of files.

Whatever backup scheme you come up with, you need to practice it to make sure you can always make it work, that your hardware and system are working as you expect, and that anyone helping you also understands the procedure. This is as true for recovery as it is for backup itself. If either operation doesn't work, the entire process is useless. Figure 4.15 shows the main page from the Backup and Restore Center, where the wizard-like steps for the Windows Backup & Restore Center start.

The tools, shown in Figure 4.15, enable you to back up specific files, file types or folders, or everything on your computer, depending on whether you click the Back up files or Back up computer button.

Backing up files

Backing up files is a necessary task in any computer administrator position, especially for a CAD Administrator. CAD data is the lifeblood of your company. Clicking Back up files option brings up the dialog box shown in Figure 4.16.

Chapter 4: Operating System Tools You Need to Know

FIGURE 4.15

The Backup and Restore Center gives you a starting place to initiate backups.

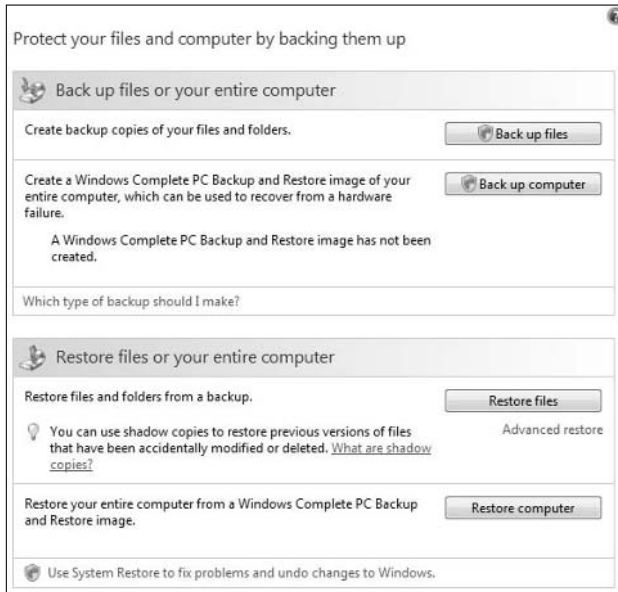
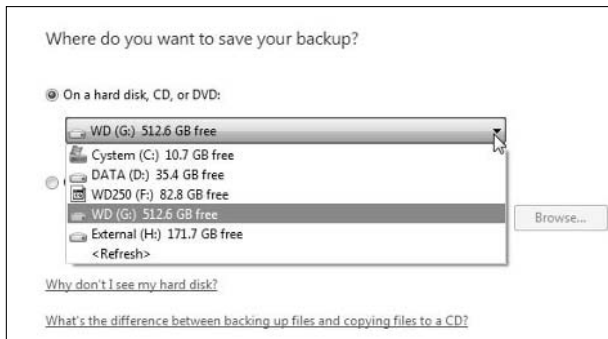


FIGURE 4.16

Choosing where you want to save your backup



Depending on what type of data you are backing up, how often it needs to be updated, and how often you need to retrieve it, you may choose different types of media to store your data. For example, if the data is not going to change, you may consider using CDs or DVDs to back it up. If there is much data, it may change frequently, and if you want maximum ease of retrieval, you may opt for a large, external drive. Smaller amounts of data may be written to thumb drives.

Part I: Preparing for SolidWorks

You can use network drives for backup. This can include available drives on computers available on your LAN (local area network), or even an NAS (network-attached storage) device

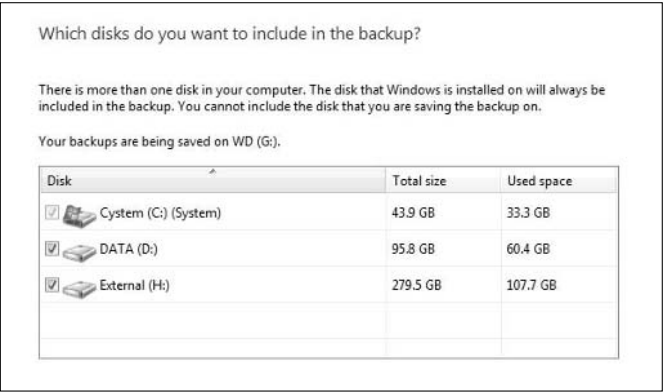
It is not recommended to use an internal hard drive for backup, and certainly not the same drive that is used for the operating system. If you are using a dual boot system, any drive that is used to store operating system files should not be used for back up. Because these drives have the most use, they are more prone to failure than other drives.

Even though the system drives must be backed up, the system files, program files, executables, and temporary files are not backed up. It is understood that e-mail files for online e-mail are not backed up because they are not stored on the local machine but rather on the e-mail server.

Figure 4.17 shows the options for selecting drives to be backed up. You cannot back up the drive that the backup data is going to be stored on. Selecting the box next to a drive does not mean that the entire drive will be backed up. If you want that option, click Back up computer in the main page of the Backup and Restore Center (refer to Figure 4.15).

FIGURE 4.17

Assigning drives to be backed up



Notice that the operating system drive is automatically backed up, and you cannot exclude it. You should also note that this wizard only backs up files on NTFS (NT File System) drives. If you are unsure about what type of drives you have, right-click My Computer (or simply Computer for Vista), select Manage, and then under Storage, select Disk Management. File system is listed for each drive, as shown in Figure 4.18.

The next option is to select what types of files will be backed up. The interface mentions only *.jpg and *.png by name. The other file types are vague. Figure 4.19 shows the selection options for file types.

FIGURE 4.18

Determining the file system for each drive

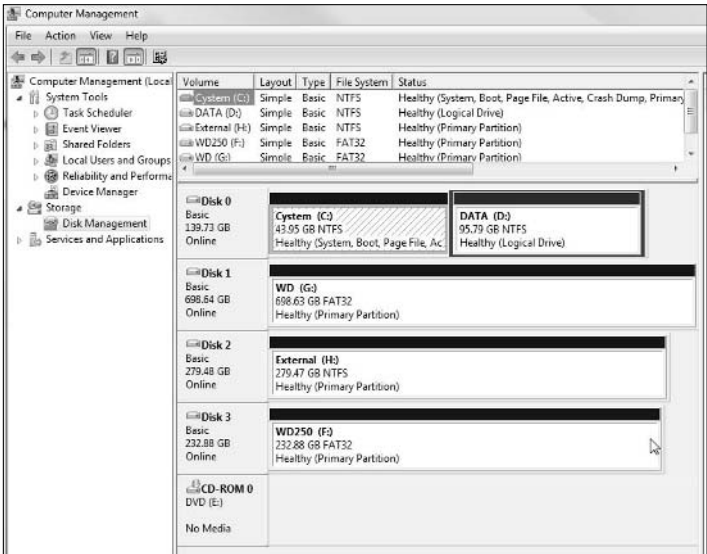
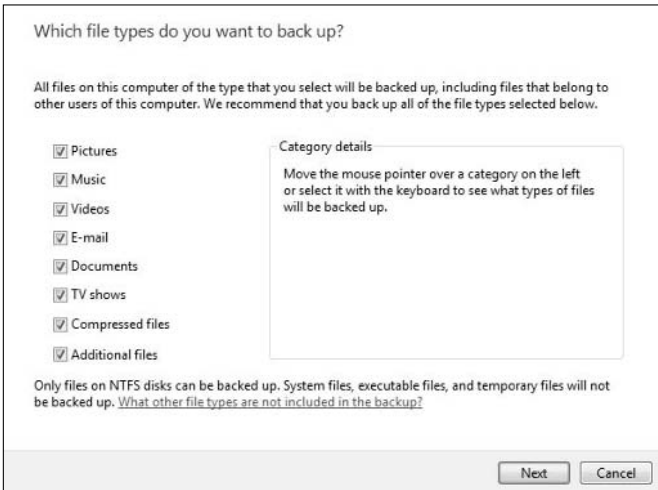


FIGURE 4.19

The file types to be backed up

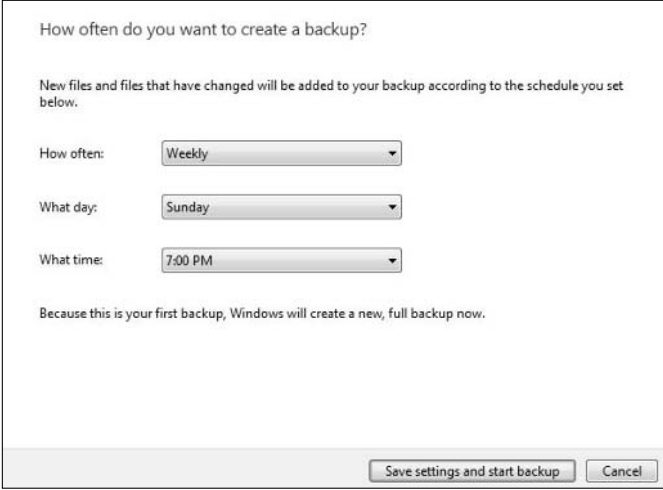


Part I: Preparing for SolidWorks

The next set of options deal with creating a backup schedule. Figure 4.20 shows this page of the backup wizard.

FIGURE 4.20

Scheduling backups



The screenshot shows a Windows Backup Scheduling dialog box. At the top, it asks "How often do you want to create a backup?". Below this, a note states: "New files and files that have changed will be added to your backup according to the schedule you set below." There are three dropdown menus: "How often:" set to "Weekly", "What day:" set to "Sunday", and "What time:" set to "7:00 PM". A message at the bottom says: "Because this is your first backup, Windows will create a new, full backup now." At the bottom right, there are two buttons: "Save settings and start backup" and "Cancel".

Backup schedules are important to maintain some consistency. If you do backups on a regular basis, users know what to expect in terms of file recovery.

Back up computer

Backing up your entire computer is a non-selective way to take a snapshot image of your computer at the present time. If you selected to back up the entire computer instead of selected files (refer to Figure 4.15), you see the dialog box shown in Figure 4.21.

In this case, both the C: and D: drives are system drives because the system is a dual boot: Vista and Windows 7.

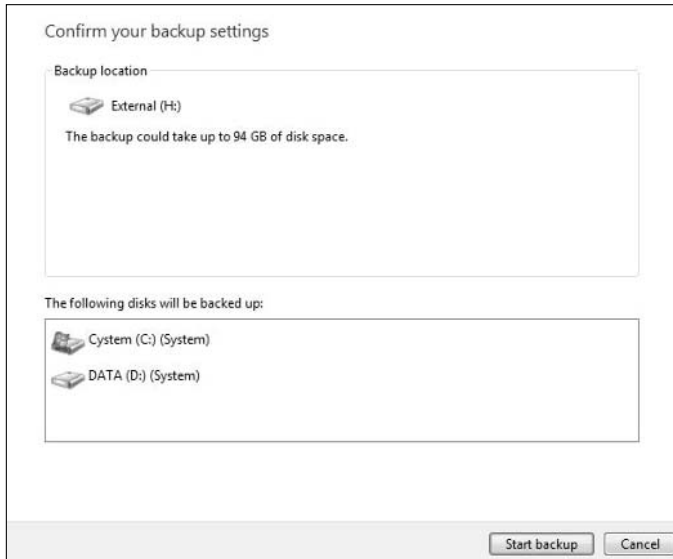
Using recovery tools

Several options are available for restoring a backup. You should be prepared to identify the kind of backup you did on the computer so you can restore the files appropriately. To initiate a restore, use the options on the lower half of the dialog (refer to Figure 4.15).

If you need to perform a complete PC recovery, that is not available from this interface, it is only available during bootup with the Windows installation disk. This is a more drastic measure than other options, because it reformats the backed up drives and writes the backed up data into the newly formatted space. This is why the backup must be on a separate drive from the original data.

FIGURE 4.21

Complete PC backup settings shows a summary of the selected settings



Using System protection and System Restore

It might have seemed a little counterintuitive that the backup capabilities exclude system files. You should really think of the built-in Windows backup capabilities as being for data files — primarily for data that you have created or collected from other sources. For recovering an operating system after something such as a drive failure or a virus attack, you should use System Restore.

You can turn on System Protection by choosing Control Panel ➔ System ➔ System Protection. It is on the left side under Tasks, as shown in Figure 4.22

FIGURE 4.22

Finding System protection



Part I: Preparing for SolidWorks

Clicking System protection brings up the System Properties dialog box on the System Protection tab. This interface enables you to check the amount of disk space required to create a new restore point, along with an option to invoke a system restoration. Figure 4.23 shows the System Protection interface.

FIGURE 4.23

The System Protection settings



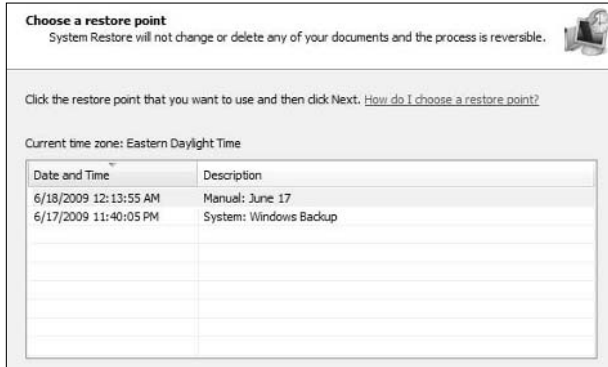
Unlike a backup, which may take hours for large amounts of data, system restore points may take only tens of seconds to create. Restore points are created automatically if the drive has a check mark next to it, as in Figure 4.23, or you can create them manually by clicking the Create button.

Figure 4.24 shows the interface that enables you to select which restore point you want to go back to. Restore points can be undone: if you go back too far or not far enough with your restore point, or even if you want to get rid of all the changes altogether, you can do it.

In the case shown in Figure 4.24, I have one automatically created restore point, and one manually created restore point. A system that has had System Protection turned on for some time will have a number of restore points from various dates. Restore points are not created on a regular schedule, but only when changes are made that affect system files or settings (system settings are Windows Registry changes).

FIGURE 4.24

Choosing a restore point



Recommending Backup and recovery options

I recommend using the System Protection settings, and making it standard on all PCs. This prevents you from needing to reinstall the operating system and all applications in the event of a hardware failure, operator mistake, or virus attack.

Backing up local files is a much less necessary requirement, depending on how you work. If you store all your design-related data on the network, local backups are not necessary. If users have the current day's work stored locally until it is finished, you might consider backing up the local machines. You might also give the user a local drive that is always backed up, and leave the matter of which data gets backed up to the user.

Understanding the BIOS

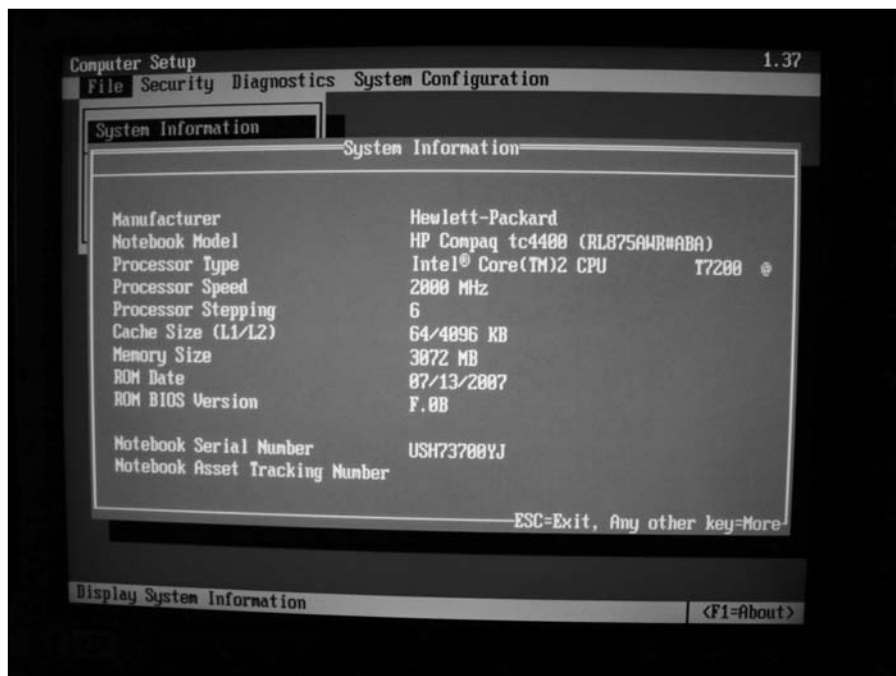
BIOS, basic input/output system, is the first code that the computer runs when booting up and is stored on a chip rather than on a drive. It enables the user to make various hardware settings. You can access the BIOS by pressing a key early in the boot sequence. The key is identified on the screen during boot and is different for various brands of computer.

When you enter the BIOS, it often looks like a crude DOS interface. It usually has basic keyboard and mouse input enabled. Changing settings in the BIOS can change the order in which your computer looks for bootable media (floppy drive, CD drive, hard drive); and it can be used to over-clock your CPU or set IRQ (interrupt request channels), all of which can adversely affect, or prevent the function of, your computer. Like the Windows Registry, the BIOS is no place for experimentation if you aren't familiar with the risk involved. Figure 4.25 is an actual photograph of the BIOS screen of an older computer. A photograph was taken because there is no screen capture utility in the BIOS programming.

Part I: Preparing for SolidWorks

FIGURE 4.25

A photograph of the BIOS screen



Going into the BIOS is usually a sign of something serious. You need to access the BIOS sometimes when installing an operating system and changing boot order. You may want to access the BIOS of a new computer to see how certain things are configured, such as overclocking or Hyperthreading. BIOSes from different manufacturers may have different options, or look different, or be accessed differently.

Working with the Windows Registry

The Windows registry is something you need to respect. To oversimplify it somewhat, the registry contains all the settings for all installed programs. In older software, settings were placed in text files that the software would read upon starting. Operating systems that are more modern moved all this information to a centralized location called the registry.

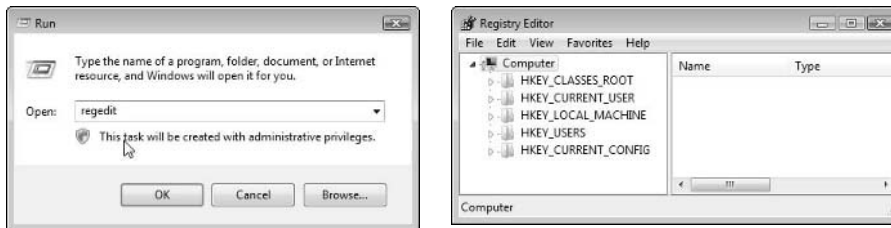
So part of the problem with the registry is that if you edit it and make any sort of mistake, you may prevent software from operating properly, or it may even prevent hardware from operating properly, because hardware settings are also stored in the registry. Editing or deleting the wrong things can cause hardware or software to fail or your system to not boot at all. This isn't meant to scare you; just to make sure you have respect for the potential consequences of experimentation. The real problem is that sometimes you have to go into the registry to make necessary changes.

Chapter 4: Operating System Tools You Need to Know

The tool for working in the registry is Regedit. To get to it you have to use the Run interface. Click the Windows button in the lower-left corner of the screen, and select Run. Type regedit into the text box, and you will see the interface shown in Figure 4.26. Optionally, the Windows Start Key and the letter “R” can be depressed to activate the Windows “Run” interface.

FIGURE 4.26

Accessing the Registry Editor



You may be called upon to access the registry in several situations, for example, to

- Copy settings from one user or computer to another
- Modify settings to change individual entries
- Clear the registry for a new installation of SolidWorks
- Delete a user profile in the SolidWorks section of a user profile

Other situations may also occur, but these are the most common.

The sections of the registry that will interest you most are HKEY_CURRENT_USER (HKCU) and HKEY_LOCAL_MACHINE (HKLM). Within each of these headings, you have Software and then SolidWorks. Under the SolidWorks heading are many settings to control a wide range of things, including an area for each release of the software that you have installed on the computer you are working on. For example, you might have SolidWorks 2007, 2008, and 2009 installed on a computer, or they were installed at one time. Each of these versions has an entry in the SolidWorks section of the registry to keep settings separate for each version.

All your System Options are saved in the HKCU area. If you wanted to copy all the settings from one user to another, you could copy the entire SolidWorks section of HKCU. This is similar to using the Copy Settings Wizard. The file type saved out from the settings wizard is a *.sldreg. The file type you can save out from the registry is a *.reg. When you double-click on a *.reg file, it is *automatically* added to your registry. The Copy Settings Wizard (by changing the extension, but not the format of the data inside) gives you a little buffer so that if you make a mistake, you are not forced to pay for it immediately. Incorporating any kind of data into your registry can be dangerous, unless you trust the data, its source, and your ability to make appropriate use of it.

The data inside a *.reg file is text and is actually readable. If you are familiar with the registry, you can see what kind of settings it is conveying. In fact, browsing the SolidWorks portion of the

Part I: Preparing for SolidWorks

registry can be quite a learning experience. You can change settings that you thought were just defaults, and even pre-turn off all the questions where SolidWorks allows you to choose “don’t ask this question again.” Still, you need to be careful. If you change settings with SolidWorks running, the settings may be automatically changed back, and it is possible to get things into such a state that SolidWorks will not run properly.

When a reseller or support person asks you to delete a section of the registry, it is probable that they will ask you to delete the entire HKCU\Software\SolidWorks section, or ask you to just delete the SolidWorks 2009 area, or ask you to do a deletion that’s even more surgical. If, for example, you get a toolbar shoved off the screen so that you can’t get it back, they may ask you to delete the User Interface folder. When the registry determines that data is missing from the registry, new data is created and populated with default values. So deleting the User Interface key resets your toolbars and interface customization back to their default settings.

Note

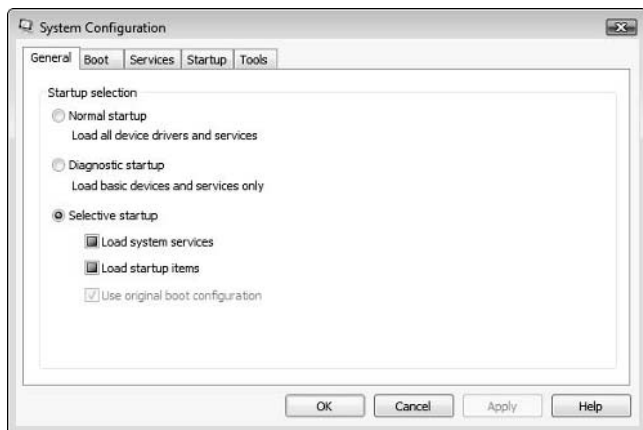
When a support person asks you to delete a portion of your registry, it’s wise to substitute the word “rename” for “delete.” If you get in the habit of renaming registry keys, you have a backup for that time when something goes wrong and you need to put your system back the way it was. Usually appending the word “old” to the end of a key name is sufficient, but if you have done a lot of renaming, a date may be an even better idea. ■

Using Msconfig

Msconfig is the command you use in the Run dialog box to initiate the System Configuration dialog box, shown in Figure 4.27. System Configuration gives you control over boot options, services that run, and services that start at bootup. It also gives you access to many system tools, some of which I have been discussed previously.

FIGURE 4.27

The System Configuration dialog box showing the General tab



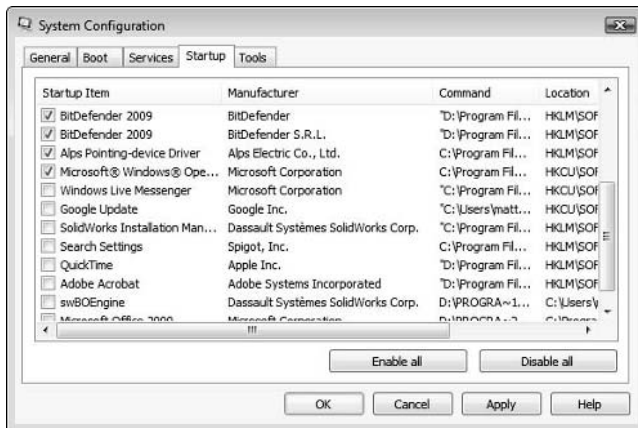
The General tab of System Configuration enables you to select how Windows starts up. A diagnostic startup is used when you have installed some software that is causing a problem and you can't fix it without getting to a stable operating system. Diagnostic startup has the bare bones, no frills services needed to run Windows. You may not be able to run all your software in a diagnostic startup because software that is more sophisticated requires additional services. Network access and OS graphics effects may be affected after you have rebooted using this option.

Using the Startup tab

The Selective startup is usually activated after you have made changes on the Startup tab. Changes on the Startup tab are usually made to control which services load at startup. For example, if you look on the Startup tab, you will see something called swBOEngine (see Figure 4.28). This is the SolidWorks Back Office Engine. It is responsible for the SolidWorks Task Scheduler and the Dissection operation. Notice in Figure 4.28 that I have turned that off, along with several other items that I simply don't want to be loaded on the computer as soon as it boots up. Many of these applications serve the purposes of the people who wrote the software rather than your purposes, and so are unnecessary.

FIGURE 4.28

The System Configuration Startup tab shows services that run on system startup



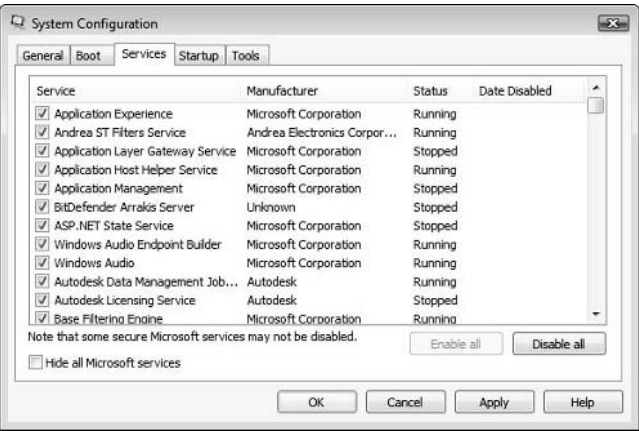
The Services tab

The difference between the Services and the Startup tabs is that the Startup lists the applications that start on startup, while the Services tab shows the services that start on startup. Figure 4.29 shows the Services tab. You can deselect the services or applications that you do not want to start.

Part I: Preparing for SolidWorks

FIGURE 4.29

The System Configuration Services tab



It is best to pare down extra services when you can. If you do not recognize something, it doesn't necessarily mean that your system can run without it. Experimentation is a harsh way to learn, but sometimes it is the only way. It would be best to avoid experimenting on production machines. If you want to poke around the System Configuration and registry, it is best to do it in a test machine in a controlled environment, rather than on a user's production machine.

The Boot tab

The Boot tab enables you to select which operating system installation will run by default if you have a system set up to dual boot. The Boot tab is shown in Figure 4.30. You can also set how long the delay should be when the system gives you the option to choose between the available installations. Several of the options on this tab are intended for booting into a safe mode environment with fewer advanced options, similar to the Diagnostic startup on the General tab.

Using the Tools tab

Possibly the most useful tab on the System Configuration dialog box is the Tools tab. It has many tools, some of which you have already seen and some you haven't seen, but need to. Figure 4.31 shows the list of tools available from the System Configuration's Tools tab. Most, if not all, of these tools are available from other locations as well, but it is nice to have a central location from which you can access many of the important tools at one time.

Chapter 4: Operating System Tools You Need to Know

To activate any of these tools, select the tool from the list, and then click the Launch button.

FIGURE 4.30

The System Configuration Boot tab enables you to select which installation should be the default.

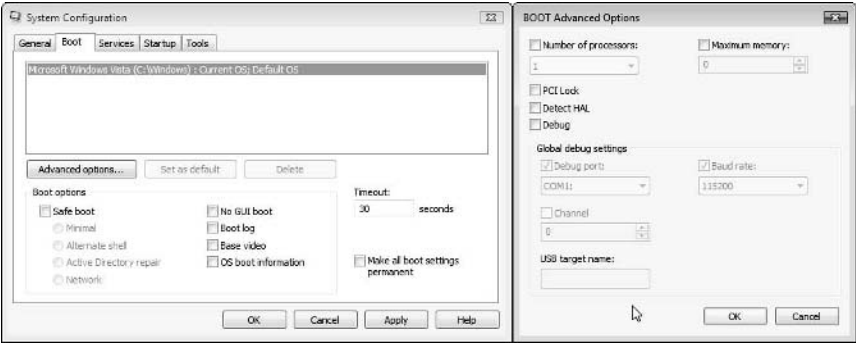
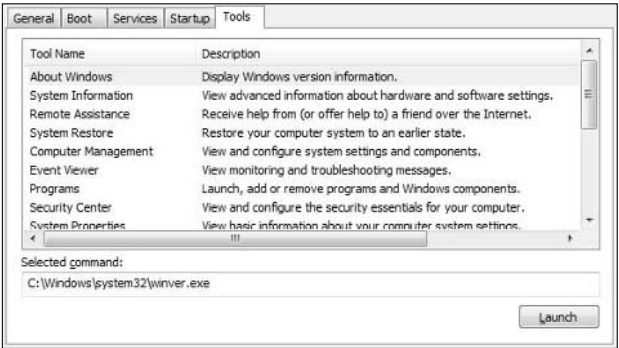


FIGURE 4.31

The System Configuration Tools tab



About Windows

The About Windows window, shown in Figure 4.32, shows the version, service pack, physical memory, and some legal notices

Part I: Preparing for SolidWorks

FIGURE 4.32

The About Windows window

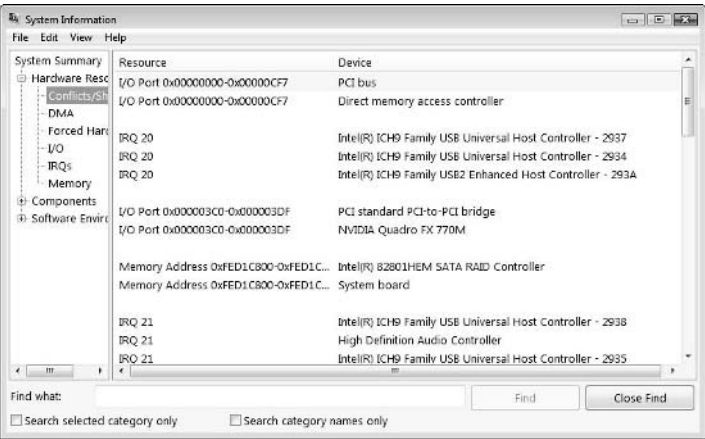


System Information

The System Information tool from the Tools tab has a lot of great troubleshooting information on it. It lists all the IRQs for hardware as well as conflicts for IRQs and other memory or I/O (input/output) issues. It includes detailed information on all your hardware and drivers, and even identifies a special category called Problem Devices for hardware that repeatedly shows errors. It's important to familiarize yourself with this area if you are not already. Troubleshooting systems will be much easier with information this close to hand. Figure 4.33 shows the System Information dialog box.

FIGURE 4.33

The System Information dialog box



Remote Assistance

Remote Assistance may turn out to be one of those things that really makes your job a lot easier. If you have an IT department available to you, you might consider asking them to help you get it set up, especially if you are providing support for many different users, and especially if some of those users are distributed about a WAN (wide area network). Remote Assistance enables you to drive another machine from your keyboard and mouse. It can be a very quick way to check a computer for what's called a PEBCAK error earlier (operator error). Consult with your IT folks to see how you can implement this tool.

System restore

You might have many reasons for needing to do a system restore. It is not intended for recovering from virus infections, but for undoing installations where a simple uninstall does not fix what the bad install did. Windows creates a restore point before installing software, which is what enables you to do a system restore. It's not as disruptive as using a ghost image to restore an entire computer or drive; it restores only the files affected by software installations.

Programs

You can access this same area by choosing Control Panel ⇨ Programs and Features (Add/Remove Programs). It is primarily used to install, uninstall, and repair software on your computer.

Windows Security Center

You can access the Windows Security Center through the Control Panel. It is used for antivirus and firewall settings. You can also use it to access the UAC settings and Internet Options.

System Properties

This area repeats a lot of information you can also access from other places, such as the OS version and service pack level, 32 or 64 bit, RAM, computer, and workgroup names, and so on. However, it also has a bit of a simple benchmarking standard available in the WEI (Windows Experience Index). This is just a general measure of the capabilities of your hardware to show the Windows interface. It's not clear how well this translates into precise benchmark results for SolidWorks, but I would say that there should be some sort of direct relationship. A higher score means a faster computer, period. This should give you an opportunity to compare machines from different suppliers.

Internet Options

The Internet Options settings are not necessarily directly relevant to CAD Administration, but every computer user should know where to find these settings, given that changes to them are frequently required for various reasons. The Internet Options are available from the Control Panel.

Internet Protocol Configuration

Internet Protocol is otherwise known as IP (not to be confused with Intellectual Property, which is also abbreviated IP). IP addresses for computers are often used when troubleshooting network or connectivity problems. The Internet Protocol Configuration tool accesses the DOS prompt and does the equivalent of the `ipconfig` DOS command.

Part I: Preparing for SolidWorks

Performance Monitor

This is the same tool you access from the Task Manager and from the Computer Management panel and it enables you to monitor several performance characteristics of the computer.

Task Manager

You can access this same Task Manager by pressing Ctrl +Alt+ Del or by right-clicking the Windows Task Bar.

Disable UAC

The UAC (User Account Control) is an option that has turned out to be very controversial in Windows Vista. The UAC is the functionality that makes a little warning pop up whenever the user does something the computer thinks the user needs permission to do. The goal is to help prevent computer viruses from taking over the computer, but it simply annoys users. If you have ever installed Vista fresh from disk, you probably were hit by the UAC three times before you figured out how to turn it off.

Your company may have a policy on the use of the UAC, but if they don't, you may just want to turn this one off. I have never had it save me from a virus, or even from doing something that I did not intend to do. The other place to access this option is through the User Accounts area of the Control Panel.

Enable UAC

The action is the opposite of the previous action: Disable UAC.

Command Prompt

You can access the Command Prompt by choosing Start menu ⇨ Run and typing in **cmd** (or using the shortcut of the Windows key and R simultaneously). In this area, you can issue DOS type commands for certain types of functions — such as **ipconfig** — to show the IP configuration of the computer. You can do many other things from this interface, such as formatting and error-checking drives; some file management operations such as creating, renaming, and deleting folders; and many other such options.

If you started using computers before the days of Windows, you may only need a little reminder to fall back into the DOS way of doing things. If you started using computers after, say, 1990, your exposure to DOS commands may be limited.

The Run interface uses commands such as

- **cmd**. Use this command to access the DOS command prompt.
- **regedit**. Use this command to start the Registry Editor.
- **msconfig**. Use this command to access System Configuration interface.
- **gpedit.msc**. Use this command to access Local Group Policy Editor.

Registry Editor

You can also access the Registry Editor from the Tools tab, but unless you are in the System Configuration interface already, it may be a longer way around than simply going through the Run interface.

Employing the 3 Gig Switch

The 3 Gig Switch is a tool used in 32-bit Windows operating systems that enables applications to access more memory. Under normal conditions, Windows enables applications to use 2GB of memory, and allots 2GB of memory for itself. This means that Windows 32-bit is limited to making use of 4GB of RAM. The actual number is somewhat smaller than that. When your Task Manager starts showing about 2.6GB or 2.7GB, you will start to see Windows crash due to memory allocation.

In my opinion, you should never need the 3GB switch with Vista because with SolidWorks, using 32-bit Vista doesn't make any sense. At the time you read this book, you should not be buying a new 32-bit operating system for use with CAD software. Therefore, unless you are using an old Windows XP 32-bit machine, you should not have the need to use the 3 Gig Switch.

I don't recommend the 3 Gig Switch with Vista because Vista is a resource hog. It takes an additional gigabyte of memory when compared to XP. For this reason, if you are using large data sets in SolidWorks, SolidWorks will have fewer resources available to it. In addition, in order to satisfy both SolidWorks and Vista, you need more than 4GB of RAM, which means that you need to use a 64-bit operating system. When you upgrade from XP, regardless of whether you used 32- or 64-bit XP, your move should be to 64-bit Vista or, preferably, Windows 7.

In brief tests I performed with benchmark models found on www.solidmuse.com, SolidWorks on Windows 7 performed 10 percent faster than on Vista. Windows 7 also used less space than Vista for installation (11GB versus 17GB), and Windows 7 had about a 30 percent smaller initial footprint on the system memory than Vista.

I include instructions on how to employ the 3 Gig Switch in Vista, but if you find yourself applying it to Vista, be aware that you may be wasting resources. Use 64-bit Vista, instead, if Windows 7 64-bit is not available yet for you.

You do not need the 3 Gig Switch with a 64-bit operating system, because the operating system memory limit is for the time being far above the current hardware limitation. (Motherboards can only accept a fraction of what the operating system can handle.)

Setting the 3 Gig Switch in XP

To set the 3 Gig Switch in XP, edit the boot.ini file, which is found in the C:\ root. Edit the file in Notepad. The default file should look something like this:

Part I: Preparing for SolidWorks

```
[boot loader]
timeout=30
default=multi(0)disk(0)rdisk(0)partition(1)\WINDOWS [operating systems]
multi(0)disk(0)rdisk(0)partition(1)\WINDOWS="Microsoft Windows XP Pro"
```

Edit or add the last line:

```
multi(0)disk(0)rdisk(0)partition(1)\WINDOWS="Microsoft Windows XP Pro with /3GB"
/3GB
```

If you add the last line, you have the option to start with or without the 3 Gig Switch. If you edit or replace the last line, it boots up automatically with the 3 Gig Switch employed. You should not be able to tell any difference between running with or without the 3 Gig Switch, except that your applications can now use an extra gigabyte of memory before they run out of memory and crash.

Setting the 3 Gig Switch in Vista

To set the 3 GB Switch in Vista, follow these steps:

1. Click Start ⇨ Programs ⇨ Accessories.
2. Right-click Command Prompt and select Run As Administrator.
3. At the command prompt, type `bcdedit /deletevalue IncreaseUserVa`.
4. Restart the computer.

Performing System Maintenance

Nothing is as important as system maintenance when it comes to keeping SolidWorks running smoothly on any computer. If you are having many unexplained crashes, part of the problem might be simple system maintenance.

SolidWorks is software, and like other software, it has its share of bugs, but when you hear of people who crash six times a day doing random tasks in SolidWorks, there is something going on other than just bugs in the software.

Doing and not doing

When you are trying to keep a system running SolidWorks efficiently, what you don't do is as important as what you do. To say that less cryptically, it is vitally important to avoid installing certain types of software on a machine you want to keep clean.

Not all software is created equal, and not all programmers are as ethical as you might want them to be. When you install some software, uninstalling it doesn't necessarily return your computer to the previous state. Sometimes some files, folders, or registry settings are left behind. Either this is often intentional for marketing, monitoring, or licensing, or in case you decide to reinstall the software again later on.

If you get enough of this type of installation remnants on your computer, and in particular, in the Windows Registry, it can have an adverse affect on the speed of your operating system. I used to blame “semi-professional” software for this type of sloppiness or craftiness, but it turns out that many major software manufacturers are also guilty. The only way to combat this is not to load software that you don’t need on your work computer.

Instead, have a spare computer around. This is more feasible than it might sound. The spare computer doesn’t have to be a top-quality machine. I use a previously retired SolidWorks computer as a test bed and workhorse for applications such as iTunes, which aren’t work related, but I like to have available anyway.

If for some reason your computer’s performance has degraded over time because of things like software installations you really couldn’t avoid, one solution to regaining some of the performance is to use a registry cleaner application such as CCleaner (www.ccleaner.com). It is available on the Internet free. It cleans out junk from the registry and cleans out Windows temporary folders.

Another option is to have a ghost image of the computer from when it was freshly installed and running clean. Reapply the image of the operating system drive, and you are right back to where you were. In this scenario, obviously you need to have your data somewhere other than on the imaged drive.

If you don’t have an image from when the machine was clean, you can reinstall the operating system. It doesn’t take as long as many people think. The part that takes a long time is reinstalling all the additional software that goes along with it.

Maintaining drivers

Drivers and driver conflicts receive a lot of attention when it comes to troubleshooting crashing systems. In particular, the video driver has a huge effect on SolidWorks functionality, but you need also to be aware that other seemingly unrelated drivers may cause unexpected problems.

To compound the complexity of the situation, be aware that the latest version of the driver is not always the best choice. Check the SolidWorks Web site (www.solidworks.com) for an approved list of video driver versions, with the approved list of video cards.

If you have built your own system or have replaced hardware in an existing system, you need also to check with the SolidWorks Web site to make sure the replacement hardware is certified. It also never hurts to stop in the SolidWorks forums and ask for a recommendation, or better yet, do a search for recent discussions on the topic, given it is covered frequently.

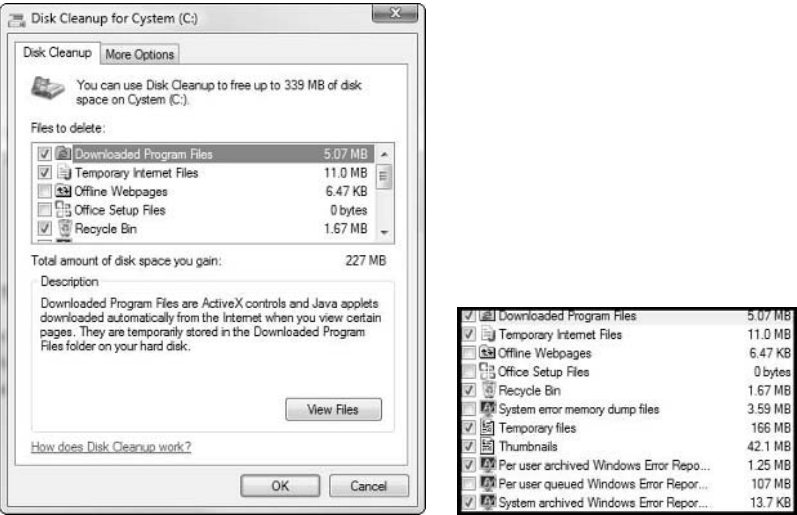
When testing for a conflicting driver, uninstall one driver at a time. You might even consider physically removing the hardware. Because the crashes are not predictable, this kind of troubleshooting can take a long time and be rather tedious.

Cleaning Windows

Windows keeps track of all the nooks and crannies where it keeps temporary files, so it is a good idea to allow Windows to clean those areas out from time to time. One of the best ways to let Windows do its own cleaning is to use the Disk Cleanup utility. You can find this by choosing Start → Programs → Accessories → System Tools. Figure 4.34 shows the Disk Cleanup dialog box and list of types of files that it clears out.

FIGURE 4.34

The Windows Disk Cleanup utility

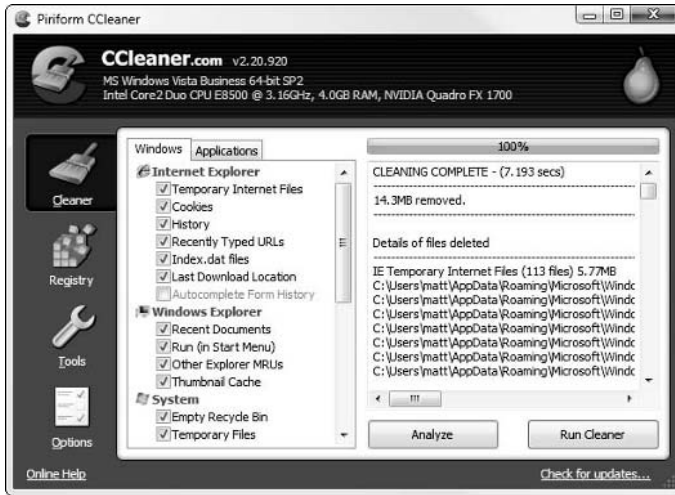


The utility tells you how much disk space will be freed up by cleaning out the files you selected, and even shows you a list of the files to be removed. You can click the More Options tab to remove programs that you don't use, as well as delete data created by the System Restore discussed earlier in this chapter.

I also recommend using a free utility you can download from the internet called CCleaner (pronounced "see cleaner" and found at www.ccleaner.com). CCleaner goes through your registry and removes unneeded or obsolete keys, and compacts the registry. It sounds a little frightening, but I have been using this for some time now and have not had it cause any difficulty. CCleaner also removes a list of files that you need to check out before you just click OK. I have found this to be an excellent tool for periodic maintenance. Figure 4.35 shows the CCleaner interface.

FIGURE 4.35

The CCleaner utility interface



SolidWorks creates folders in locations that need to be cleaned out from time to time. One of these paths is `C:\Users\<user name>\AppData\Roaming\SolidWorks`, and another is `C:\Users\<user name>\AppData\Local\TempSWBackupDirectory`. Depending on how much you use the software, these locations may fill up with data that you don't need.

Using antivirus software

Antivirus software has a long and contentious history with SolidWorks. For several releases, some brands of antivirus software would block portions of the SolidWorks installation process because the software thought it was doing things that it shouldn't be doing. For this reason, users had standing orders to turn off any antivirus application prior to installing any SolidWorks software or service pack. Many random crashes were blamed on the antivirus running during installation.

The situation has changed over the years, and now you can get a list of which antivirus applications are compatible with SolidWorks and which ones are not. This information is located on the SolidWorks Web site (www.solidworks.com) in the same System Requirements area where the video card data is found.

As of this writing, in the summer of 2009, the only antivirus application that failed SolidWorks testing was Kaspersky AV. You should check to make sure your antivirus application is on the list and passed.

Running disk checking and defragmenting

You may be feeling a little badgered by this point, but I'm going to keep on going. I'm going to remind you of all those little things that you always forget to do, but really do need to remember. Chief among those is defragmentation. Depending on how much you use, edit, and move files on your local hard drive, you may need to do this more or less frequently. Doing it once a week is probably sufficient. Defragmenting makes sure that all the data for a single file is located in a single block of space on your disk instead of scattered in fragments wherever there is space available on your hard drive. The less available space and the more file traffic a drive sees, the more likely that your files are fragmented. Your computer can read and write data faster to a defragmented drive.

You can use the Task Scheduler to help you keep up with the maintenance schedule. Find the Task Scheduler by choosing Start ⇨ Programs ⇨ Administrative Tools.

Microsoft Update

Regardless of what you think about automatic updates, they really are a good idea. Sometimes they can repair problems on your system. There have been examples where automatic updates caused problems (for example, when Internet Explorer 8 was installed as part of an automatic update), but Internet Explorer 8 is more a new product than an update to an existing one.

Using the SolidWorks Network Monitor for Distributed Computing

The SolidWorks Network Monitor is not a system tool, but it works like system tools. Network Monitor enables distributed tasks such as converting a set of files to a newer version by connecting multiple computers that have SolidWorks on them. At this time, the conversion is the only function available, but it is easy to see that this tool is headed to broader application in distributed computing for FEA and rendering applications within the SolidWorks family. To access the Network Monitor, go through the Windows Start menu in the SolidWorks area. Figure 4.36 shows the initial Setup dialog box.

Setting up the Network Monitor

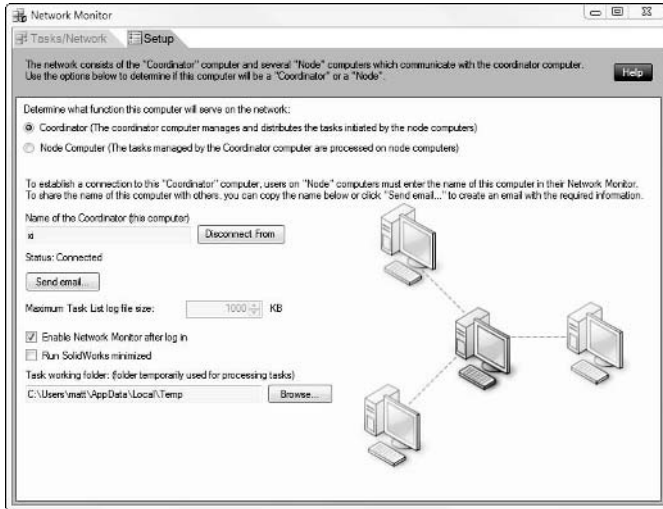
The Network Monitor requires that you choose a Coordinator computer from where all of the central commands are issued. All other computers are called Node computers. Node computers must have the software being used for the distributed task. For example, if you want to use several

Chapter 4: Operating System Tools You Need to Know

computers to update files from 2009 format to 2010 format, you can establish a group of computers with SolidWorks on them to handle this task after hours. The Coordinator computer must be established before any of the Node computers can be added to the network.

FIGURE 4.36

Accessing the Network Monitor Setup dialog box



The Send email button only sends a set of instructions to e-mail recipients to tell them how to set up their computers as Node computers. It does not send a link or a button that automatically does the set up for the recipients.

Node computers are set up in the same way as the Coordinator by using the Node option at the top of the dialog. Figure 4.37 shows that this interface is slightly different from the Coordinator setup.

The Network Monitor will only run when the Coordinator or Node machines are not busy, similar to anti-virus or search indexing.

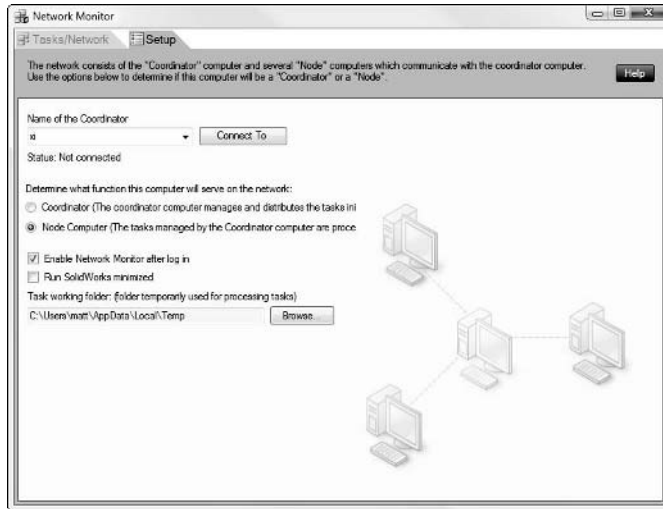


When the monitor is up and running, you will see an icon in your system tray. If the icon is gray, the monitor is inactive. If the icon shows blue screens with a red SolidWorks symbol, it is active. You may only set up tasks on the Tasks/Network tab when the network is active.

Part I: Preparing for SolidWorks

FIGURE 4.37

Connecting a Node computer to the Network Monitor



Running tasks on the Network Monitor

To access the Task/Network dialog box, click the Tasks/Network tab. In the Network Monitor, click the Task Scheduler link on the far right, as shown in Figure 4.38.

In the Task Scheduler, the only task that has the Distribute Task option is the Convert Files task. The Task Scheduler is integrated with the Network Monitor for this one purpose at this time, but it is easy to see that other tasks could be added to the list. Figure 4.39 shows the Task Scheduler window with the Convert task dialog box being edited.

Chapter 4: Operating System Tools You Need to Know

FIGURE 4.38

Initiating the Task Scheduler from the Tasks/Network tab of the Network Monitor

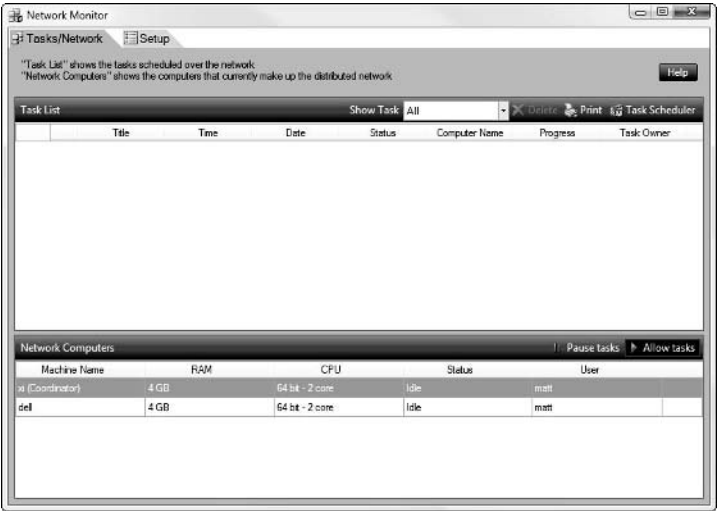
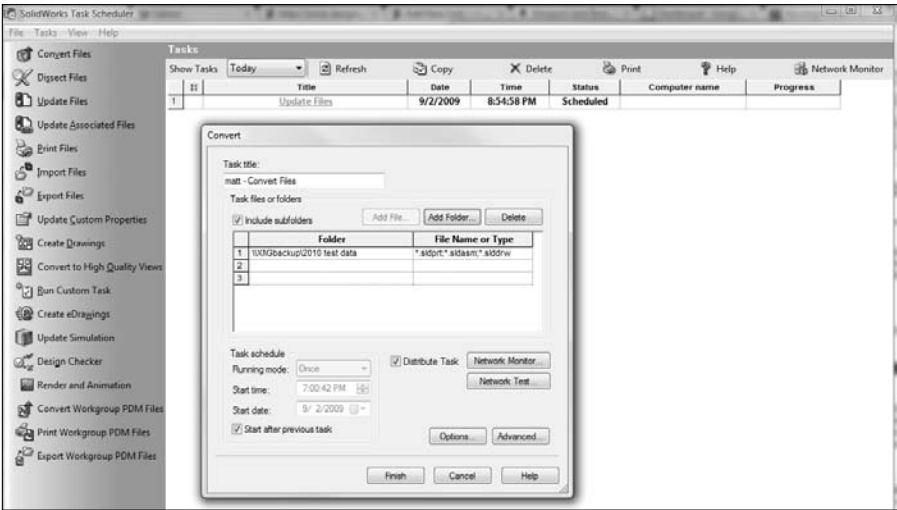


FIGURE 4.39

Editing the Convert task in the Task Scheduler



Summary

SolidWorks makes intensive use of your hardware, and it requires that your hardware and software be configured for speed and flexibility. As a SolidWorks Administrator, you may be called upon to perform some IT-type duties, and you will need to know several of the tools mentioned in this chapter. Troubleshooting hardware and knowing where to find your operating system's settings will help you greatly in the tasks you need to accomplish.

As an aid for in-depth information about settings, I recommend the TweakGuides Tweaking Companion documentation, provided free of charge for most Windows operating systems (through Vista as of this writing). These guides are more than 200 pages, in PDF format, and provide information beyond what you can find in the basic Windows Help file. You can find this information on www.tweakguides.com.

Training Users

CAD training is a topic that gets uneven attention and treatment throughout the industry. People who are focused on the short term will rarely be able to justify training, even if it is clear that a significant improvement in operations would take place. On the other hand, some CAD Administrators take an academic view of their jobs and may not be as practically focused, as they should be.

Training can be viewed by management as a money pit. First of all, if users are being trained, they are not doing work. Second, hiring a trainer is always expensive, so you're paying double to not have work done. To add insult to injury, you often wind up paying for training on topics that aren't immediately profitable to the company. Why would an injection molder want to learn sheet metal techniques in SolidWorks?

Reseller training is usually billed on a per-seat basis, with little if any discount for multiple students. Reseller training materials are written by SolidWorks Corp, and are generally highly regarded, but the classes for end users are delivered by reseller employees, which can vary from highly expert in a field to novice SolidWorks users with no industry experience.

Fortunately, positive answers exist for all these questions and concerns. Training can be targeted to only cover relevant topics, to not take up production time, and to not cost as much as reseller training on a per-seat basis. Many different solutions exist, and again, you can pick the ones that are best for your company.

As a CAD Administrator, if you can make the case for training in a specific area, in a cost-effective manner, you may be able to convince management to

IN THIS CHAPTER

Setting up a training and evaluation criteria

Recognizing and mentoring power users

Using the Web as a resource for learning

open up the purse a little bit. You may need to be open to handling the training in some possibly unconventional ways, but if the situation warrants, and you make your case correctly, you should be able to secure some training time for your CAD users.

Establishing Training and Evaluation Criteria

The first step in getting training for the CAD users you serve is to understand from where they are starting. Your job description may not include career counselor in its list of duties and expectations, but as CAD Administrator, you may need to play that role a little bit. You should be familiar with the work from each individual who uses software you support. Again, all these duties vary considerably from one company to another, but support is usually a big part of the CAD Admin job. Because of this, you will quickly come to know the people who need training most, because they will be the ones who ask you the most questions, or the ones about whom you hear the most complaints.

Think a little about what a CAD user needs to know to do his job effectively. A CAD user may also be a product design engineer, electrical engineer, packaging engineer, manufacturing engineer, tooling designer, drafter, documentation specialist, or one of many other job descriptions. As a CAD Administrator, you are probably not responsible for the other more highly specialized skills, but you may be responsible for the user's CAD capabilities. Each different type of user will approach the CAD tools differently, so you need to be prepared for this.

CAD users need most to understand how to use the tool; the special company standards for models, drawings, and other documentation; and your documentation processes. Other types of things like understanding manufacturing processes or how to choose a material for a typical application fall into the engineering realm rather than the CAD/documentation realm.

Once you know what each user is supposed to know, you might go about finding out how much each user actually knows. You can certainly talk to the users, asking them to evaluate themselves; and looking at some examples of their work will help. If your company or reseller has a training or support log, it might be useful in evaluating people's needs. If you don't have either one of those items, this might be a good opportunity to start one. The main point of this is to come up with a training recommendation, not to snoop around in other people's business. Make sure it is clear that you are doing a training evaluation.

Next, look for overlap among the users' needs. If you see or notice that most of the users have some sort of trouble with advanced mates, that information can be extremely useful. On the other hand, if your company doesn't use any plastics functionality, eliminating any training that specifically targets those areas is an area of potential savings.

Using this method, you can develop a grid of who needs what kind of training. Based on the topics, you should also be able to identify sources for training on those topics or a single resource for all the necessary training. If most of the training seems to be extremely basic, you may consider recommending the basic reseller class for some individuals. The basic class is very good at covering a wide range of information relatively quickly, while giving both classroom instruction and hands-on time. If you find that you may have to send more than a couple of people, you may want to consider a “train the trainer” scenario where you or one of your potential power users attend the class, then train other users on the basics.

Using outside trainers

With your pared-down list of topics and trainees, consider asking your reseller to quote a set of custom training sessions. If you are familiar with local SolidWorks experts who have training experience, you might also consider asking them to quote a custom class on your set of topics. Bringing in an outside trainer to train your people in your facility can be both positive and negative. It is obviously positive because it is less expensive to transport and house one consultant than five employees. It might also be beneficial because the employees will still be around the office and available to keep things running through e-mail and phone calls. On the other hand, it might be a bit of a drag, because the employees might be interrupted too frequently to learn as much as they otherwise could. A response to this is to make sure the trainer takes a 15-minute break every 90 minutes or so, to allow people to address office matters quickly and then get back into the uninterrupted training.

By considering bringing in an outside trainer, you vastly increase the options that are available to you. Outside trainers can specialize in topics important to your company that you will not be able to find in any pre-set training materials. This kind of flexibility is very valuable and may be cheaper than your reseller solutions.

Another thing to think about with outside trainers is that there is often a break-even point. Outside trainers may charge a flat fee and allow you to put as many people into the training as you can, up to a certain limit, beyond which it becomes difficult to teach effectively. I've seen that number set range from anywhere from between 6 to about 20.

When you use an outside trainer, you have to supply the computers. This is usually a source of difficulty for companies. Generally, either you will have to relocate individual computers to a central location for the training or the training simply happens lecture-style without the hands-on element.

One drawback of outside trainers is that your reseller can have the “hit-and-run” mentality, where they take your money and do your training, but you don't really have an on-going relationship with the trainers, and you don't benefit as much as you could. When dealing with outside consultants, you need to try to get involved with them in some other way that creates a link. This might be where some of the new tools in social media become important. If you link to the consultant

Part I: Preparing for SolidWorks

through, say, Twitter or Facebook, or even just engage them from time to time on the SolidWorks forums or Eng-Tips (<http://eng-tips.com>), you might find fast and easy ways to receive help while not incurring consultant fees. Consultants are usually happy to stay in touch with customers if that means there is the possibility of securing more work in the future.

Training in-house

Just as consultant-based training can be conducted at your facility, other types of training can also happen at your facility and may be lower cost than a consultant, though they will probably be less specific and less flexible. Various forms of in-house training include desk reference books, academic texts, books on specialized topics, training DVDs, or online pod casts.

Using resellers for free in-house training

Yes, you read it right. You may be able to use resellers, those most over-priced of the training options, for free, to lead in-house training. However, there are catches. The first catch is that it may not be completely free, but linked to that over-priced subscription plan that you are already paying for. If you are paying for subscriptions, and your company is big enough (read: paying enough subscriptions), you may be able to get your reseller to come to your facility and have an engineer present a four-hour What's New session. Is that a big deal? I think so. The reseller engineers are trained by SolidWorks folks on the new versions before they are publicly available. Getting this kind of a head start on understanding the new functionality in a new version of the software is very valuable.

Even if you can't get the reseller to send someone out just for you, resellers hold free meetings for every release, where they give the same sort of presentation. It may not be tailored specifically for your company, but it is an up-close, advanced look at the software. This will help you decide if you really need the software, if it has fixed any of your outstanding critical bugs, or if it is going to introduce chaos and cost you productivity and increase support calls.

Using books as learning materials

While basic publications about SolidWorks seem to be a dime a dozen, more advanced information tends to be hard to come by. With every one of my books, I have set out to serve a group that has been traditionally underserved. The SolidWorks software has matured significantly since its inception in 1995. In the early years, only basic books were needed because it matched the level of the users. But now, more than 14 years on, the maturity of the SolidWorks user base has significantly grown in depth.

For this reason, I think every office needs to have a good general desk reference on the SolidWorks software. SolidWorks has stopped producing printed manuals, and in recent years their Help documentation has suffered. As a remedy for this, I recommend my book *SolidWorks 2010 Bible* (Wiley, 2010). The first edition of this book was for SolidWorks 2007, and it has become popular with users all over the world as a critical source of information. This book is not just a tutorial that

holds your hand through a couple of specific tasks, but it is also the book that helps you answer the question, “Why does this work or not work? Why should I do it this way or that way? Why is this the way it is?” Every SolidWorks-using office needs to have this book on the shelf to answer your questions on the software.

As SolidWorks software becomes even more popular, specialty topic books are showing up in various places. A series of sheet metal books are available from “The Sheet Metal Guy.” My book on surfacing, *SolidWorks Surfacing and Complex Shape Modeling Bible* (Wiley, 2008), is the only book dedicated to SolidWorks surfacing available. Devon Sowell has self-published a book called *SolidWorks PDM*, comparing SolidWorks Explorer, Workgroup PDM, and Enterprise PDM. Mike Spens has a SolidWorks API book available for 2006 and 2009.

In addition to these desk-reference type books, you can also find several academic texts aimed at college students in college level classes covering SolidWorks. These texts are often written by people who have used multiple CAD packages, but SolidWorks may not be their primary tool. Also, these academics tend to have spent a lot of time teaching college students, and not much time in their more recent careers designing in the real world. Academic texts certainly have their place, but training in industry and academic teaching are two different things, with different goals.

Training on DVDs

Many options exist for training on DVD media or even online training. Some of this training is better quality than others. Some does not offer much beyond the basics, and some does not cover the entire SolidWorks product. The one I recommend most is Solid Professor.

Electronic format training materials have several things to love. Some of them have a great selection of topics, including specialty and niche topics that may be important to your business operation. Some also include evaluation tools to show the people paying for the materials that the investment is paying off. This makes a big difference. The proof is always in improved speed and quality of work, but exams are a great way to quantify learning before the real test.

Some of the more obvious advantages of electronic format learning materials include:

- Users can learn at their own pace.
- Users can view the materials when they have time or even during off-hours.
- Users can watch the training again and again.

Setting up CAD Administrator led training

As the CAD Administrator, you may or may not have the time and resources to put together your own training materials for your users to view with the same advantages as the training DVD and online materials. You can create PowerPoint presentation files to use as training, as well as author custom movies. Tools such as Snagit are inexpensive and enable you to capture video from your computer screen. You can use Microsoft Movie Maker (which comes with Windows and combines

Part I: Preparing for SolidWorks

the audio and video) to produce your own training videos with a total investment of about \$50. The output from this process could be anything from AVI (Audio Video Interleave) to mp4 (Movie Picture Experts Group 4) or Flash movies which can be played and delivered in a number of formats, including the iPod.

In addition to your existing employees, you might think of setting up some orientation videos to teach your standards to new hires. If you do a good enough job with producing training videos, your Human Resources department will be next to employ your skills.

These video formats tend to handle short tips nicely, say, up to 5 minutes long. It takes more skill, planning, and patience to make a longer video hold together, say, up to 15 minutes long.

As the CAD Administrator, you may also have enough experience to lead a live class. If you get to this point, you are probably leading a class on custom topics meant especially for your people.

The CAD Administrator is also in a unique position to conduct two other economical kinds of training. One is “Lunch and Learn,” a popular term for brief training sessions held in your office during lunch. Often companies buy their employees lunch in return for donating their free time during lunch to learn about work-related tools. This is a fantastic method to help people learn, without overwhelming them with a whole day’s worth of material. It allows people to be informal and social while still learning useful information.

The other type of training comes from another popular term, “Train the Trainer.” In this type of training, the company sends one or two people to a paid reseller training class, and then those people come back and teach the rest of the company what they learned in class. It’s good if your list of CAD users is long enough to justify sending two people to training. Often one person will pick up what the other missed. Training materials may be difficult to come by in a situation like this, but the quality of the training (as long as the people who go to the initial training take careful notes and ask a lot of questions) and the savings can be difficult to beat.

Mentoring Power Users

In an earlier chapter, I discussed power users, but I will talk about it again here from a different point of view. The bigger your community of CAD users, the more you will need power users. As a single person, the CAD Administrator cannot answer everybody’s technical questions. A SolidWorks subscription provides your users with direct access to the reseller support engineers, but many companies have a policy to ask questions internally before resorting to outside help.

Depending on the skill level of your users and the difficulty of the modeling tasks they undertake, one skilled power user should be able to support approximately ten users. So if you have a large installation of SolidWorks, you need to depend on power users to help you keep the rest of the users productive, and it becomes imperative for you to encourage and keep the power users going. They are extensions of you.

Recognizing the type

Not every advanced user is destined to become a power user. The kind of user you want to encourage in that direction is not only gifted with technical skills in computers and software, but also geometry, math, engineering, and manufacturing. This user must also have some people skills, obviously. Potential power users typically help other users because of a natural aptitude with the tools and a desire to help. To some extent, you can't miss the type, because they truly excel at what they do and they love doing it and helping others. It's a terrific combination.

Mentoring your protégé

Once you identify someone with the skills to groom as a potential protégé, she will need some attention one-on-one and some encouragement in group settings. You cannot spend time with her at the exclusion of all other users, but you should do what you can to feed her advanced information and offer learning opportunities, such as user group meetings, trips to SolidWorks World, and links to Web sites and forums where skills can be developed and recognized.

Identifying Sources of Learning

In this era of infinite online social networks, you don't need a big budget to set up a networking site, and let's face it, everything is happening online. All you need is a little Web savvy and a good vision of where you want to go. Many of the tools are free, and many people with high-level knowledge are willing to share it, which is evident by the number of forums, blogs, and social media outlets that has simply exploded on the scene. Some of these online sites are run by professionals, some semi-pro, and some amateur.

In some situations, the word "amateur" is not considered a positive thing, but in this case, the only aspect about some of these sites that is "amateur" is likely to be the administration, the lack of glossy front pages, and advertising. The content you find on many of these sites can be absolutely top-notch. Sites that are backed by more money only have better infrastructure, not better content. The fact that amateurs can thrive is bad news for big ticket outlets, but good news for people searching for information.

Google search is your gateway to many learning experiences. You have to have a plan if Google is going to be a training aid. You must first have a tolerance for a lot of small pieces of information coming from different locations. Centralization is a convenience, but when information is centralized, you wind up paying for it, and someone has a lot of control. I recommend that you keep a folder of links in your browser of specifically the ones to great information. Know where your links are stored so you can back up the data and send the links to other people. Better yet, keep a blog yourself full of links to your favorite information sources.

Using online forums

Online forums are proliferating. You simply can't make the rounds to each of them every day. To get the most out of a forum, you have to spend time there and become a regular. If you are in a position to suggest online activities to several people, you might think about making a list of forums and dividing them among several users so that each user spends enough time with each forum to get to know it, then make a selection based on the quality and quantity of the discussions.

Getting the most out of the forums

Whether you are an administrator or an end user, to get the most out of online forums, you must follow some simple commonsense rules. In general, people forget that courtesy is courtesy, whether you are online and no one can see your face or you're in church on Sunday morning. The rules are as follows:

1. Always give as much detail as you can about any problem you are asking about.
2. Allow for the possibility that you may be mistaken in some assumptions (such as "this is a bug").
3. Remember that people on the other side of your questions do not have a crystal ball. Screen shots are often helpful, as are uploaded models and accurate descriptions with correct terminology. The better your question, the more complete and accurate and useful the answer.
4. Always, search for an answer before you ask a question, especially if you are new to the topic and the question might seem obvious.
5. If you are expecting people to answer your questions, don't be insulting. This seems like it should be obvious, but believe me, people need to be reminded.
6. Thank people when you get an answer, even if it isn't perfect.
7. If you can, try to answer questions to give something back to the community.
8. Humor, and sarcasm in particular, does not translate well in writing. Judicious use of smilies might make all the difference between a nasty growl and an empathetic response.

Forums can be great places to develop friendships and business relationships alike, as well as find answers to your questions that are beyond the textbook knowledge of reseller support people. For every regular participant in forum discussions, most forums have several lurkers, or folks who only read and never contribute. This is okay, but remember the more people that contribute, the stronger the overall forum, and a few people can have a great influence on a forum by asking interesting questions and giving answers that come from experience or even speculating a little bit from time to time.

I have known many people who glean pearls of wisdom from various forums and then keep a notebook of some type linking back to the source and any supporting data.

Addressing confidentiality

One of the drawbacks about online forums is that posting real models to get better answers usually conflicts with the confidentiality of company data. Trying to describe a model with words is usually a losing battle. It is much faster and more accurate to send an actual part. One thing you can do is to simplify the part, showing the same modeling situation on an incomplete part, or you could cut out a section of the model and export it if you just need to show a shape without feature data.

Of course, the alternative to taking measures like that is to set up a secure forum for only people from your company. This really isn't all that effective unless your company is very large and you have many active users.

Another option for security conscious people is to post under a pseudonym. You still get the same benefits, but with lower risk to yourself, your company, and your company's data. Some companies may not allow their employees to participate at all on forums. If you are in that situation, see if you and your IT department can come to some sort of compromise. You simply cannot ignore the wealth of information that awaits you online. It is also hard to account for the pleasure that people seem to take in answering other people's questions. The range of questions can be incredible, from the most complex SolidWorks modeling questions to engineering calculations and material selection.

Getting value from live user groups

Live user groups are an important source of information, but I believe that in the not-so-distant future they will begin to diminish in influence. I don't think they will ever completely disappear, but with all the online activity available, the pull of live groups is going to hit a plateau. Live groups are comparatively expensive to administer, and if you don't make it on the night of the meeting, you've missed it.

Online "user groups" are usually called forums, and they have a complete transcript of the 24/7 meeting online accessible to all, all the time.

Live user groups have continued to be popular because of face-to-face interaction. There is a segment of the SolidWorks population that feels more comfortable in the online setting. Live user groups meet either monthly or every other month, generally are given food paid for by SolidWorks, and have a SolidWorks technical presentation, along with other types of events, including guest speakers, plant tours, giveaways, panel discussions.

On the other hand, you could view live user groups as not very time efficient. You don't set the schedule yourself, and you can't just fast-forward through sections that don't interest you. In areas subject to bad weather, you have to brave the elements. The pace of live user groups tends to be slower than online groups because the meetings only happen, at most, once a month.

Summary

SolidWorks corporate training materials are generally well done, but the delivery is variable. People learn in many different ways, so there are many options available for training SolidWorks users beyond the traditional reseller classes. In fact, the most advanced training available usually comes through subject matter experts rather than through the corporate channel. You can find books, online classes, and even customized expert consultant training.

Non-traditional sources of training also exist such as live user groups, SolidWorks World, blogs, on-line forums, power user mentoring, and others. As a SolidWorks Administrator, you may be responsible for conducting the training yourself, or bringing in an appropriate organization to deliver it for you.

Part II

Installing SolidWorks

Installing SolidWorks is much more than just throwing a disk in the computer and clicking Next several times. You will have to deal with issues such as licensing, computer hardware and operating system questions, detail setup or optimization of SolidWorks including implementation of Toolbox and libraries. None of this is something you can get right the first time off the top of your head, and this book has been written from the experience of people who have both succeeded and failed, and you can learn from both experiences.

IN THIS PART

Chapter 6
Installing the Software

Chapter 7
Licensing

Chapter 8
Standardizing Settings

Chapter 9
Maximizing the Use of
Templates

Chapter 10
Implementing Toolbox

Chapter 11
Establishing Libraries

Installing the Software

Installing the SolidWorks software properly is critically important. Many frustrating user issues are caused by faulty installs. Some general rules of thumb to keep in mind before installing the software include:

- **Exit out of all running software, and pay particular attention to antivirus software.** Programs may be running that lock up system files that SolidWorks needs access to in order to run properly. Antivirus applications have been known to block SolidWorks from completing some tasks that it needs to complete during the course of an installation.
- **Install from an account with administrator access on the local computer.** When people install from accounts with less than administrator access, they often find they do not have sufficient rights for the software to make necessary changes to the Windows registry.
- **Make sure the machine where you install SolidWorks has not been abused from a “computer hygiene” point of view.** The best installation is a fresh operating system on a freshly formatted drive. So much depends on computer hygiene: many reported cases of frequent crashing stem primarily from basic system maintenance problems. Avoid installing a lot of junk applications on any computer you are going to use to run SolidWorks or other computer-intensive professional applications.
- **Make sure you have plenty of space for SolidWorks and any add-ins you may be using.** Toolbox takes up a lot of room. An installation of SolidWorks 2009 Premium can take up more than 4GB of disk space, and it may not be all added to the same folder. For example, SolidWorks puts information in Program Files,

IN THIS CHAPTER

Choosing an installation method

Installing multiple versions of SolidWorks

Installing floating license manager software

Removing SolidWorks software

Part II: Installing SolidWorks

Program Data, SolidWorks Data, and possibly other custom-named folders that you have specified for Toolbox and other add-ins. I recommend placing the main SolidWorks installation along with all ancillary data on a drive other than the system drive.

Successful installs require attention to detail and a high level of computer hygiene. Problems that later turn out to be nearly impossible to troubleshoot often stem from sloppy practices and a cavalier attitude toward the software installed. While you “should” be able to put whatever kind of software on your computer that you want, software developers are not always aware or vigilant about the different kinds of conflicts their software can create. As long as SolidWorks is mission critical software at your organization, you need to treat your computer that runs SolidWorks with a great deal of care.

SolidWorks is a complex piece of software composed of many separate parts of independently maintained software. It also has several prerequisites needed on an operating system basis. Particularly if you must install one of the advanced levels (SolidWorks Professional or SolidWorks Premium), the installation can take a significant amount of time and hard drive space. Depending on what you are installing and from what type of media you are installing, installation can take anywhere from 10 minutes to 30 minutes, using 2GB to 8GB of drive space. This includes the libraries and files for Toolbox but does not include the SolidWorks Workgroup PDM (Product Data Management; formerly PDMWorks Workgroup) vault files.

The installation process for SolidWorks 2010 has changed from that of SolidWorks 2009. The 2010 version is simpler, cleaner, and less confusing; and it includes a workflow legend on the left-hand side to show you where you are in the installation process.

You do need to be aware that different installation files exist for 32-bit and 64-bit installations. If you have a 64-bit operating system installed, you need to use the 64-bit version of SolidWorks. This is true for the full version of the software as well as service packs and add-ins. These are not interchangeable, and the wrong version will not install correctly. If you have the option, I strongly recommend going 64 bit, because it is the new standard going forward and it offers more memory. All new hardware is 64-bit compatible, but you need to be careful about software and operating systems.

Note

Starting with SolidWorks 2010, you can no longer have parallel installations of the same major release of SolidWorks using different service packs, also known as SP. That is, you can still have SolidWorks 2009 and 2010 installed and even running at the same time, but you cannot have SolidWorks 2009 SP3 and SP4 both installed. ■

Selecting an Installation Method

Several methods exist by which you can install the SolidWorks software, and each has its benefits and conveniences. Some methods take more time up front but save time in the long run. Choose a method that best suits your needs.

When you initially install SolidWorks, you have a choice between several methods:

- Install SolidWorks from DVD
- Copy the DVD to your hard drive and then install SolidWorks from your hard drive (local or network location)
- Download the full version from the SolidWorks Web site
- Download individual installation files and install them manually
- Install from Administrative Image

The most straightforward approach is installing from the DVD. The main difference between the direct from DVD and from copied DVD methods is that when you install directly from the DVD and you want to install a service pack, you have to have the DVD in the drive to install the service pack. If the DVD is copied to the hard drive, you can reference the DVD data from the drive, but you need to keep access to that copied data (don't delete it). When the service pack starts looking for the original installation media, it will find it on the drive. This works from either local or network drives.

Introducing the SolidWorks Installation Manager

All types of initial installation require that you use the SolidWorks Installation Manager (SWIM). The SWIM is a separate program that manages the installation of the various components of the SolidWorks software. When you install from the DVD, the Installation Manager is the program that automatically runs. The SWIM also helps manage service pack installation. For every service pack you install, you need a new version of the SWIM. For SolidWorks 2009, the SWIMs are about 20MB. For SolidWorks 2010, the SWIM download is about 18MB and unpacks to about 43MB. (This is for the beta 1 version, which is available at the time of this writing.) The downloadable file is called SolidWorksSetup.exe, but regardless of the name, rest assured that this is still the SolidWorks Installation Manager (SWIM).

The SWIM determines if your system needs some or all of the prerequisite system components in the installation list, such as redistributables and debuggers. SolidWorks strongly recommends that you use the SWIM instead of downloading components and installing manually, because the SWIM can save you from downloading many extra data you don't need.

Note

Installing service packs and beta versions work differently from installing a complete version of the software. Beta versions are usually installed from downloaded files, and service packs are usually installed using the SWIM, which downloads files for the install. ■

Installing SolidWorks from downloaded data

Sometimes, it is most convenient to install the software from downloaded data. Sometimes this is the only way, such as during beta, prerelease, and early release periods of the release cycle. During

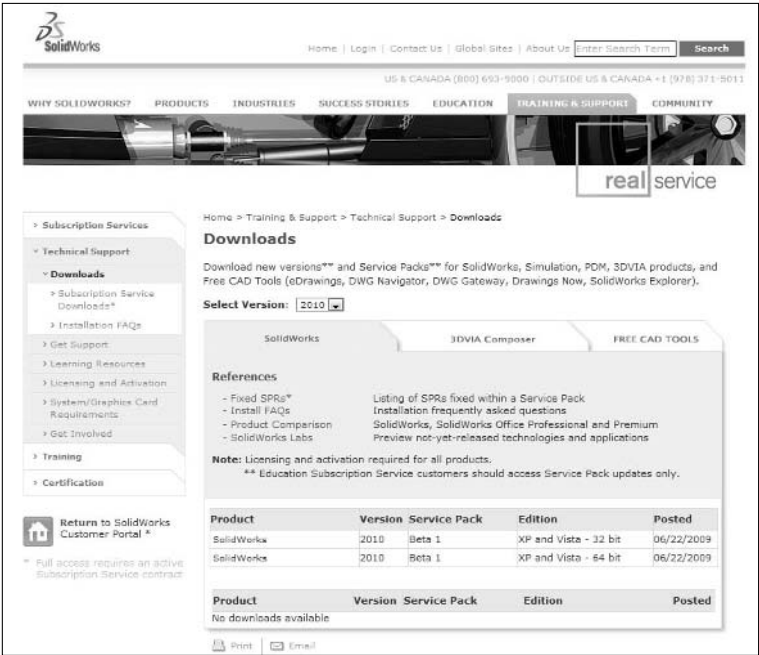
Part II: Installing SolidWorks

these and other times when you simply do not have an installation DVD handy, the download provides a great convenience. The download size, however, may be very large (multiple gigabytes, as many as 5 or 6GB).

Here I walk you through an installation from downloaded data. With SolidWorks 2010, you can install the software from a complete file set download for that release throughout the life cycle of the software.

To download data from which to install SolidWorks, start from the Customer Portal. You need to be on subscription to do this. How about this text instead: From the `SolidWorks.com` home page, click the Training & Support tab, and then click the Downloads and Updates link in the Quick Links section. You then see the page shown in Figure 6.1. Make sure you set the Select Version drop-down to the correct version, and select the correct tab. Figure 6.1 shows three tabs: SolidWorks, 3DVIEW Composer, and Free CAD Tools. Be sure to select the correct 32- or 64-bit version for your needs. If you need both (you are running SolidWorks on both 32- and 64-bit operating systems), you need to download both versions.

FIGURE 6.1
Download installation files from the SolidWorks Customer Portal.



Note

The screen images here show only a choice of XP and Vista operating systems, but this installation also works on Windows 7 RC1 (Release Candidate 1), which is the latest Windows operating system available as of this writing. All projections are that Windows 7 will not require a different install than Vista, or XP for that matter. ■

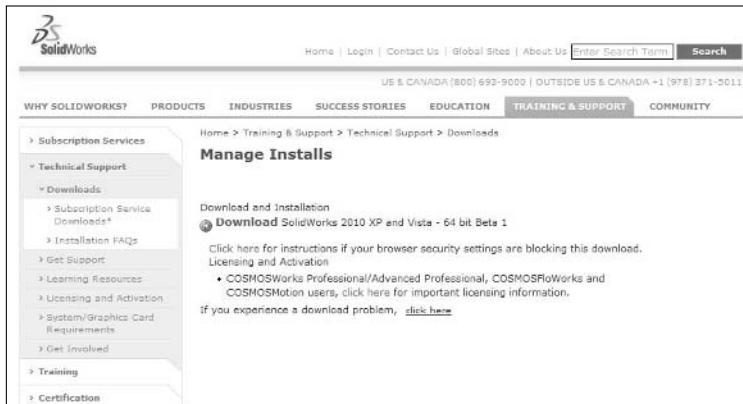
After clicking through the End User License Agreement (EULA), you come to the Manage Installs page, which prominently offers a link from which to download the selected version. However, you are not actually downloading the installation data directly: you are first downloading the SWIM, which, in turn, separately downloads the individual installation files required for the level of software you are licensed to run.

Next, the Download link, shown in Figure 6.2, downloads then installs and runs the SWIM, for which you may need to click through some confirmation boxes, depending on your settings for the UAC (User Account Control in Vista and Windows 7).

If you have difficulty with the SWIM and are unable to complete the installation, you can download the SolidWorks installation data directly from the click here link at the bottom of the screen.

FIGURE 6.2

Download the SWIM or manually download the files you need.



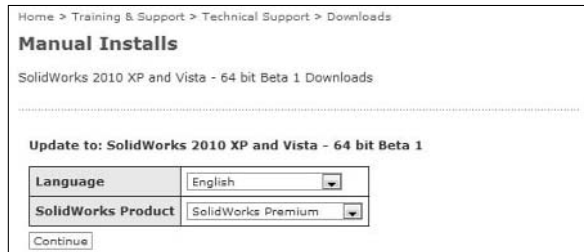
Installing SolidWorks manually

To install SolidWorks manually, click the click here link shown in Figure 6.2. The Manual Install page, shown in Figure 6.3, appears, displaying language and software level options.

Part II: Installing SolidWorks

FIGURE 6.3

Download and install individual components of SolidWorks.



SolidWorks tries to encourage users to use the automated SWIM instead of just downloading and installing files manually. When you install manually, you probably do not know which if any of the prerequisite files need to be installed on your computer. The SWIM figures this out automatically. If you choose to download and install manually, you have decisions to make regarding which files to download and install for each of the following headings:

- Installation and download instructions
- A list of functions included in the various levels of the SolidWorks software
- Prerequisite downloads that must be installed before SolidWorks can run properly on your machine
- Updates for core SolidWorks
- Updates for optional software (such as DWGeditor, eDrawings, SolidWorks Explorer, and PhotoView 360)
- Optional updates (for various software development kits, CosmosM, SolidWorks Flow Simulation, SolidNetWork License Manager, SolidWorks Workgroup PDM server, and the Activation Wizard)
- SWIM (you may be trying to avoid it, but SolidWorks gives you many options to still get it)
- Language packs (required for Administrative Images in languages other than English)

Figure 6.4 shows only the initial part of the download list. Downloading all this manually, deciding what you need, and then installing it all manually is a bother with which you don't need to deal. The SWIM in SolidWorks 2010 seems to be greatly improved over past versions.

Note

When you download files from SolidWorks, rarely are the files named in such a way that you can tell which version they are for. For example, the 2010 SWIM is named `SolidWorksSetup.exe`, and all the service pack versions will likely have the same name, in addition to the 2011 SWIM. For this reason, you may want to rename them or place them in named folders at the time of download. Otherwise you need to run the file to be able to tell what version you have. ■

FIGURE 6.4

The Download Packages page presents you with many download decisions.

Home > Training & Support > Technical Support > Downloads

Download Packages

Step 1 - Read and review and following instructions and notes:

- Read the **Installation and download instructions**.
- To find what is included with the different SolidWorks packages, [click here](#).

Step 2 - Required Prerequisite Downloads.

| Product | Size | Notes |
|---|-----------|-------|
| Microsoft C++ Runtime | 2.56 MB | |
| Microsoft C++ 2008 Runtime x64 | 3.01 MB | |
| Visual Studio Tools for Applications | 47.86 MB | |
| Microsoft Office Web Components 11 | 17.33 MB | |
| Windows Search (Windows XP/2003 64-bit) | 8.20 MB | |
| Microsoft Visual Studio Debugger | 3.44 MB | |
| Microsoft .Net Framework 3.5 | 231.44 MB | |
| Microsoft C++ 2009 Runtime | 4.03 MB | |
| Microsoft C++ 2008 Runtime x64 | 4.74 MB | |

Step 3 - Required update for core SolidWorks.

| Product | Size | Notes |
|------------|------------|-------|
| SolidWorks | 1666.13 MB | |

Installing using the SWIM from downloaded data

Sometimes you simply need to download manually because the automatic download doesn't work for whatever reason. You can still use the SWIM after a manual download. In these cases, it is best to have downloaded all of the available options to make sure that the SWIM has what it needs for the installation. For the SWIM method, click the Download link shown in Figure 6.2. From there, you download the SolidWorksSetup.exe (SolidWorks Installation Manager), and extract it.

The first step in using the SWIM to install SolidWorks is to select which type of installation you want to perform. Figure 6.5 shows the SWIM for SolidWorks 2010 with the available options.

The options are

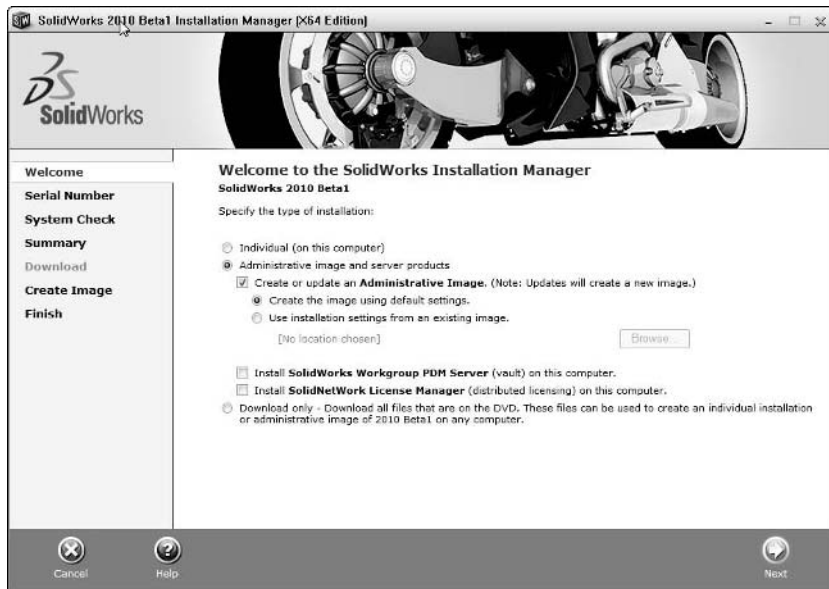
- Individual
- Administrative image and server products
- Download only — download all files that are on the DVD

Network (also known as SNL [SolidNetWork License], floating, or FlexLM) licenses are determined by your serial number, which comes up in the next screen, and are handled differently in SolidWorks 2010 than in previous versions.

Part II: Installing SolidWorks

FIGURE 6.5

The SWIM Welcome screen for SolidWorks 2010 includes a new workflow legend on the left.



Note

Before SolidWorks 2010, network licenses were handled with a physical dongle (USB or parallel port) on the license server. With SolidWorks 2010 and later, network licenses use the online activation process just like other types of licenses. ■

The next step is to enter your 24-digit serial number. From this number, SolidWorks can tell your license type and to what software you are entitled, for example, SolidWorks Standard, Professional, or Premium.

Note

If you are creating an Administrative Image, serial numbers are entered differently to allow for various options, such as specific serial numbers on specific machines, network licenses, and so on. I cover Administrative Image installation in more detail later in this chapter. ■

SolidWorks can also tell the difference between stand-alone and network licenses. Once it determines what you are entitled to, it knows what to download and install, and how to license it. Some longtime customers may have additional serial numbers for other add-ins, which are licensed separately from SolidWorks.

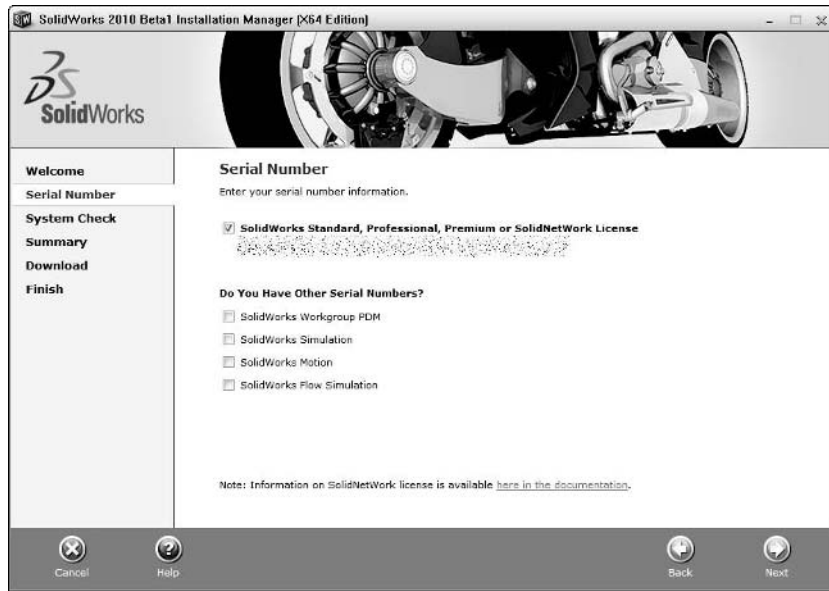
Cross-Ref

Read more about licensing, including international and network licenses in Chapter 7. ■

In Figure 6.6, I have blurred out the serial number because the screen capture shows the serial number for my license of SolidWorks.

FIGURE 6.6

Entering your SolidWorks serial number and serial numbers for separately purchased add-ins



If you have purchased SolidWorks add-ins separately from the packaged licenses (Professional and Premium), you can select one of the lower boxes to enter in those serial numbers separately. Be aware that in early (pre-sp0) versions of SolidWorks 2010, these separate serial numbers cause some problems. To correct the problem, install sp0 or later, or contact your reseller for technical assistance.

Caution

You need to keep your serial number information private. If this information gets into the wrong hands, other people can potentially use a license that you are paying for. The serial number is displayed in the Help ➞ About menu, and there is no way to prevent your users from seeing what serial number they are using, but you might want to be aware of this if you have guests in your facility using your SolidWorks software. ■

Part II: Installing SolidWorks

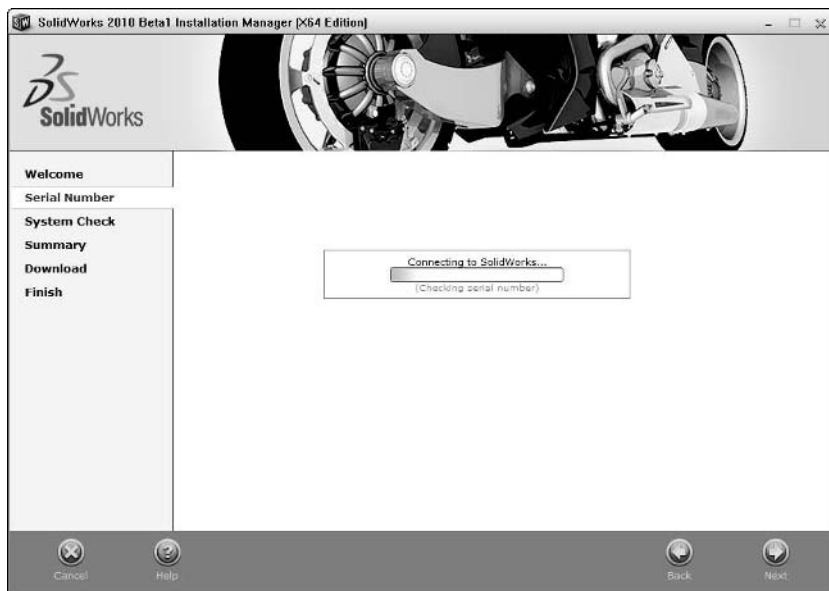
When you enter your serial number, you see a series of six boxes where you must enter four alphanumeric characters in each box for a total of 24. This 24-character serial number is on the original box that you got from SolidWorks when you bought the software.

From here, I assume you are performing an installation from downloaded data, but most of these steps are identical to an installation from copied or DVD data and the same as the steps for the Administrative Image creation and installation. Refer to each of the specialized topics in this chapter for more detailed information on those types of installation.

After you have entered your serial number, SolidWorks Installation Manager enters the System Check part of the installation workflow and checks your serial number against the on-line server to see to what level of the SolidWorks software you are entitled. Figure 6.7 shows this connection.

FIGURE 6.7

SWIM connecting to the on-line database



Now that the system knows what you are entitled to, it gives you options for what to install. These options may also include which files to download, if you are not installing from copied or DVD media.

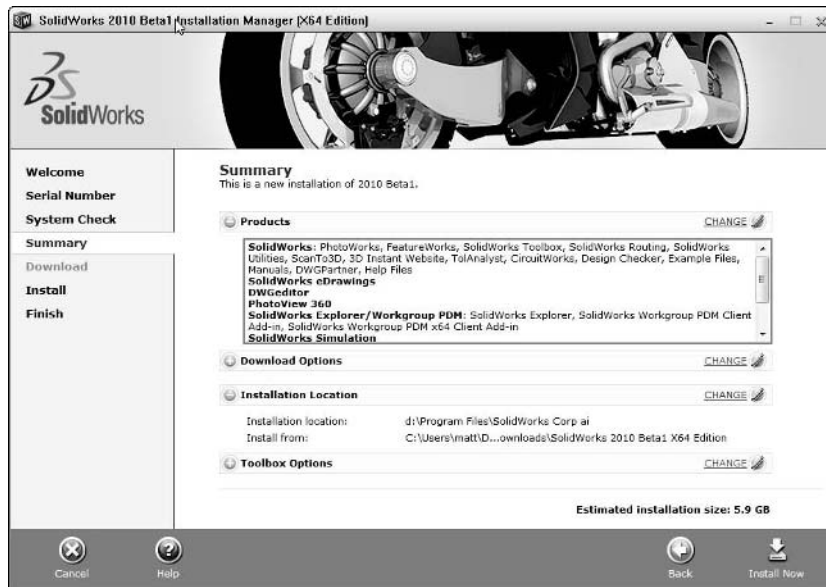
Note

If the computer on which you are installing SolidWorks does not have an active Internet connection at the time you are installing the software, the SWIM issues a warning saying that you can continue with the installation if you have a local source for the installation files (meaning that you have already downloaded them). ■

The Summary section of the installation workflow offers you options for changing or accepting the defaults for installation. This is a major point of difference between the SolidWorks 2009 and 2010 installations. Figure 6.8 shows that you can change the options for the products to install, the download location, if any, the installation location, and Toolbox installation options.

FIGURE 6.8

Summary of installation information enables you to change installation settings



Clicking the Change link at the upper-right corner of the Products area displays the set of options for what you are eligible to install. You may deselect options if you want to, but you cannot add software to which your serial number does not entitle you.

Note

If you find that you cannot install software that you think you should be entitled to, the best way to address this is through your SolidWorks reseller, who should be able to explain or fix the situation. ■

Figures 6.9 through Figure 6.12 show the change options for the Product Selection, Download Options, Installation Location, and Toolbox Options boxes in the Summary step of installation. To access each of these areas, click the Change link to the right of the title bar shown in Figure 6.8.

Part II: Installing SolidWorks

FIGURE 6.9

Product options

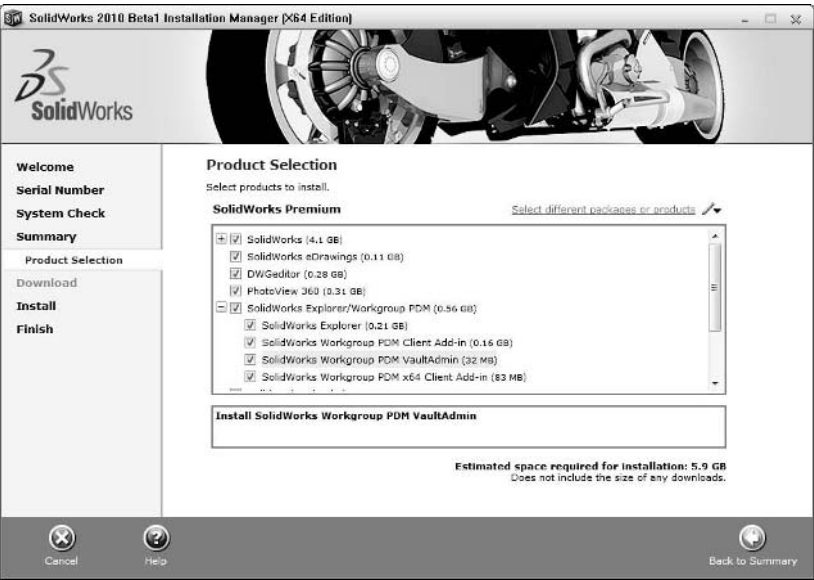


FIGURE 6.10

Download options

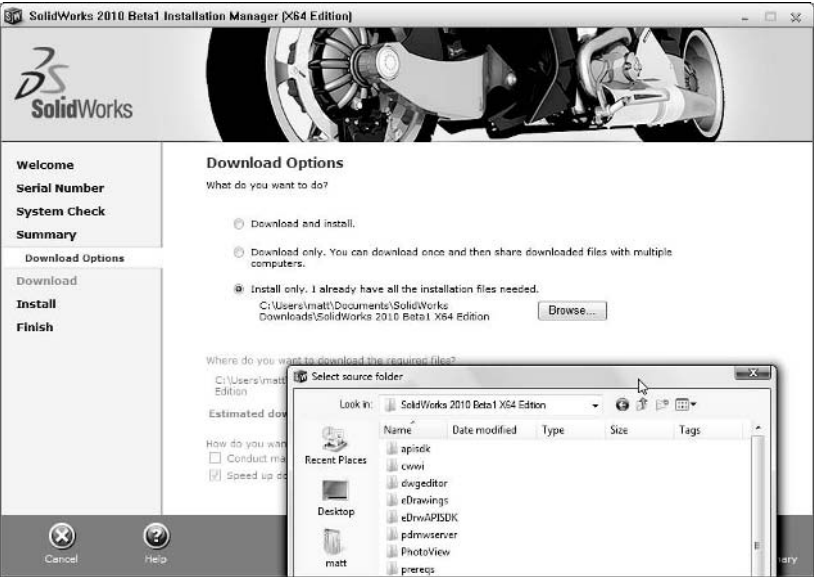


FIGURE 6.11

Installation location options

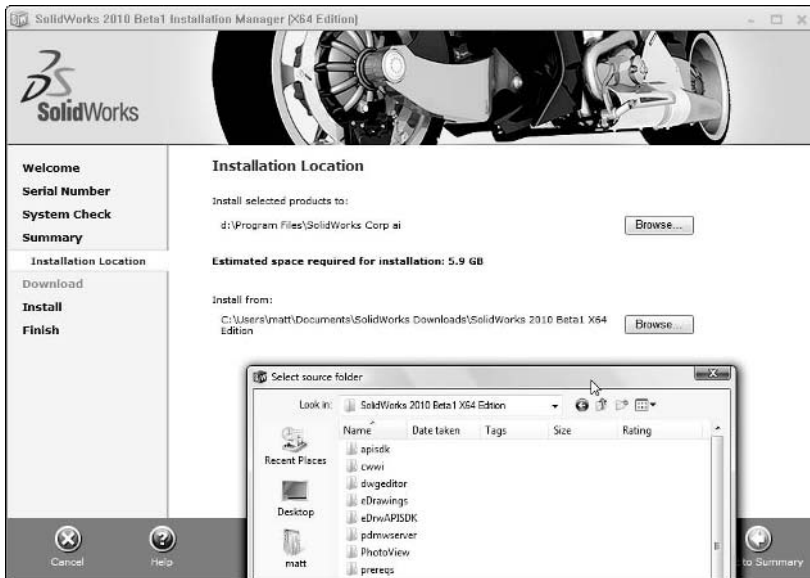
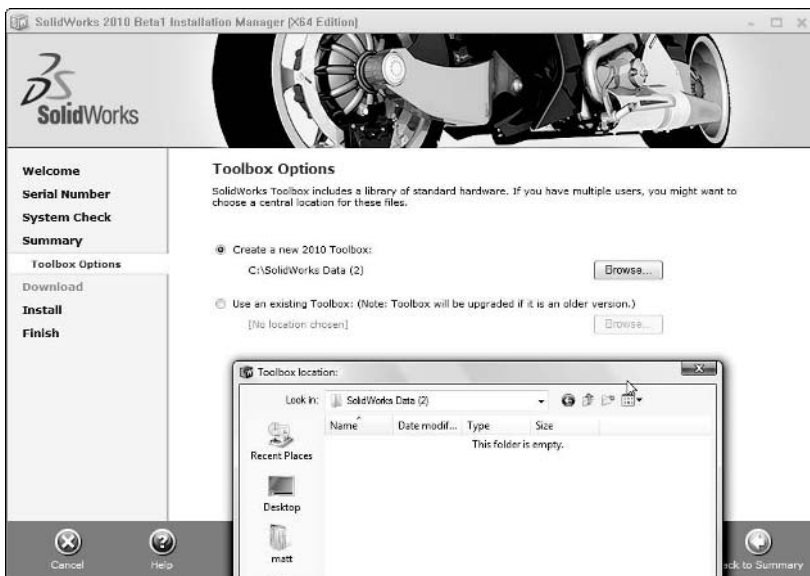


FIGURE 6.12

Toolbox options



Part II: Installing SolidWorks

When you establish the locations for data in these options, you need to keep a few things in mind:

- **If you need to return to these options after the installation is complete, some will make sense to change and others won't.** For example, you can change the list of installed software, and the SWIM either adds or removes software to accommodate, but changing the download or install directories is best accomplished by either moving or uninstalling and reinstalling.
- **Pay attention to warnings that the SWIM gives you — in particular, disk space warnings.** The files involved here are very large and can take up a lot of space on a hard drive. Disk space is relatively inexpensive these days, and your computer likely has some available connections for an additional internal or external hard drive.
- **Toolbox installation can be critical to any existing assemblies you have that contain configured Toolbox parts.** If you overwrite an older version of Toolbox, make sure that you don't want to use it again, because older versions of SolidWorks are not able to use the newer files. In addition, if you overwrite an older Toolbox library with a fresh library, your parts will be missing many configurations. At the same time, installing a fresh library in a new location can cause the same problems.

Cross-Ref

For details on implementing Toolbox effectively, see Chapter 10. ■

From here, the installation should proceed until it completes, unless for some reason your downloaded files are not correct. Figure 6.13 shows the installation phase of the software proceeding normally.

Installing from the DVD

When you insert the SolidWorks installation DVD into the drive, the SWIM starts up automatically, depending on your operating system autoplay settings. The main difference between the DVD and downloaded data installations is that the download options are not available. Remember that SolidWorks tries to access the Web when installing to pick up the software eligibility from your serial number, and then again after installation when running you through the activation wizard.

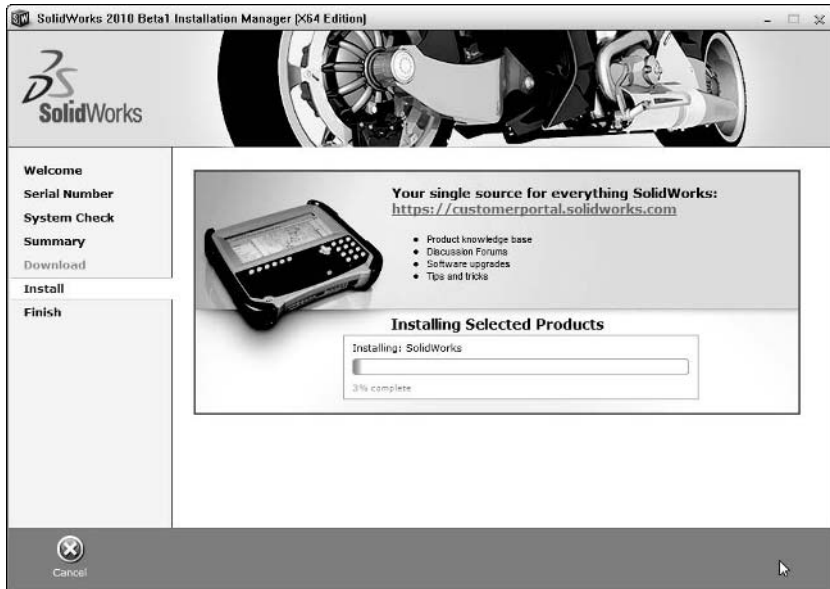
Cross-Ref

I cover installing service packs in depth in Chapter 19. ■

When you install from the DVD, downloaded data or data copied from the DVD, and then install a service pack, the service pack does not require the original installation media to be accessible as SolidWorks has in past releases.

FIGURE 6.13

The installation phase



Installing from an Administrative Image

An *Administrative Image* is a set of files used to create an installation of SolidWorks on multiple computers such that you don't have to set all the options multiple times. It is meant to be a means of automating multiple installs. You create the Administrative Image by essentially going through the same steps as a regular install to create the image on a network location, but then send the users a link to build the install locally on their machines. In this way, the Administrator can install software to many computers without physically going around to each one individually.

Service packs are deployed to all users by updating the Administrative Image, and redeploying it. The basic Administrative Image technique is straightforward and does not require extensive IT skills, but to make use of more advanced options, you need skills that are more advanced.

The basic steps for deploying SolidWorks installations with the Administrative Image technique are:

1. Identify a network location to store the Administrative Image.
2. Create the Administrative Image by following the steps in the SWIM.
3. Send users the *.hta link to the Administrative Image.
4. When users activate the link, the Administrative Image installs SolidWorks and related software with the options established in the Administrative Image.

Part II: Installing SolidWorks

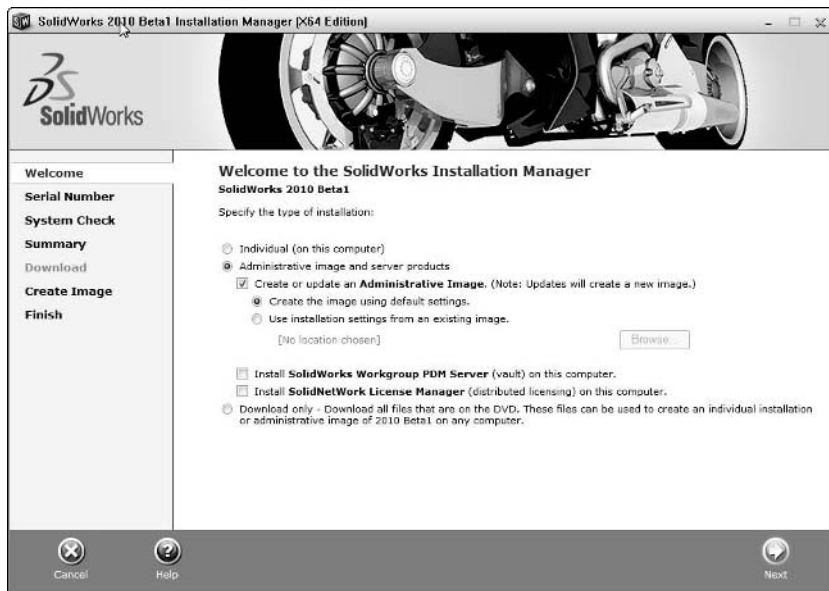
The Administrative Image installation can be used for software licensed as stand-alone or floating (SNL, network) licenses.

Welcome to SWIM

The SWIM functions the same for the initial screen regardless of the installation method that you choose. SWIM acts as an installation wizard, allowing you to check off the options, and it takes care of the details behind the scenes. Figure 6.14 shows the SWIM when it's used to install an Administrative Image.

FIGURE 6.14

The SWIM allows you to select the type of installation you want to perform.



The settings for an Administrative Image are established after it is installed, or can be copied from an existing image. You might copy the settings when creating an image to deploy a service pack, for example. You learn how to set Administrative Image settings later in this section.

From here, the steps from entering your serial number to setting the software to be installed, and even the download options shown earlier in Figures 6.6 through Figure 6.13, are the same as when you install from downloaded or DVD data.

Using the Administrative Image Option Editor

After the image is created, an option to open the Administrative Image Option Editor interface appears. Figure 6.15 shows the screen that is displayed after the Administrative Image is finished.

FIGURE 6.15

Access the Option Editor to set up Administrative Image installation options.



If you do not run the Option Editor immediately after creating the Administrative Image, you can start it later by double-clicking the StartOptionEditor.hta file in the folder in which you created the Administrative Image, or sldadminoptioneditor.exe in the [admin image]/sldim/ folder.

Figure 6.16 shows the Serial Numbers tab of the Option Editor.

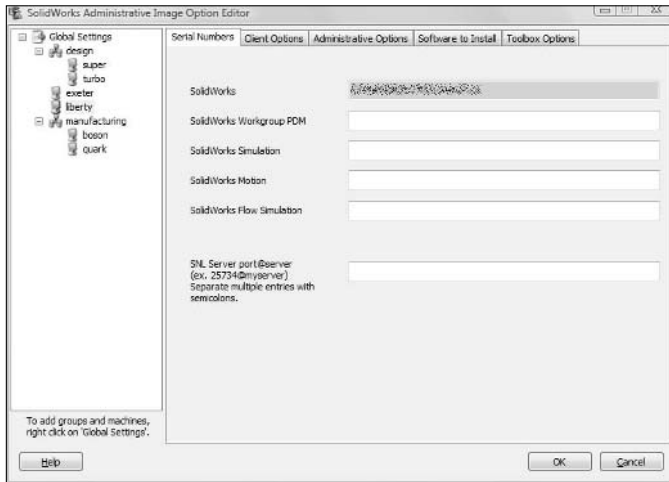
The additional fields underneath the SolidWorks serial number entry, for the add-ins SolidWorksWorkgroup PDM, Simulation, and so forth are for customers who have purchased those specific add-ins separately from the packaged bundles Professional and Premium.

The panel on the left in Figure 6.16 shows only the Global Settings text by default. You can use this area to list computer names or to create groups of computers. In Figure 6.16, Design and Manufacturing are groups. Super and Turbo are computers in the Design group, and Boson and Quark are computers in the Manufacturing group. Exeter and Liberty are individual computers listed without groups. Setting up serial numbers for individual computers, or a SolidNetWork license server port, makes it easier to use the Administrative Image installation for multiple kinds of licenses.

Part II: Installing SolidWorks

FIGURE 6.16

SolidWorks Administrative Image Option Editor enables you to set up different serial numbers for different machines.



Note

The computer name being referred to here is the one listed in the System information in the Control Panel and is how individual computers are identified on the network. ■

When you have a group or machine using a global serial number, the SolidWorks serial number field is white or gray. When you are using a special serial number (anything other than the global default) for a computer or group, the serial number field is yellow. This scheme seems to hold true in the rest of the Option Editor interface: Whenever you select a non-default option, it is highlighted in yellow.

If you right-click on Global Settings, you get the options Add Group, Add Machine, and Delete All. The SolidWorks Help documentation on the topic of the Option Editor explains how to do specific tasks in the Option Editor, but it stops short on telling you why you would want to do each of those things. If you want the same settings to apply to all installations, do not make any additions in the Global Settings list. You might consider listing only the machines that have serial numbers or other settings that differ from the one used for Global Settings. The second tab in the Option Editor is called Client Options and is shown in Figure 6.17.

FIGURE 6.17

The Option Editor Client Options tab enables you to set options for the client install on each machine.

The screenshot shows the 'Option Editor' window with the 'Client Options' tab selected. The window has a tabbed interface with 'Serial Numbers', 'Client Options', 'Administrative Options', 'Software to Install', and 'Toolbox Options'. The 'Client Options' tab contains three sections of configuration options:

- How do you want to perform client installations of major versions?** Note: Both options will update the installation when applying a service pack.
 - ☒ Upgrade an existing major version (if one exists).
 - ☐ Create a new major version.
- Location for new installation (if needed):**
 - Text field: d:\Program Files\SolidWorks Corp\si
 - Button: Browse...
- How do you want to apply SolidWorks settings?** Note: These settings can be obtained by running the Copy Settings Wizard.
 - ☒ Use SolidWorks 2010 default settings.
 - ☐ Use a settings file exported from the Copy Settings Wizard.
 - Text field:
 - Button: Browse...
- How do you want to activate client installations?**
 - ☒ Activate automatically when each client installation is complete.
 - Text field: D:\psw2010 admin image\Activation.xml
 - Button: Browse...
 - ☐ Do not activate automatically.

SolidWorks 2010 does not allow you to have multiple installations of the same major version at different service pack levels on the same computer. This means there is no longer a reason to have multiple installations of SolidWorks 2010 on a single computer. Therefore, if you are installing SolidWorks 2010 from an Administrative Image, there are only two possibilities: upgrading an existing 2010 installation to a newer service pack or installing a fresh 2010 installation where there is no existing one. The first client option gives you the option to upgrade or do a fresh install in a specified location.

The second option enables you to specify a settings file to apply to an installation or just use the SolidWorks default settings. The settings file is simply the *.sldreg file acquired by running the SolidWorks Copy Options Wizard (available through the Windows Start menu).

Cross-Ref

The Copy Options Wizard is described in more detail in Chapter 8. Activation and licensing are covered in detail in Chapter 7. ■

The final option on the Client Options tab deals with how to handle licensing activation. In order for a computer to activate the newly installed version of SolidWorks, it needs to have an active Internet connection at the time of activation. Activation is not required for new service packs, but it is required if a major version is newly installed on a computer or SolidWorks has been deactivated on the computer for some reason. You can initiate manual activation through the Help menu.

Part II: Installing SolidWorks

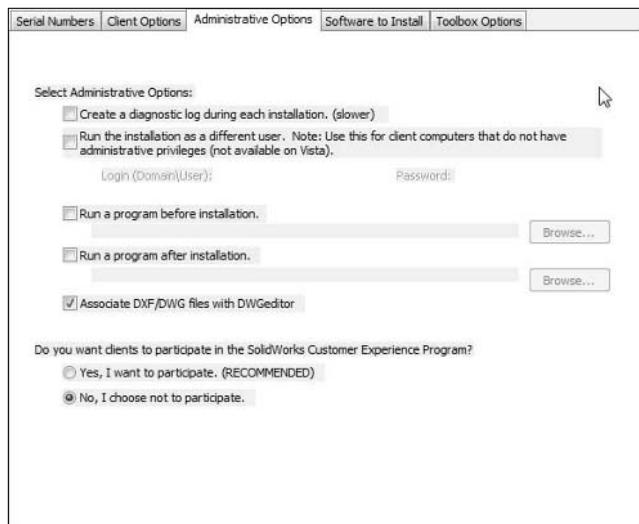
Caution

If you are using the Administrative Image to install different serial numbers onto multiple computers, you need to make sure you match the serial number to the correct machine name. If you exchange serial numbers on a single computer, you may have difficulty with the activation for the affected machines. This is because SolidWorks keeps a database of which computer names have had each serial number activated. When this exceeds the maximum allowable, the activation fails. Depending on the circumstances, you may get a 30-day courtesy period, or you may not. In any case, your VAR should be able to help you with this type of difficulty. ■

The Administrative Options tab offers the options shown in Figure 6.18.

FIGURE 6.18

The Option Editor Administrative Options tab enables you to set common options on the client installs that interest mainly administrators.



Diagnostic logging is recommended if you want to capture a potential problem with the install. The cost of using logging is speed. If you have had problems in the past and troubleshooting was time-consuming, then the logging option might be time well spent. If your network and systems are healthy, and especially if you have previously not had problems with Administrative Image installations, you might consider bypassing the logging.

If the users who are logged in to the Windows workstations when SolidWorks installs do not have sufficient privileges (are not administrators), then you probably want to run the installation as an admin user.

If you are handy at writing scripts to clean up temporary files or other items, you might want to take advantage of the options to run programs before or after the installation.

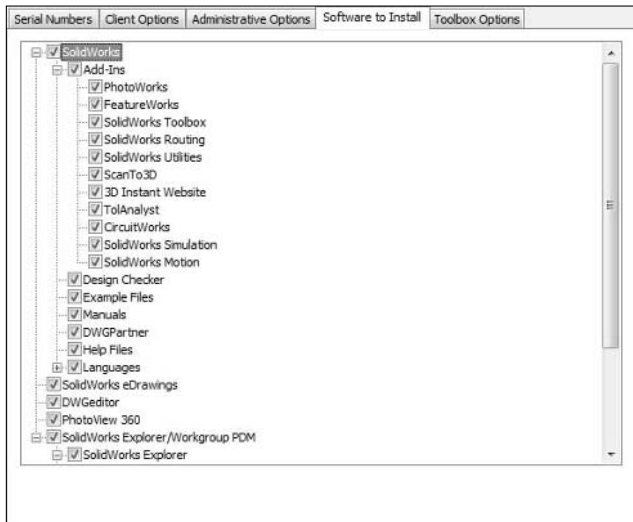
When you install SolidWorks, you may or may not want your DWG and DXF file extensions to become associated with the DWGeditor instead of whatever other program they might be associated with (such as AutoCAD).

The final administrative option is the SolidWorks Customer Experience Program. SolidWorks claims that they collect information on crashes, what types of features you use to create parts, and generally record little bits of how you use the software, then send this information back to the SolidWorks mothership and make decisions based on it. They claim the information is anonymous, but they don't provide a way for users to examine the actual information they capture and send back to headquarters. If this option is on, SolidWorks will from time to time try to access the Web to send the information.

The Software to Install tab (shown in Figure 6.19) shows the software that you selected or software that was selected by default when you created the Administrative Image. The tab gives you an option to install the software or not on selected computers listed in the Global Settings area.

FIGURE 6.19

The Option Editor Software to Install tab enables the administrator to customize each Administrative Image installation.



The information and available options on the Software to Install list should be self-explanatory. Remember that you can mix these options with selected machines or groups from the Global Settings list to use a single Administrative Image to install specific software on some machines and not others.

Part II: Installing SolidWorks

The Toolbox Options tab (shown in Figure 6.20) gives some appropriate warnings about multi-user environments and upgrading old versions of Toolbox. Unfortunately, none of the information is detailed enough to help you make an informed decision about what to do. The only option on this page is where to put the Toolbox library.

FIGURE 6.20

The Option Editor Toolbox Options tab allows you to specify a location for Toolbox on each Administrative Image install computer.



Cross-Ref

You can find more detailed information on installing, implementing, and upgrading Toolbox in Chapter 10. ■

Deploying and installing from the Administrative Image

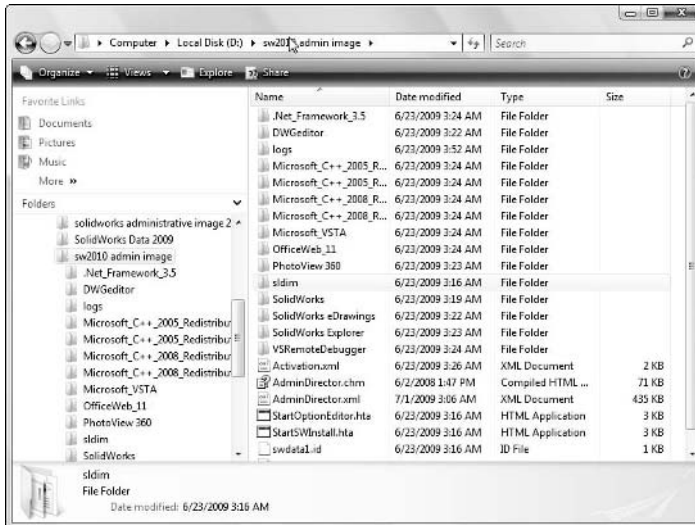
Administrative Images enable the administrator to distribute his or her workload for installing a large number of SolidWorks licenses. The administrator does not need to go around to each computer physically, but only to send a user on each computer a link to initiate the Administrative Image from the home of the image somewhere on the network.

Once the Administrative Image exists, and the settings for each of the individual installs have been established, it is time to deploy the installation. To do that, send the file StartSWInstall.hta around

to all users in an e-mail. The files in the Administrative Image folder are shown in Windows Explorer in Figure 6.21.

FIGURE 6.21

Files in the folder where the Administrative Image resides



Once the user double-clicks the StartSWInstall.hta file, and then clicks the button shown in Figure 6.22, the installation of the software should commence just as if it began with the Install step in the SWIM workflow wizard (shown in Figure 6.13).

FIGURE 6.22

StartSWInstall.hta initiates an interface to start the installation.



Part II: Installing SolidWorks

During this installation, it is recommended that the user close SolidWorks, and generally observe any rules of thumb that they would normally observe during installation of other mission critical software. The installation time varies depending on network speed, installed software options, and local computer speed, but it is wise to allot at least 20 minutes for the process, and possibly as much as an hour; again, this depends on your situation.

This may look like a complex process, but if you have more than a couple of machines to administer, the Administrative Image is certainly the way to go.

Installing an Administrative Image from the command line

Command line deployment enables you to install software remotely on a computer across the network without sending a user on that computer a link to initiate the installation. It also bypasses the need for the local user to have administrative rights to complete the installation correctly.

To initiate an Administrative Image silent installation from the command line, choose Start ➤ Run to open a command prompt window. At the C:\ prompt, enter `msiexec /a msi_path`, where `msi_path` is the path to the data folder of the Administrative Image, for example, a valid entry might be `msiexec /a D:\AdminImage\swwi\data\SolidWorks.msi`

When you do an installation of this type, you still have to communicate all of the installation options as you would in a normal Administrative Image install. Adding the options via the command line can become complex, and must be done carefully, and tested to make sure you have it correct. This type of installation generally pays off for large installations with a lot of software to install, and is probably not worth the time to learn, test, troubleshoot, and correct for just a few seats.

A complete guide to silent installations is beyond the scope of what I can offer in the limited space of this chapter. All options and requirements amount to an equal amount of space as the rest of the information in this chapter. For the complete set of information on the silent install option for Administrative Image, refer to SolidWorks Installation Guide available at http://files.solidworks.com/Supportfiles/SW_Installation_Guide/2006/English/install_guide.htm

Using Microsoft Active Directory

SolidWorks Administrative Image installations support the use of Microsoft Active Directory. Active Directory uses a database to store information about users and hardware, and allows system administrators to assign policies and use things like the SolidWorks Administrative Image to deploy software installations and updates. As the CAD Administrator, you would probably not be responsible for creating or maintaining an Active Directory, but you would need to work closely with your IT professionals to use it to deploy SolidWorks and other software.

Active Directory is not required to perform a SolidWorks Administrative Image installation. This technique involves more IT expertise than the rest of the topics in this book. Therefore, I refer you to the SolidWorks Install Guide for the details on how to accomplish installations by using the Active Directory at http://files.solidworks.com/Supportfiles/SW_Installation_Guide/2006/English/install_guide.htm.

Active Directory deployments are probably not worth the additional effort if you don't have an IT staff to assist and a large amount of software to deploy many users. Large corporate settings are the most likely places to find this type of installs.

Installing Multiple Versions

Starting with SolidWorks 2010, SolidWorks no longer supports the installation of multiple instances of a single version of the software at different service packs. This means that you cannot have SolidWorks 2010 SP0 and SolidWorks 2010 SP3 on the same computer at the same time. I imagine that few customers actually did this, but it will be a bigger inconvenience for support organizations who frequently installed versions at multiple service pack levels to aid in troubleshooting and looking for regressions.

On the other hand, I imagine this was done to simplify installations and the SolidWorks area of the Windows registry. SolidWorks registry corruption has become an all too familiar problem. Speculation of this sort may be out of place in a desk reference book, but I just want to point out that the problems SolidWorks faces when they try to improve the software sometimes require giving up some little-used or at least problematic areas in the software. I also want to point out that the old functionality is no longer available for people who may have become accustomed to using it.

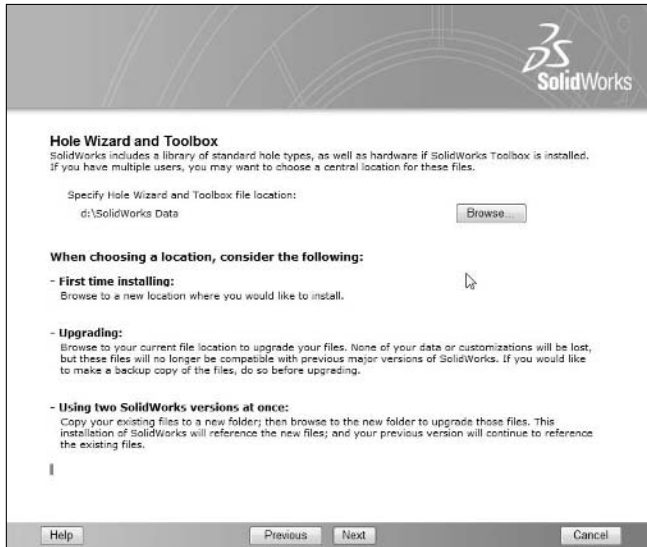
You are still able to install multiple versions of SolidWorks: you can have SolidWorks 2009 and 2010 installed and even running at the same time. Some pitfalls come with multiple versions, however. For example, you cannot read newer version files in older version software. This is commonly called *backwards compatibility*, and because the compatibility doesn't really exist, it might be more accurate to call it simply "incompatibility" instead.

One of the biggest problems with installation and version incompatibility is Toolbox. If you upgrade a Toolbox library, older versions of SolidWorks will not be able to read the parts in the library. Figure 6.23 shows a frame from the installation of SolidWorks 2009 software, which has some advice for dealing with where to place the new Toolbox data.

Part II: Installing SolidWorks

FIGURE 6.23

Advice for placing Toolbox data folder during installation of SolidWorks 2009



Installing the SolidNetWork License Manager

Installing the SolidNetWork License Manager is a step that only needs to be done if you have floating licenses. You need to install the license manager before you install the software itself on the user's computer. The SNL manager is installed like other SolidWorks components, through the SWIM or manually by downloaded files. The SWIM option to install the SolidNetWork License Manager is found on the Welcome installation screen, shown in Figure 6.5.

The SWIM SNL manager installation goes on to ask for a serial number which will start with the first four digits being 0010 or 9010, which identifies it as an old or new network license.

Activation of the SNL is done after the install of the license manager. This process has been in place since SolidWorks 2009 SP4.0. You are no longer required to obtain new license files or dongles. The function of both the license files and dongles has been rolled into the automatic on-line activation, similar to stand-alone licenses. The remaining difference between floating licenses and stand-alone licenses is the license manager server software.

After the SNL manager software is installed, you can start to install client SolidWorks software. During the installation, you will be asked for your serial number. There is only one serial number for all licenses served by the SNL. When you enter the serial number, the SWIM recognizes it as a network license, and asks you to locate the license server in the Add Server dialog box. Identify the server as `port@servername` for which a valid entry might be `25734@Bullwinkle`. For example, 25734 is the default port number for the SNL, and Bullwinkle is the name of the computer on your network on which you have installed the SNL manager.

Uninstalling SolidWorks Software

You may need to uninstall SolidWorks to move it to another computer, to free up disk space on a computer with multiple versions of SolidWorks, or to try to correct an installation that has a problem with persistent crashing.

Before you uninstall SolidWorks, you need to consider a few things:

- **Are you going to reinstall on the same machine?** SolidWorks warns you if you uninstall SolidWorks on a machine that has an active activation.
- **Do you have any data in the SolidWorks installation directory that you want to keep?** Frequently users are lazy about storing library type data and sometimes put it in the SolidWorks installation directory, which will be lost on uninstall. Keep separate locations for storing data you need to save.

Note

Always take advantage of SolidWorks by choosing Tools⇨Options⇨System Options⇨File Location settings to find where you have customized data stored. ■

- **Do you need to keep your Toolbox library or database?** If you have made customizations to your Toolbox library or database, you may want to save it. Whether you are moving to a new computer or will eventually reinstall on the same computer, you need to have a copy of this data saved somewhere safe.
- **Do you want to keep the settings of interface customization that you have created?** Use the Copy Settings Wizard to save out a portion of the SolidWorks section of the Windows registry. This can be moved to other computers or other users on the same computer.
- **Do you want the computer to retain the SolidWorks serial number?** If you are retiring a computer and giving it to someone else, you might uninstall SolidWorks and think that you are safe from someone stealing your SolidWorks serial number, but that's not true. A simple uninstall allows the SolidWorks serial number to remain in the Windows registry so that it comes up automatically the next time the software is installed. A resourceful person can even find your serial number right there in readable digits.

Part II: Installing SolidWorks

- **Is there something unexpected going on with your SolidWorks installation?**
Sometimes the solution to a recurring problem is something you will never actually find, and can only fix by clearing the registry or, more dramatically, reinstalling the entire operating system.
- **Are you hoping that any changes you make can be reversible?** When you make drastic changes to your operating system, you do not have an undo button, but you can save or back up files or data that you would otherwise simply delete.

Do a normal uninstall through the Control Panel. For most situations, this should be both appropriate and sufficient. However, you may run into situations where an uninstall does not happen as expected, or you need an additional level of uninstall due to security, troubleshooting, repair, or other concerns. For the purposes of this book, I assume that you know how to uninstall software from the Windows Control Panel.

To completely remove SolidWorks, the first thing you should do is go through the list of items at the beginning of this section to see which apply to you and which suggest that you have some additional work to do prior to actually starting the uninstall proper.

It is safest at this point to go to the Help menu and select Transfer Licenses, which essentially deactivates SolidWorks on that computer. This makes it possible to install SolidWorks and activate on an entirely different computer, or reactivate on the same computer.

If you intend to rebuild your computer, SolidWorks considers the computer a combination of hardware, and after you change a certain number of components, it considers the computer to be a different computer. These components may include the computer name, processor, motherboard, network card, or primary hard drive. Be aware of this if you are rebuilding a computer.

After you have determined a course of action and followed through with it for each item in the list, you can go to the Control Panel and use the uninstall functionality. If you are using XP, you will find it in Add/Remove Programs, and in Vista, you will find it in Programs and Features.

Note

The uninstallation process may warn you that a license is still activated on the computer. If you want to move the license to another computer, you should deactivate the license. If you will reinstall the license on the same computer you are uninstalling it from, you do not need to do anything to the license. ■

Note

To access the Registry Editor, choose Start ⇨ Run. In the Run dialog box, type regedit. This opens an interface that looks like Windows Explorer. ■

To clear SolidWorks from your machine completely, you need to delete the following keys from your Windows registry. Remember that registry editing is a dangerous business, and if something goes wrong can result in errors that cannot be repaired, or can be repaired only by reinstalling or recovering backup data. Also, be aware that in order to delete some of the following keys, you may need to have the license activation transferred.

HKEY_CURRENT_USER\Software\SolidWorks
HKEY_CURRENT_USER\Software\SolidWorks BackOffice
HKEY_LOCAL_MACHINE\SOFTWARE\SolidWorks
HKEY_LOCAL_MACHINE\SOFTWARE\SolidWorks Corporation

Be aware that for the first key (HKEY_CURRENT_USER\Software\SolidWorks), a list of versions of SolidWorks appears under the top-level item. If you only want to get rid of some versions, and not all, only delete the folders that you want to get rid of. Otherwise, simply erase the entire SolidWorks folder.

All the user settings for interface and system options are stored in this (HKEY_CURRENT_USER) area. Be aware that if multiple users are on the computer, you may have SolidWorks data in the registry for each user. Also, be aware that you cannot reach the registry of one user while logged in as a different user.

The HKEY_LOCAL_MACHINE settings contain primarily licensing information. You cannot have two different serial numbers installed on the same machine. This question often comes up when student-licensed software is involved. If you delete these folders, the SolidWorks installation will not remember your SolidWorks serial number.

Frequently, this uninstall and registry purge is the only way to ensure a clean install. Make sure that you are logged in as an administrator-level user whenever you perform install or uninstall operations. Remember to be extremely careful whenever you are working in the Registry Editor.

Summary

A good, clean install is your best defense against a problematic and crash-prone experience with SolidWorks. The SolidWorks software offers many routes to accomplish this, including installing from the DVD, from DVD-copied data, from downloaded data, and from an Administrative Image. An Administrative Image is an automated way to deploy a consistent SolidWorks installation to a large number of computers in widely different locations.

Removing the software can be trickier than loading it, and fewer methods for doing it exist. Use caution: Some of the methods I describe in this chapter can be potentially harmful to your computer if you perform them incorrectly. Also, remember that you have many implications and things to consider when uninstalling SolidWorks, including license activation, saved data, stored macros, templates, and other accumulated data, such as blocks. Make sure to move all this data to a safe location prior to uninstalling the software. Regardless of the method used, be it from the steps outlined in this and related chapters or some homegrown method, you should create a checklist to be used consistently within your particular CAD environment.

Licensing

One of the most important tasks assigned to you as the CAD Administrator is to maintain the licensing information for a company's SolidWorks seats. While it is undoubtedly a tedious task without much reward, it is also very important to manage this information closely because dropping support for the wrong license could cost the company extra money to reinstate it or eliminate key functionality needed for your business.

Licensing has changed somewhat over the years. Product naming has also changed. Even the serial numbers and activation processes have changed. If you are a longtime customer or a returning customer, you may not recognize some of the new licensing schemes or the names of products.

Despite the changes, many of the policies and even prices for the various levels of software and subscription have remained unchanged from the very beginning.

Some aspects of licensing are different inside the United States from outside the United States. Outside the United States, the use of dongles, even for stand-alone licenses, is common. This type of licensing is most similar to network licensing.

IN THIS CHAPTER

Choosing a SolidWorks bundle

Tracking each assigned serial number

Activating and transferring licenses online

Using IT to administ floating licenses

Taking advantage of home use licenses

Selecting the Version of SolidWorks

When you purchase SolidWorks, you have the option to buy one of three bundled packages, SolidWorks Standard, SolidWorks Professional, or SolidWorks Premium. The bundled functionality is outlined in the Table 7.1.

If you require more detail to determine, for example, if Simulation Xpress has enough functionality to meet your needs, your best bet is to go to the SolidWorks Web site (www.solidworks.com) or ask a SolidWorks reseller. SolidWorks software is sold only through resellers.

Table 7.1 lists the functionality available with the three levels of SolidWorks software, along with the license and subscription costs for each.

TABLE 7.1

SolidWorks License Types

| License Types | SolidWorks Standard | SolidWorks Professional | SolidWorks Premium |
|--|---------------------|-------------------------|--------------------|
| Part modeling | ✓ | ✓ | ✓ |
| Assembly modeling | ✓ | ✓ | ✓ |
| Drawings | ✓ | ✓ | ✓ |
| Sheet metal | ✓ | ✓ | ✓ |
| Weldments | ✓ | ✓ | ✓ |
| Molds | ✓ | ✓ | ✓ |
| Surfacing | ✓ | ✓ | ✓ |
| Dynamic Assembly Motion | ✓ | ✓ | ✓ |
| Motion Manager (formerly Animator) | ✓ | ✓ | ✓ |
| Simulation Xpress | ✓ | ✓ | ✓ |
| Toolbox hardware configurator | | ✓ | ✓ |
| eDrawings Pro | | ✓ | ✓ |
| FeatureWorks | | ✓ | ✓ |
| SolidWorks Utilities | | ✓ | ✓ |
| Workgroup PDM (formerly PDMWorks) | | ✓ | ✓ |
| PhotoWorks/Photoview 360 | | ✓ | ✓ |
| Simulation (formerly Cosmos) | | | ✓ |
| Mechanism Simulation (formerly Motion Works) | | | ✓ |

| License Types | SolidWorks Standard | SolidWorks Professional | SolidWorks Premium |
|-------------------|---------------------|-------------------------|--------------------|
| ScanTo3D | | | ✓ |
| Routing | | | ✓ |
| CircuitWorks | | | ✓ |
| Software Cost | \$3,995 | \$5,495 | \$7,995 |
| Subscription cost | \$1,295 | \$1,495 | \$1,995 |

SolidWorks software and subscription costs have never been increased since the software was released in 1995. The higher priced bundled packages offer much more functionality, and have been the only way that SolidWorks has raised their prices. The functionality included in a license of SolidWorks Standard 2010 is far greater than what was offered in SolidWorks 95, but the price is the same.

Formerly a version existed called SolidWorks Office, and the upper two levels were called SolidWorks Office Professional and SolidWorks Office Premium. In addition, the SolidWorks Standard was formerly called simply SolidWorks. If you are a longtime customer, you may have also purchased licenses of PhotoWorks, Toolbox, or Animator separately.

Additional types of licenses are also available for educational, research partner, and others. Some of these are available at prices reduced from those listed. Again, you will need to check with your reseller or with a SolidWorks representative for the non-commercial license types.

Understanding your CAD needs

In many cases, the basic capabilities may be overkill and at the same time not enough. If you are involved in machine design-type applications, the sheet metal and weldments functionality will be useful for you, but the surfacing is likely unneeded. You might want the Toolbox hardware configurator, but the \$1500 jump just for that is too much. This is part of the unfortunate scheme of pricing bundles; in order to buy what you need, you wind up paying for many things you will never use.

If you happen to find yourself in that specific situation where you are faced with upgrading just to get Toolbox, I suggest having a look around for another solution for a library. Toolbox is not that remarkable a tool to pay out extra money. On the other hand, if you will make use of any three of Toolbox, Utilities, eDrawings Pro, PhotoWorks, or Workgroup PDM, the investment may make more sense.

Fortunately, for just about any modeling work, the Standard package has the tools you need. Some exceptions may include some point cloud or mesh modeler translation options only available with ScanTo3D, as a part of the Premium package. If ScanTo3D is the only part of Premium you need, it might be advisable to see if it is possible to negotiate a deal to pay for just that software separately.

Matching needs to capabilities

If you are in the evaluation phase to buy more SolidWorks software, it might be useful to take some of your more advanced users along with you on a visit to the reseller's office to determine which level of SolidWorks makes the most sense for you. With the three bundled packages, SolidWorks offers a very broad range of functionality from motion analysis to highly automated wiring libraries. With the availability to buy network (floating) licenses, you may be able to justify advanced functionality for part-time users distributed around your network.

You may find yourself in the position of having some specialty needs that SolidWorks cannot help you with right out of the box. For example, if one of the tasks you have to complete often is to design auger blades from the flat pattern so your shop can form them, SolidWorks cannot flatten parts that are not bent on a brake press. Still, there is some functionality that SolidWorks calls Lofted Bends, which enable the software to approximate flattened ruled surfaces. To evaluate if this functionality is “enough” to get you where you need to go, it is well worth your time to evaluate that functionality in SolidWorks along with reseller engineers. If you have to buy an additional package of software just for that particular task, it may take a little of the shine off the deal you thought you got with SolidWorks.

Tracking Serial Numbers

Serial numbers for your supported software are vitally important company information. Tracking where each serial number is installed, the primary user, its corresponding bundle level, and its subscription status are all things you should keep track of. You also need to protect your serial numbers from theft if possible. This can be difficult, because any user with access to your computer can find the SolidWorks serial number easily if they are interested in finding it. You should consider only allowing authorized users to have access to your SolidWorks workstations to keep the numbers as secure as possible. Anyone who has access to your serial number also has access to the tech support and service pack downloads for which you have paid.

SolidWorks serial numbers are 16 digits followed by eight characters. For many years, serial numbers were only 16 digits with a four-character registration code, and the code changed to eight characters. If you have some old licenses, you may have separate serial numbers for products such as PhotoWorks or Toolbox. If you are in this situation, in newer versions, SolidWorks has probably rolled it all into the single serial number for your SolidWorks license, but I would verify that before getting rid of any old paperwork.

I recommend keeping an Excel spreadsheet with your serial numbers, the level of the software, the user that it is registered to, and any other pertinent information such as contact numbers for your reseller, renewal dates, and so on.

Identifying types of serial numbers

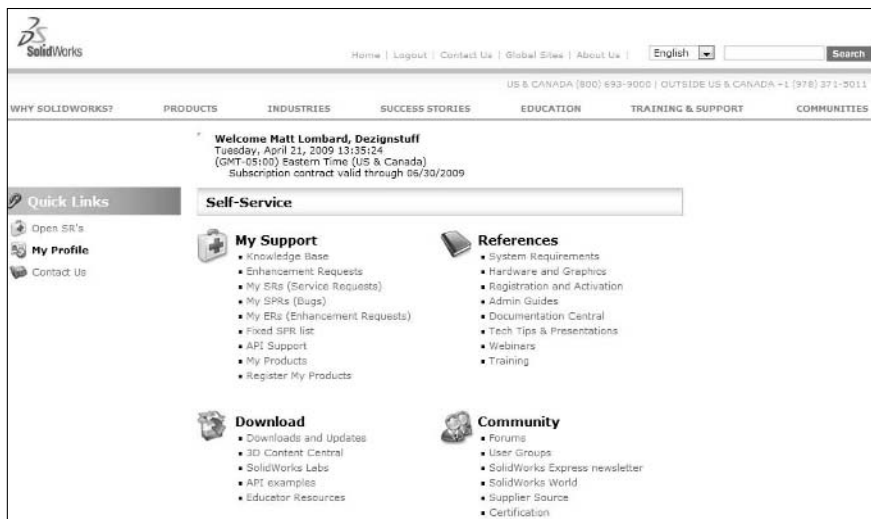
If you have a long list of serial numbers to manage and your company has different types of licenses (including floating licenses or research partner licenses), you may find it useful to identify the licenses by sight rather than checking them against the list. For example, full commercial licenses start with 0000, while research partner licenses start 9000, and floating licenses are 0100. When you manage a large number of serial numbers on the SolidWorks Customer Portal, being able to associate a license number with a user or a support issue can be useful.

Using the Customer Portal to manage assets

The SolidWorks Customer Portal has some very useful tools to help you manage your SolidWorks serial numbers. You can access the Customer Portal at <https://customerportal.solidworks.com>. You will be required to sign in. If this is the first time you have been to the Customer Portal, you may need to enter your serial number information. Figure 7.1 shows the interface at the Customer Portal.

FIGURE 7.1

Customer Portal home page showing the interface



Under the My Support heading, click the My Products option. This takes you to the page that lists your SolidWorks serial numbers, registration, and service dates. You can also use this area to get registration codes for older versions of software. SolidWorks versions of 2007 and prior require eight-character registration codes. SolidWorks 2008 and later use online activation.

Activating and Transferring Licenses

Starting with SolidWorks 2008, SolidWorks started using online activation instead of registration codes for individual stand-alone licenses. The new system is easy and reasonably reliable. You enter your serial number during installation, and then go online and the activation is automatic. You are allowed two concurrent activations for every license because of the Home Use License provision of the End User License Agreement (EULA). Every license entitles you to install the software on your work computer and a home computer or laptop for the primary user of the work license. Refer to the actual EULA in the software for exact wording. You can find the EULA by clicking the Help menu in SolidWorks, selecting About SolidWorks, then clicking the License Agreement button.

The EULA can be some dull reading, but you should read it at least once to understand what SolidWorks offers and requires from users.

Before attempting to activate a new SolidWorks license, you should sort out any activation that you have for a particular serial number. Again, you are allowed two active activations per license. This does not work like Microsoft activations; it is far more reasonable and generous. In Microsoft activations, once you activate, that is one. You never have the option to deactivate or transfer that activation. If you install and activate again on another computer, that is two. Again, there is no going back with Microsoft.

SolidWorks does allow you to deactivate licenses in order to move them to another machine. Therefore, you can install the software, even if it is a third installation, deactivate one of your two allowed activations, and activate the third installation. This is very useful if you work in multiple locations, or have different computers for different purposes. A fair yet flexible system also protects SolidWorks from people misusing licenses.

If you already have activated a particular serial number on two different computers, but you want to activate it on a third, you have to make a choice. You will need to deactivate one of the first two.

Activating a license

If you have a freshly installed SolidWorks, activation should start up automatically when you first run the software. If you have installed it and put off the activation, you can start the activation manually by clicking the Help menu and selecting Activate Licenses, as shown in Figure 7.2. You are given a 30-day trial license, which should be more than enough to activate the license.

This opens the SolidWorks Product Activation Wizard interface shown in Figure 7.3. You can activate automatically over the Web or by e-mail. If you choose e-mail activation, the wizard prompts you to save a file containing the activation information and sends it to SolidWorks. SolidWorks then sends you back a file that you need to open with the Activation Wizard. This process works well for computers that are not hooked up to the Internet. The file can be e-mailed from and to a different computer but must be used to activate the SolidWorks license on the computer that the file came from originally.

FIGURE 7.2

The license activation menu selection

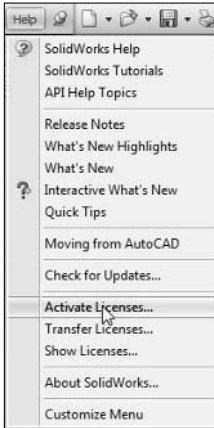
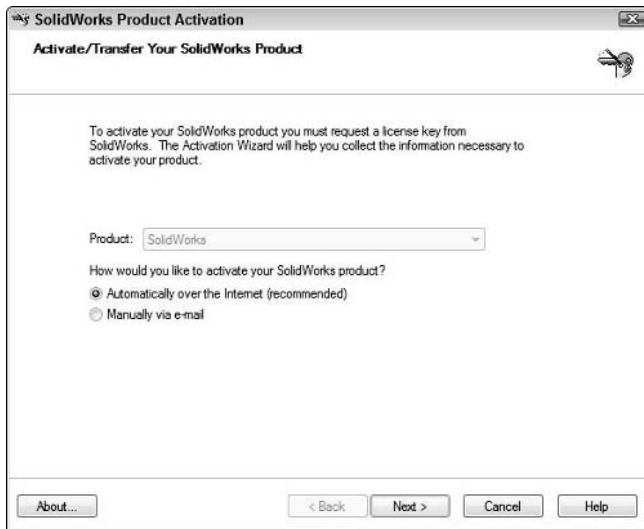


FIGURE 7.3

The SolidWorks Product Activation Wizard



This process is painless and almost instantaneous. If you have the license available, it is better mannered than other activation schemes you might run into, including Microsoft. That is all there is to it. Well, almost all there is.

Part II: Installing SolidWorks

Working in spite of an activation error

In the circumstance that a problem with the activation occurs, your reseller should be able to straighten it out for you in prompt order. This is not a 24/7 option unless your reseller has that availability. Also in the case of an activation problem, the SolidWorks installation gives you a 30-day trial license. This is a nice cushion to make up for the lack of 24/7 support from the reseller. You still have complete access to your software, but you have to get it activated within 30 days, which is generous.

If you exhaust the 30-day trial and you are stuck on a weekend or after-hours, you are going to be without your SolidWorks license until working hours resume. A little housekeeping is a necessary thing.

Identifying where licenses are used

From the Customer Portal, go to the My Products link, and you will see Figure 7.4. This list all the serial numbers registered to your company and shows which ones are activated. For licenses prior to 2008, the chart lists a registration code; for 2008 and later, the word “activated” appears. The image in Figure 7.4 is altered to protect my personal data, including serial numbers and registration codes.

FIGURE 7.4

A list of your registered products

| Serial # | Product | End User | Reg Code | Service End Date | Registered Date |
|-------------|-----------|-----------|-----------|------------------|-----------------|
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |
| > [blurred] | [blurred] | [blurred] | [blurred] | [blurred] | [blurred] |

If your SolidWorks serial number has not been registered, you need to go to the Customer Portal (see Figure 7.1) and click Register My Products under My Support heading. Enter your serial number, the version of the software, and fill in your personal information.

If you need to get a registration code to install an older version of SolidWorks, you can get that on the My Products page; highlight the arrow to the left of the entry (serial number and version) you want the code for, and click the Get License link in the upper left. This triggers an e-mail to you with the registration code for that serial number and the selected version of the software.

Troubleshooting license problems

Almost all the troubleshooting tools are in the hands of either your reseller or SolidWorks Corp. Aside from keeping your own records and knowing which licenses are activated and how many times they are activated, you cannot do much to find or fix problems. Even if you are not on subscription, the reseller is obliged to service your licensing requests.

If you transfer a license activation frequently, you may run into the situation where the software has been activated from several computers. The SolidWorks activation server may deny additional activation requests if you have activated the same license from more than four computers. For this reason, it may not be practical to use license activation as a poor man's floating license. Your reseller or SolidWorks support direct should be able to clear the accumulated machine names from the server to allow you to start fresh again.

Replacing a system drive in a computer may be another reason for an activation not working. In this situation as well, the SolidWorks reseller should be able to set things right for you.

Administering floating licenses

Floating licenses are also called *network licenses* or *SNLs* (SolidWorks Network Licenses). These types of license allow you to install software wherever you need it, and the software pulls from a central pool of licenses available. Floating licenses are driven by an application on a server called FLEXlm.

Starting with SolidWorks 2010, floating licenses use activation just like standalone licenses. The main difference is that when you install the SNL Administrator software on the license server, you must perform the activation from the SNL Administrator instead of from the SolidWorks software itself.

The *license server* is just software that controls the number of licenses that are passed out to users who request them. The server does not necessarily require an actual server computer. It requires any kind of computer that is always running when users need to use the software. The computer should not require frequent rebooting during working hours.

Using license management software

When you install the software on the local machine, the serial number you enter tells the software it is a floating license, and it asks you for a license server. For this reason, you need to have the license server installed and set up before trying to install the clients. All the client machines will use the same serial number, even though there may be different numbers of multiple levels of the software. For example, you might have four SolidWorks Standard licenses and six SolidWorks Professional licenses. If a user is using a SolidWorks Standard license, but then turns on the PhotoWorks add-in, it takes a Professional license and releases the Standard.

Also, be aware that after 15 minutes of inactivity, the license automatically reverts to the pool. This action prevents a license from being locked up all day by someone who forgot it was running. You can adjust the 15-minute time-out period in the FLEXlm.

Part II: Installing SolidWorks

The FLEXlm application makes available a broad range of tools for management and reporting usage. A hefty user manual for FLEXlm exists called the *FLEXlm End Users Guide*. It is also available from the Customer Portal.

Other documentation available in the Customer Portal area includes an SNL FAQ to help you find answers to many of the basic questions you may have about floating licenses, which are not duplicated here.

Using dongles

One of the most exciting enhancements in the SolidWorks 2010 version is that SNL licenses no longer have to use dongles or license files. SNLs are now activated through the same online activation process as stand-alone licenses. The activation must be done from the server before any of the licenses can be used. The same rules apply to SNL activation as apply to stand-alone software installations.

A *dongle* is a hardware lock, usually through a USB (Universal Serial Bus) port. Old dongles were attached to a parallel port. The dongle is only necessary on the server, not on the client machines.

Activation enables you to move the license server easily by deactivating the SNL on one machine and reactivating on another. Activation of a 2010 SNL can also administer older licenses that used dongles, so the dongles are no longer needed once your activation goes through.

Installing Floating Licenses

Floating licenses add to the convenience and utility of your SolidWorks investment. However, along with this convenience comes some overhead in the form of extra administration tasks that you don't have with other types of SolidWorks licenses. License administration is often a task handled by IT professionals even when a CAD administrator is available. SolidWorks or engineering skills are not required to administer floating licenses, but network and general IT skills are.

The main steps in administering floating SolidWorks licenses are:

1. **Install the license server software on the machine you will use as the license server.**
2. **Activate the SolidNetWork License.**
3. **Install the SolidWorks software on a client machine through the DVD, downloaded data, or an Administrative Image.**
4. **During client installation, use the serial number that identifies the license as a SNL.**
5. **Point the client installation to the license server machine name, or IP address and port number.**
6. **Start SolidWorks on the client computer to make sure it gets the license from the server.**
7. **Check the license manager software to make sure it is working as expected.**

The rest of the administration of SNL licenses is the same as stand-alone licenses. For details on SNL administration, see the `Install_Guide 2010.pdf` available from the Customer Portal.

Cross-Ref

More information on the details of license server and client installation is also available in Chapter 6 of this book. ■

Developing a Home Use License Policy

SolidWorks' End User License Agreement allows for the use of the software at home or on a laptop computer. The policy works slightly differently depending on what kind of license you have at work. Home use licenses are extremely valuable for both employees and employers. The employee can use the software for hobby type work or home projects, and the employer allows an employee to develop skills off the clock.

Weighing advantages and disadvantages of home use

The home use provision can be a big advantage for everyone — SolidWorks, companies, and users. SolidWorks benefits when its software becomes more widely used. Companies benefit when its users gain more experience and confidence with the software. Of course, users benefit from gaining skills they can use at work, but 3D CAD can be a hobby for some people, as well.

Home use licenses are not intended for moonlighting arrangements, where individuals make extra money on the side, but they can be used by employees for occasional telecommuting or laptop computers.

On the other side of the issue, companies are sometimes reluctant to allow users to take installation disks home with them and often question what to do if the employee's employment with the company ends. For example, how do they make sure the software is uninstalled and they get that activation back? Is the employee moonlighting to make some additional money on the side? These are questions you should be able to resolve. SolidWorks or your reseller should be able to clear rene-gade activations, and a sign-out policy for install disks should be sufficient for most companies. The potential benefits of having users learning at home are just too big to sacrifice to ease a little paranoia.

Identifying who is eligible

For stand-alone licenses, there is a one-to-one eligibility, meaning that the user of the license can also install the software at home or on a laptop. The goal of the policy is to be reasonably flexible and allow access without permitting abuse of the spirit of the policy. The home use license is not meant to allow people to give expensive software to their friends, enable people at companies to double up on a serial number, or to use to subsidize your income by working on the side. It is

Part II: Installing SolidWorks

meant to help you and your company by giving you access to the software in convenient locations, so you can take design work on the road, use the software for learning, or use it to have fun.

According to the EULA, if the primary user of the software at work uses it more than 80 percent of the time it is in use, that person is eligible to take the software home. This policy is really aimed at SNL users, which I address in the next section.

Obtaining home use licenses for SNL users

SNL license holders require special arrangements with SolidWorks to allow home use licenses. I have already mentioned the first requirement: the 80 percent rule. One of the advantages of SNL licenses is that many people can use the same license. If all SNL users were allowed to take advantage of the home use license, there would be more home use licenses than floating licenses. The spirit of this rule is that the primary user of a license should have the software available to take home.

Because SNL licenses aren't as simple to install as regular stand-alone licenses, the users of SNL licenses have to obtain special permission to install home use license. This permission comes through your reseller.

Summary

Licensing is a tedious but necessary component of the SolidWorks Administrator's experience. License activation greatly simplifies the administrator's tasks, especially if you are using floating licenses. Few will mourn the passing of dongles.

Make sure to make use of home licenses for whatever type of license you have. This is an important benefit for every one from management to administrators to the end users. SolidWorks licensing enables great flexibility within the terms of the licensing agreement.

Standardizing Settings

Maintaining standardization is one of the most important duties of a CAD Administrator. Standardization has many benefits, such as getting all users on the same page, speaking the same documentation lingo, and communicating with vendors with the same graphical style. Consistency in drawing practices as well as modeling practices is important for streamlining tasks to reduce errors and make sure all types of documents are interpreted consistently. Standardization also helps when you have users who are not SolidWorks *wizards*. Wizards can figure out anything you throw at them, but average users generally have a limited range of functionality they can navigate successfully. Standardization can help keep your work in the range of functionality for which most users have been trained.

Beyond the technical advantages, management will be pleased that standardization saves money, helps define relationships with outside organizations, and makes those relationships flow more smoothly.

One of the things you have to be able to allow for when creating a new standard or adding to an old one is that users are people. You cannot make users into clones who all function the same way under all circumstances. Individuals are different. We understand things differently, learn differently, and execute differently. Compounding the difference between users is the fact that SolidWorks is a tool that has an immense range of features that enable users to do many different types of specialized work. It would be rare to find an individual or even a company that actually used all the various types of tools inside SolidWorks, from sheet metal, to weldments, plastics, mold tools, machined parts, electronics, wiring, piping, complex surfaced parts, chart driven parts, and several others.

IN THIS CHAPTER

Normalizing system options

Regulating modeling practices

Part II: Installing SolidWorks

While it is not possible for me to help you create detailed standards (otherwise known as “best practice suggestions”) for all these types of parts or manufacturing methods in this book, I can help you with a framework on which to base a standard for specific types of modeling and documentation work.

Creating a standard is a tough job. You have to balance input from existing national, international, and industry standards, suppliers, vendors, and your own company’s needs. A standard will consist of the following:

- **Templates.** Provide a common starting point.
- **Tools.** Provides a standard palette of tools and settings for work.
- **Techniques.** Provide standard operating procedures for common tasks.

The tools you choose are important because of training issues and the range of available tools. When you limit the ones available to users for certain types of tasks, you ensure that all users can work with and replicate the results. Limiting tools by saying, for example, that splines should not be used on certain types of work will go a long way to stretching your training dollar by ensuring that parts are built from a consistent palette of features that everyone understands and expects.

Techniques are important to product control from a standards point of view because of user training. For example, you may be able to come up with some very clever ways of creating sheet metal parts that do not use the standard SolidWorks sheet metal tools and techniques, but if you are the only person in your company who understands it, a serious workflow impediment may arise at some point. It doesn’t make sense to train all new and existing users on a niche technique that still has limitations, is nonstandard, and is understood only by you.

In this chapter, I focus on settings, which fall under templates, even though there are many settings that are not part of the standard SolidWorks templates. Standardized settings are very much like best practice suggestions. In fact, the entire 3D standardization idea is essentially one where you document best practice suggestions. If you are having difficulty putting into words exactly which type of controls you want to implement in a modeling standard, start thinking of it in terms of best practice instead. Best practice tends to refer to suggestions, whereas standards are absolute, but the controls should be the same.

You will find most existing CAD standards are drawing or drafting standards that apply as much to pencil drawings as to computer-printed drawings. Standardizing 2D drawings is certainly something you need to be concerned about. Standards for 2D drawings created from 3D data are as important as 2D drawings created with a pencil or AutoCAD. These standards have been around for a long time, and some are well developed. However, as an administrator of a SolidWorks installation, there is much more than just 2D standards.

When it comes to layer names and line thicknesses on a 2D drawing, you will find no end of advice and different systems for different purposes. However, when it comes to standardizing your 3D data or, more specifically, your SolidWorks native feature data, you will find less guidance. In SolidWorks, the end product is certainly important, but how you achieve the end product is nearly

as important, so it becomes necessary to standardize settings and options as well as procedures that you might not normally think of standardizing if you are using a program like AutoCAD for creating product documentation.

Standardizing System Options is important in some cases and not so important in others. System Options control settings for the local computer and do not affect settings on other computers. The settings are sometimes tricky or deceptive — sometimes simply control defaults that actually do affect document settings — so you have to take the names “document” and “system” with a grain of salt.

When you talk about standardizing Document Options, you are really talking about creating and using document templates, which I cover in Chapter 9.

Depending on your company’s practices, other people may or may not ever get to see and work with your SolidWorks parts and assemblies. The more likely it is that anyone other than the model’s creator must work with the native SolidWorks data, even inside of your company, the more important it is for SolidWorks users to be careful about how they model. Getting users to model the same way in SolidWorks is a bit like asking them to think in the same way. While you may not be successful enforcing that kind of standardization, you can get people to follow some simple rules that ensure anyone in your group can pick up a model that anyone else in the group has done and be able to understand it well enough to work with the part.

In this chapter, I discuss what it means to have standards in general, as well as what it means to create, follow, and enforce 3D standards and general SolidWorks software standards.

Standardizing System Options

Standardized SolidWorks System Options are important because they give users a common environment to work in. It is true that some System Options may be considered personal preference because they do not affect other users or the output data; other settings such as Backup/Recover, File Locations, Collaboration, and Default Templates have very definite effects on other users and administration.

The SolidWorks System Options are the settings you find by choosing Tools ⇨ Options ⇨ System Options. I describe all these options in detail in Appendix C. Figure 8.1 shows the Tools ⇨ Options ⇨ System Options interface.

Traditionally in SolidWorks, System Options are settings that affect your work on your local computer but do not affect the files used by other people on other computers. However, some document specific settings override System Options, and some System Options are converted into Document Properties.

An example of a System Option that affects Document Options is the Large Assemblies mode. The settings for Large Assemblies mode are shown in Figure 8.2.

Part II: Installing SolidWorks

FIGURE 8.1

Choosing your SolidWorks System Options from the Tools⇨Options⇨System Options interface

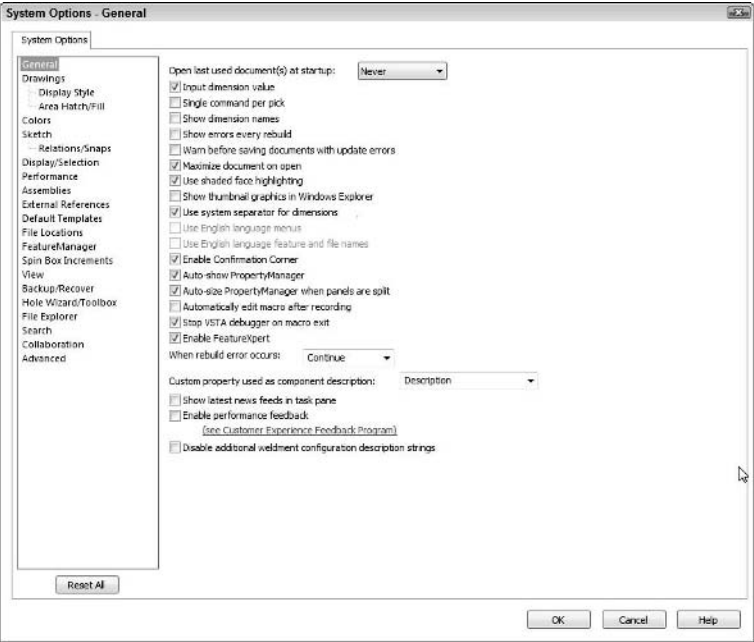
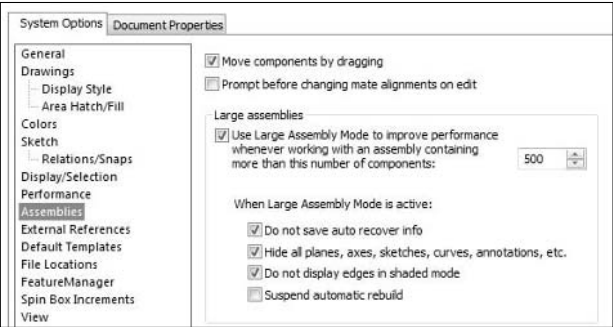


FIGURE 8.2

Setting for Large Assemblies mode in the Tools⇨Options⇨Assemblies interface



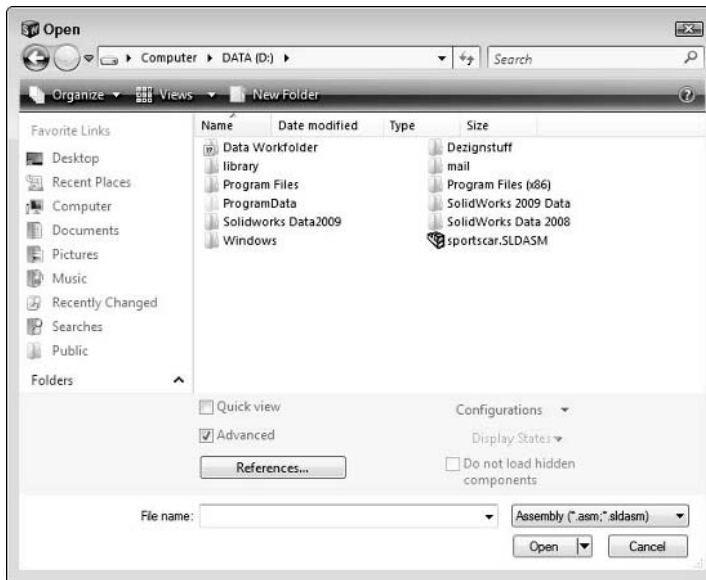
When an open assembly exceeds the limit specified in the Large Assemblies spin box, SolidWorks turns on Large Assembly mode for that assembly. The problem is that Large Assembly mode is not a System Option at all; it is a Document Property that will stay with the assembly the next time it is

opened, even if it is opened on another computer. This example of mixing system- and document-based settings can lead to confusion.

SolidWorks Help says, “You can load an assembly with its active components fully resolved or lightweight but you can’t do that through the System Options or the Document Properties.” The lightweight settings have been moved to the Open dialog box, but where are they? Figure 8.3 shows the SolidWorks 2009 Open dialog box (substantially the same as the SolidWorks 2010 dialog box), and the option is not there.

FIGURE 8.3

The SolidWorks 2009 Open dialog box without the Lightweight option

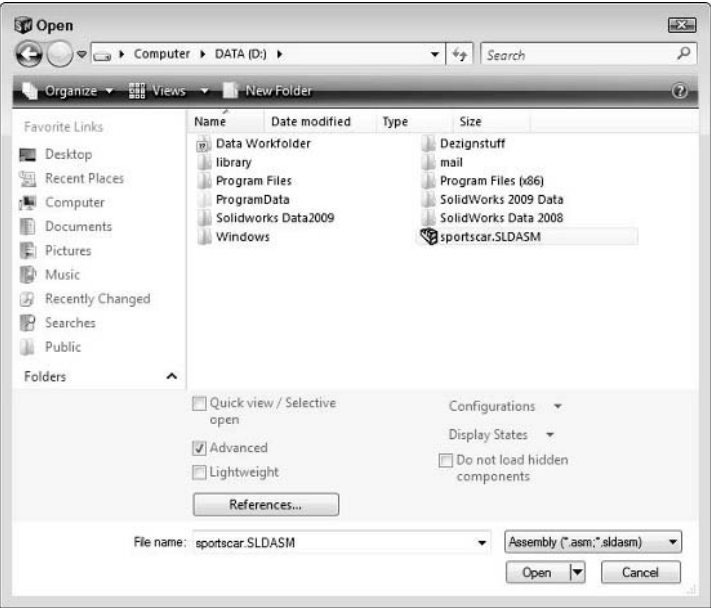


Note

It turns out that you need to have an assembly selected in the Open dialog before the Lightweight option becomes available. The Lightweight option is shown in Figure 8.4, where an assembly is also selected in the File Name box. Is this unnecessarily cryptic or something that should just be obvious? You run into the same sort of conundrum with imported parts, where the option to find the import settings are not even available until you have changed the Open dialog box filename filter drop-down menu to an imported file type. With imported parts, you do not need to have a file selected, only the filter set to the right file type. Setting the Assembly file type in the filter does not enable the Lightweight option. Some options are grayed out (such as Display States and Do not load hidden components) when they are not available. You can see this in the difference between Figure 8.3 and 8.4. The use of hidden functionality in the user interface is both cryptic and inconsistent. ■

FIGURE 8.4

The SolidWorks 2009 Open dialog box with an assembly selected



You can see how System Options can affect Document Properties, and that the settings may not be consistent, obvious, or where you remember them. With the new placement of some options comes a cleaner interface, and the settings are closer to where the end user needs them rather than in a central location, which is easier for Administrators.

Identifying the need for standardization of settings

Standardizing the format of drawings is a well-established concept. People know that all the drawings from one company need to look like they come from that company, and those drafters need to follow certain conventions when creating drawings to make them easier to interpret consistently.

Inasmuch as the System Options really do affect only the local computer, they *shouldn't* affect other users, so they shouldn't matter in a discussion about standardization. Unfortunately, the local-only affect is not always true. This means that when making a decision about which settings to control for all users and which settings to let individuals control themselves, you will have to pay particular attention to the System Options that determine document defaults.

Selecting settings to standardize

When it comes time to select specifically which settings to standardize, the decision may not be as difficult as you are anticipating. Settings that you can allow users to control on their own are usually obvious, as are those that require some sort of control. In this section, I recommend some settings to control, with reasons for your needing control.

In most cases, the safest option is generally the most conservative one; however, often conservative means least convenient. Convenience usually comes at some price, though once you are well acquainted with the software, you may choose to go a more convenient, more risky route.

Best Practice

Recommended settings tend to vary with the experience of the user. Beginning users should try to avoid highly automated settings because they tend to hide important options. Although using less convenient settings is often tedious, new users will really understand the software before adopting potentially habit-forming shortcuts. ■

All the settings that follow are specified in Tools⇨Options but are stored in the Windows registry. This means you don't have any sort of settings file you can text edit like in some other CAD software, but you can copy settings to other users on the same machine or other machines if you want by using the SolidWorks Copy Settings Wizard or standard Windows registry functionality.

Caution

Manually editing the Windows registry can be risky. If you get things wrong, you could easily wind up needing to reinstall SolidWorks, other software, or Windows itself. If you aren't careful with hardware settings stored in the registry, it could even result in catastrophic hardware failure. Successfully editing the Windows registry is not difficult, but you need to make sure you are paying attention to detail and all your changes are correct as you do it. ■

Standardizing colors

Looking at the System Options from the top down, the first System Option category is Colors. Setting up your own SolidWorks interface is a highly personal thing, but when you do it, you need to do so within some rules that enable everyone to work on your models.

A simple way to handle color standardization is to set up several color schemes and distribute them to users. Color schemes enable you to establish the combinations that users put together. In addition, remember, Document Properties also has a section on Colors. Also, remember that in SolidWorks, your background color (controlled in System Options) has no relationship with your text color (controlled in Document Properties) as you may be used to in AutoCAD.

If you have a document standard that says that your text must be black, and a user decides that he wants his background to be black, well, you've set up an unusable situation. Should you also ban users from using certain color backgrounds? Do you specify exactly which colors can be used or do you say, for example, "any color with a Lum value of more than 150 is usable"?

Part II: Installing SolidWorks

The safest thing to do is to standardize on the SolidWorks default colors, because these colors are referred to in the SolidWorks Help and Training documentation as well as other publications, such as this one. One difficulty with this is that SolidWorks Corp itself changed the default colors for selected items and dynamic highlight in the SolidWorks 2008 release.

Another aspect of this that you need to understand is the relationship between Scenes and background color. The Scene is a document property, but the background is a System Option. Scenes override backgrounds. This makes administering color even more difficult. Still, the safest thing to do is to stay with the default colors for the most part.

With the addition of Scenes, SolidWorks also added a setting to the Colors page of Tools & Options called Use document scene background, and turned this setting on by default. If your design process is highly sensitive to colors or viewport backgrounds, you will have to turn this setting off on every older part or assembly brought forward, and make sure it is off when you create new parts.

Tip

Sometimes you need special background colors for special purposes. For example, for the screen shots for this book, I am using a white SolidWorks background and removing all Scenes. A common technique if you want to use a tool like Photoshop to make a background transparent is to make the SolidWorks background some color not found on the subject, like a bright green or yellow, and then remove that color. ■

Leaving users with some options is a good plan. Being too rigid won't score you any points, regardless of whether you are strict or lenient. The best policy may be to lean toward practical implementations of commonsense rules.

Standardizing Sketch settings

The Use Fully Defined Sketches option is an option with which former Pro/ENGINEER users are probably familiar. Pro/ENGINEER had the option to force users to define sketches fully before the software would allow them to continue. I'm sure this option was added to SolidWorks because many of the people who were involved with the early stages of SolidWorks development were also involved in Pro/ENGINEER development.

Pro/ENGINEER enforced some things we have come to know as best practice. When SolidWorks was developed, people realized that you could model without carrying around all the additional baggage of the rigid rules that Pro/ENGINEER seemed to force on users. Best practice rules can be onerous. They slow you down and force you to be more careful. Therefore, when SolidWorks was developed, the developers left out all the enforced best practice and just let users model however they wanted to. Of course, most users loved this freedom, but many Pro/ENGINEER users scoffed at the seeming lack of discipline.

Little did developers know, but eventually, SolidWorks users came full circle to embrace those same best practice rules of which they were so eager to free themselves. When you start using SolidWorks with all the best practice rules that people develop, it starts looking more and more like Pro/ENGINEER with a better interface.

In any case, the Use Fully Defined Sketches option forces you to define sketches fully before it lets you close the sketch. Everyone that you talk to will tell you that fully defined sketches should be part of any thorough 3D CAD standard, but very few people that you talk to are using this option. Why is that? I think it is because there are times when rules are just too rigid. Is it important to lock down the length of a construction line?

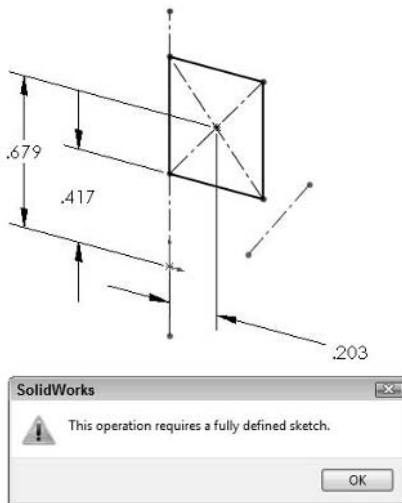
Note

The truth is that the under defined ends of a construction line do not leave a sketch with the under-defined title. If a construction line's position or angle is under defined, it is reported as under defined, and if the Use Fully Defined Sketch option is selected, SolidWorks will not let you exit the sketch. ■

Figure 8.5 shows you the warning that SolidWorks gives if you have the Use Fully Defined Sketches option selected and try to exit a sketch.

FIGURE 8.5

The Use Fully Defined Sketch warning



Standardizing Performance settings

The Verification on Rebuild setting Performance setting can have a huge effect on other users, and you need to have a policy on its use. Under normal error checking, SolidWorks checks every face in your model against every adjacent face to make sure the faces intersect only at edges and don't penetrate one another. With the Verification on Rebuild option selected, SolidWorks instead checks every face against every other face in the same body to make sure that they don't intersect except at edges. As you can imagine, this is a much more thorough check, but might have huge effects on rebuild time. The setting does not travel with documents; it remains on the system

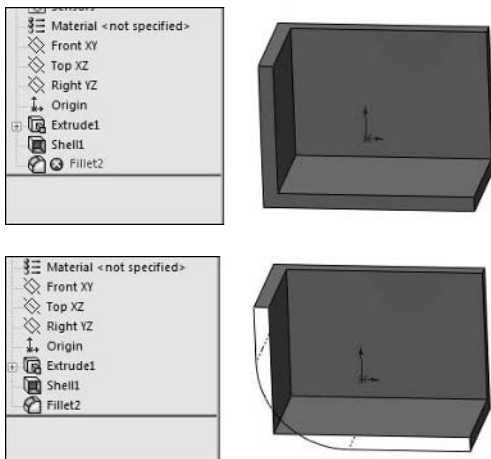
Part II: Installing SolidWorks

where it was set, but the parts that are checked, or more importantly not checked, travel between systems. If you have complex shapes, and especially if your models have any behavior that makes them suspect, you need to check parts with the setting selected at least once after all changes and before the part data is sent out to other users. If this setting is deselected, it is possible that a geometry error could get through even though no error shows in the Feature tree. To use the option, select it, and with a part open that you want to check, press Ctrl+Q. If no errors show up in the FeatureManager, the part has passed the test. Failing the test would mean that some feature that didn't fail with the option deselected fails with the option selected.

An example of a part that SolidWorks can create with Verification selected, but fails when Verification deselected, is shown in Figure 8.6.

FIGURE 8.6

On the left is the part with the Verification option selected; on the right is the part with the option deselected.



Controlling External References

If you are a SolidWorks Administrator in charge of a team of SolidWorks users who need to collaborate, you need to consider controlling some of the settings in the External References page of Tools ➤ Options. I'll go through a few of them to show you why it is so important in this section.

Open Referenced Documents with Read-Only Access

The first setting is called Open Referenced Documents with Read-Only Access. This is one of the most important settings when you are managing files manually across a network. When this setting is selected, and you open an assembly, all the subassembly and part files are opened with read-only access (unless any of them is already open with write access). When you have several people working on the same pool of files, it is important to avoid situations where one user opens a top-level document and winds up locking all other users out of the rest of the files.

Don't Prompt to Save Read-Only Referenced Documents

The second setting goes hand in hand with the first, and is Don't Prompt to Save Read-Only Referenced Documents. When you get on one of those loops where you have opened a large assembly with read-only access, and it is an older version, SolidWorks will identify every part as changed and requiring a rebuild and save, and will prompt you for this for every part. This setting simply enables you to avoid this frustrating situation.

Allow Multiple Contexts

Allow Multiple Contexts is a setting that you need to control very closely. It is possible to misuse this setting simply to allow sloppy modeling, or more precisely, sloppy file management. Many users and even SolidWorks Administrators have an irrational fear of in-context references. It is certainly advisable to have a healthy respect for this type of technique, because if it is used poorly, it can cause a lot of trouble with file management, and even prevent you from putting parts into certain assemblies. None of this is voodoo or even bugs in the software; it comes from file management rules that work and can be understood. In-context design techniques are certainly valid, and you shouldn't avoid them simply because you don't understand them or someone has told you a tale of woe, the upshot of which was to avoid in-context references.

The truth is that in-context references and other types of external references are powerful techniques to enable you to do types of design that otherwise would be very difficult or impossible. This type of reference enables you to keep parametric links between separate part documents.

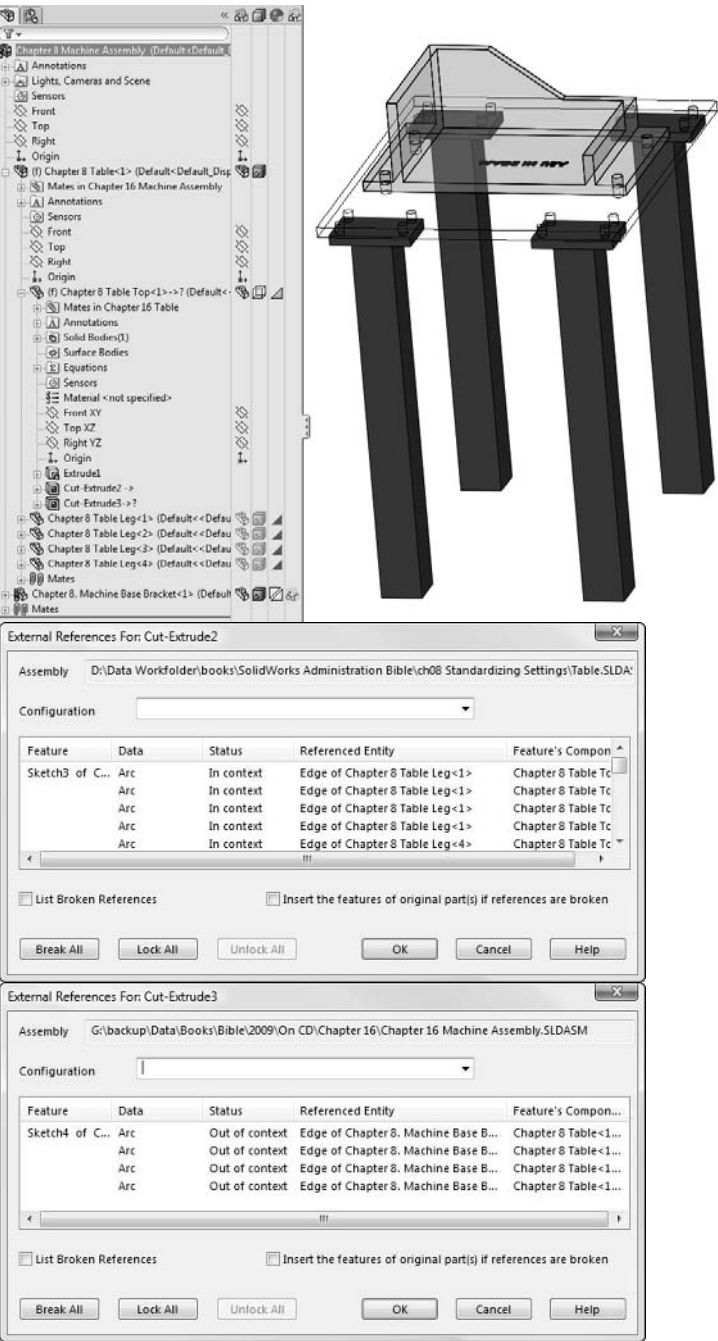
Allow Multiple Contexts has some specific purposes. One is when you have a part that has an in-context relationship in a subassembly and needs another relationship in an upper-level assembly; the part needs to be able to accept in-context references from multiple assemblies. SolidWorks doesn't do this by default. By default, a SolidWorks part can only accept references from a single assembly.

Examples of when sloppy modeling has lead to desperate users selecting the Multiple Contexts switch include when users put a part that already has in-context relations into an assembly (which may be either named or unnamed) and get a warning message telling them there may be a potential conflict. People who know about this setting may select it just so they can put the part into the assembly. However, the best method is to edit the feature with the in-context relation and edit out the relation, or simply put the part into the assembly in which the relation was created in the first place. You may have reasons for either selecting or not selecting either of these options. Make sure that fear of the unknown or misinformation is not one of them.

Figure 8.7 shows a case of multiple contexts where the tabletop has in-context relations within the table subassembly, but it requires more in-context relations in the top-level assembly when the machine base is placed on the table, and these relations are made within the top-level assembly. The External References dialog boxes show in the top row Assembly field that the relations for Cut-Extrude2 are pointed to the Chapter 8Table.SLDASM and the relations for Cut-Extrude 3 are pointed to Chapter 8 Machine Assembly.SLDASM. To create or edit this situation, you need to have the Multiple Contexts option selected.

FIGURE 8.7

An example of multiple contexts



Search File Locations for Referenced Documents

Too many times setting names sound like you need a lawyer to interpret them for you, but that isn't the case with this particular setting. As the name implies, when this option is selected, when you use SolidWorks to open an assembly or other document with referenced documents, SolidWorks looks for those referenced documents in the paths established at Tools ⇨ Options ⇨ File Locations ⇨ Referenced Documents.

Earlier in this book, I mentioned the thirteen different locations where SolidWorks looks for files referenced by an assembly or a drawing. The paths identified when you choose Tools ⇨ Options ⇨ File Locations ⇨ Referenced Documents are the second location. The list of these thirteen search locations is in the SolidWorks Help under the heading “Referenced documents, search for.”

This is important to know, because if you want to push a path to the beginning of the search routine, the Referenced Document paths are a great way to do it.

Update Out-of-date Linked Design Tables

However, you decide to use this option, all your users should probably use it the same way, especially if you frequently use externally linked design tables. If you are accustomed to receiving a prompt when an external linked file has been changed, and no longer receive the prompt, it may cause problems.

Automatically Generate Names for Referenced Geometry

The Automatically Generate Names setting is deselected by default, and I recommend you leave it that way for all users. When the option is selected, SolidWorks creates special names for faces used in mates and other types of features. This can be handy if you replace the part with another part, and then manually rename the faces (using the Face Property options from the right mouse button menu), but to do that, you need to have right access to the part. With the setting deselected, SolidWorks just uses the automatically generated face IDs, which can be accessed if the part is read-only.

Update Component Names when Documents are Replaced

This option is selected by default, and again, I recommend you use this default for all users. With this setting selected, the filename will be used in the FeatureManager, and if you replace a component, the name will be replaced. With the setting off, the original name used in the FeatureManager remains, even if the component is replaced.

Do Not Create References External to the Model

This option originated as a switch in the interface. There is even a toolbar button for this option. Some of the in-context misunderstandings have carried over to this particular option. You may hear people recommend that you always use this option because external references are somehow evil. This point of view is completely irrational. External references provide a completely valid way of working, whether you are talking about inserted parts or in-context references. Nothing is intrinsically bad about external references.

Part II: Installing SolidWorks

You should use this option when you want to avoid in-context relations, but want to position items as if they did have the relation. For example, if you turn the select the setting, select an edge in context (select an edge from part 2 while editing part 1 in an assembly), and then select the Convert Entities tool, the edge will be converted to a sketch entity, but it will not get the customary On Edge relation. When you use this tool, you should also manually constrain any sketch entities.

If you use this option when you do a knit or offset surface, the knit or offset will be created with a broken reference. The symbol for a broken reference looks like $\rightarrow x$. The external reference is \rightarrow . A broken link cannot be repaired or re-linked; the only thing you can do with it is to edit the model to remove the reference. For offset and knit surfaces, there is nothing you can do. Those types of surfaces require a reference, and they cannot be reattached.

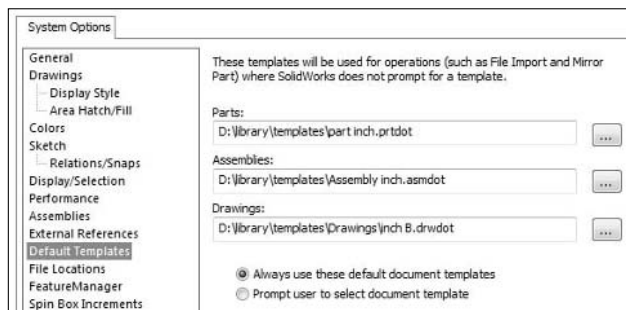
For these reasons, I don't recommend using the Do Not Create References option when creating in-context surface references. You should learn how to control external references. Using Lock references is one option. Figure 8.7 shows the List External References dialog box, which shows external references for a feature, which assembly they were assigned in, and enables you to lock or break the references. Breaking external references does not have any advantages over locking them, and locking is reversible, while breaking is not. I wish the Break option were removed from the software. Some people have come to believe and even tell other people that breaking references is the most appropriate thing to do with external references. It is simply not true.

Using Default Templates

I cover creating templates for use in SolidWorks in the next chapter, but here, I still need to discuss how to set up the default templates that are used when the user does not select a template to be used to create a new file. Templates in their own way are a type of default; they are like choosing from one of a number of groups of default settings. Selecting default templates is just like a default, if that makes any sense. Figure 8.8 shows the Default Templates option page from Tools \rightarrow Options.

FIGURE 8.8

The Default Templates settings



When you create a new part, you must select a template. If you have been using SolidWorks for some time, you have probably developed several templates for parts, assemblies, and drawings. You may run into situations, however, where you are not prompted to select a template, even though the software is creating a new document. This can happen for imported parts, split parts, in-context parts, and so on.

In cases where SolidWorks automatically selects a template for you, you can actually specify which templates are selected for the three types of documents: parts, assemblies, and drawings. To find the settings, choose Tools ⇨ Options ⇨ Default Templates.

As an Administrator, it may be a good idea to store all the company's templates in an accessible network folder and to set these three paths to the appropriate templates. If you have a large selection of purpose-specific templates, it is probably best to choose the most generic templates for use as default templates.

This area also offers you the choice between two options: Always use these default document templates, and Prompt user to select document template. The names should be self-explanatory. If you import assemblies with large numbers of parts, it may be advisable to use the Always use these default document templates option rather than the Prompt user to select document template option. Always use these default document is automatic, and Prompt user to select document is manual. If you import an assembly from STEP format that has 100 parts, and you use the Prompt user to select document template option, SolidWorks will stop you for every part and ask you what template to use for it, even before you get to see the part.

CAD Administration is all about little decisions like this that add up to a total usability situation for your users. Automating things when practical, and offering clear options when not, is a good rule of thumb to follow.

Specifying file locations

Figure 8.9 shows the list of locations in which SolidWorks wants you to place certain types of data. You will probably not use all the file locations, but depending on what type of work your company does, you may use many of them. Once you get the data set up for one user, you can copy the settings to other locations. Settings are probably best copied en masse using the Copy Settings Wizard, found in the SolidWorks section of the Start ⇨ Programs menu. You can single out individual settings, but it is more work and can be somewhat dangerous.

The File Locations settings are important for the SolidWorks Administrator to establish library locations and shared areas for files. Specifically the Referenced Documents setting can be very useful if you want to push a folder to the top of the list of thirteen search locations mentioned earlier in this chapter.

You can add multiple paths to each of these data types, so your part library can come from multiple network locations. Blocks could be stored in one local and one network location. The order of the paths matters in some cases.

Part II: Installing SolidWorks

FIGURE 8.9

Types of data that you can link to specific file locations

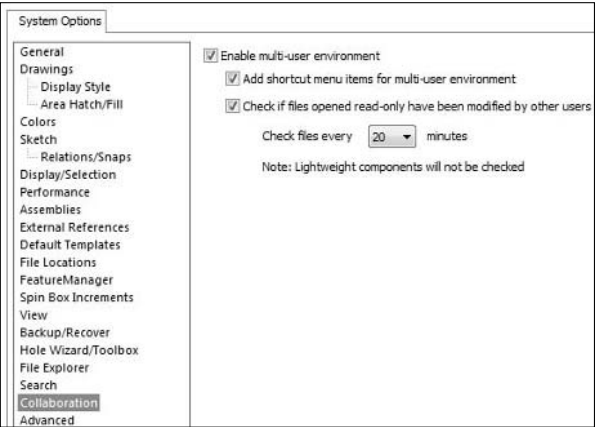


Collaborating

The Tools⇨Options⇨Collaboration page is another must-use for those of you managing situations in which SolidWorks users are manually sharing data. The available options are all meant to help you share files with other people without getting in one another's way and enabling you to see when files that you need to use have been updated. Figure 8.10 shows the Collaboration option in the Tools⇨Options dialog box.

FIGURE 8.10

Managing file settings from the Collaboration page



Cross-Ref

The settings on this page must be mixed with best practice techniques to form a stable, usable, manual file management environment. Getting this right is not a matter of flipping a couple of switches. Everyone involved must understand the tools. I address the collaboration topic thoroughly in Chapter 17. ■

Standardizing Modeling Practice

Standardizing modeling practice sounds like a formidable topic until you start thinking of how much of this work has already been done for you in the form of best practice documentation for various types of modeling work. Most modeling standards can be related to simple commonsense best practice. It may be difficult sometimes to separate best practice from urban legends when it comes to some SolidWorks functionality, such as the external references myths discussed earlier in this chapter. When you are guiding a group of users, you need to rely on researched information rather than gut instincts. Some answers may be difficult to find, such as why mates often seem to flip for no reason that you can discern, or why the results of a Trim Surface will often flip. In cases like this, where accurate information is not forthcoming from SolidWorks directly, and no reliable user-based source has stepped up with a reasonable explanation, all you may be able to rely on is instinct. Situations like these should be rare. You cannot always guide users around potholes in the road like these, and often new versions of the software will introduce unexpected bugs and quirks.

Standardizing for productivity

For these reasons and others, many CAD Administrators often recommend a very conservative approach against keeping up to date with the latest software. Sometimes the unexplainable bits that aren't cleaned up in new releases are simply not worth the lost predictability and productivity that you would otherwise have with a tested and fully patched version. The discussion about version upgrades is a long way from the discussion of standardized modeling practice, so this is as far as I will go down this road at this time.

Establishing a best practice modeling standard

Establishing a best practice standard is a Holy Grail of SolidWorks Administration. If you could create a standard that would work for everyone, you would have achieved something great. Unfortunately, a standard that satisfies everyone does not exist. This is one of the reasons that best practice recommendation lists are so elusive. Everyone has a different idea of what best practice actually is. For almost every situation you can name where you can identify a practice that you think should be universal, someone else will come up with a situation in which you can't apply that practice as a standard.

For that reason, I can't offer a definitive list here for you. Creating a list that works for everyone would require a list for every type of modeling and manufacturing method. Instead, I can offer some sample basic ideas that most users can apply to general modeling.

Relating the first feature to the part origin

One of the most basic things you can get your users to do is to relate the first geometry in the part to the origin. This makes them think about the orientation of the part in terms of Front, Side, or Top as well as about the symmetry of the part or features of the part about the standard planes.

Creating symmetry

Although symmetry was mentioned in the best practice related to the part origin and standard planes, it is often used as a generalized best practice to cover symmetry that is not about the standard planes. Users should always be looking for symmetry about planes or axes. This symmetry can be established using 3D features or 2D sketches, although features are generally preferred over sketches.

Also as a part of symmetry, remember that mirroring or patterning bodies is faster than mirroring features and mirroring features is faster than mirroring sketches. The idea that the “geometry pattern” option is faster for larger patterns is a myth dispelled in the *SolidWorks 2009 Bible* (Wiley, 2009).

Making relations to stable references

The concept of horizontal modeling as patented by Delphi is a summary of many best practice concepts centered on making relations to stable references with the goal of creating a parametric model that is stable through changes. The practice was developed on Unigraphics; however, the practice is applicable to SolidWorks and many other modelers. When translated into SolidWorks language, horizontal modeling says that when you refer between items in a part or assembly, you should only use the most stable references. The most stable references in SolidWorks might be listed as such (in order of stability):

1. Origin
2. Standard 3 planes
3. Planes created by offsetting a standard plane
4. Other reference geometry such as axes created from some combination of Origin and standard planes
5. Fully defined sketches on standard planes or offsets. 2D sketches have preference, but 3D sketches are also acceptable.

Other references are not considered highly reliable because they are based on items that could change too easily. For example, notice that sketching on model faces is not on the list. Notice that making dimensions to model edges is not listed. If you think about times when you have had to deal with rebuild errors on models that you made changes to, what types of failures did you encounter? Chances are that missing edges, faces, or even flipped sketch normals figure prominently in the list of errors.

Horizontal modeling is considered an extremely conservative way to model, but it does go far to help guarantee stability of a model. I have never met anyone that actually follows this method all of the time, but most of the best SolidWorks users I know use certain techniques from the method as appropriate.

Putting fillets at the bottom of the tree

One of the best practice recommendations for part modeling is to put your fillet features at the bottom of the tree. Putting fillets too high in the tree might make other features dependent on fillet faces or edges. Fillets are changed or deleted frequently, so referring to these is usually a bad idea.

A corollary to this is that you should avoid using sketch fillets. There are many reasons for this recommendation:

- You cannot suppress sketch fillets; however, you can suppress fillet features.
- If a fillet in a sketch causes a failure, the entire feature dependent on the sketch also fails. If a fillet at the end of the tree fails, it matters less.
- The sketch relations involved in solving sketch fillets and then creating a feature from the sketch are actually slower (compute time) than the fillet feature.
- Fillet features can create a much wider range of fillets than sketch fillets, regardless of the feature used to drive the sketch.

Using Verification on Rebuild

The Verification on Rebuild option should be used before declaring any model finished.

Fixing rebuild errors

You should always fix rebuild errors in parts as they appear. If you continue to build a model by adding new features on top of features with errors, SolidWorks continues to try to resolve the errors. Errors in the FeatureManager cause unnecessarily long rebuild times, and may result in the model changing geometry when the errors are resolved. Errors are always more difficult to resolve after you have forgotten what was going on in the model when the error showed up. Best practice says just fix the errors immediately.

Modeling detail

Avoid modeling excessive detail when you can. Excessive detail might include trying to model threads or knurls when these will be applied by a manufacturing process rather than being machined directly. When you design threads into plastic parts, you have to model the threads because you are probably not going to apply a threading process to the mold die; you will either machine the thread or machine an electrode to cut the thread. However, when you model a threaded rod, a threading die will be used to cut the threads, and having the actual 3D data for the thread does not help you.

Modeling this kind of detail is difficult to model and can slow down SolidWorks operation significantly.

Creating simplified configurations

When possible, create simplified configurations of parts and assemblies in order to have a version of the assembly where you can maximize performance, and possibly sacrifice some of the modeling detail. You do not need to see the detail all of the time.

Standardizing assembly modeling

Assembly modeling practice is also prone to the same types of errors and inefficiencies to which part modeling is subject. When establishing a standard of best practice for assembly modeling, consider some of the following suggestions.

Using subassemblies

When creating an assembly, SolidWorks only solves the top-level mates when the top assembly is rebuilt. Segmenting your assembly data into subassembly helps you compartmentalize troubleshooting and rebuild speed problems.

Limiting flexible subassemblies

When you need to create top level assembly motion between parts that are both in a subassembly, you have to use flexible subassemblies. Flexible subassemblies essentially solve the subassembly mates in the top-level assembly.

Mating to in-context features

When creating mates between parts in an assembly, it is best to avoid creating mates to faces of features that were created with in-context relations. When you do this, the assembly must solve the parts, then solve the mates, then solve the in-context feature, then solve the final mates. Iterating on solving the mates causes unnecessary slowdowns.

Limiting in-context modeling

In-context modeling makes relations between parts in the context of an assembly. The technique gets a bad reputation from users who have made mistakes with it, but correctly used, it is a very powerful tool for many types of modeling. Still, it is not for everything, and you should use it sparingly. Any modeling standard needs to make clear a position about in-context modeling.

Summary

Every company that uses 2D CAD has come up with a drawing standard to try to make drawings consistent from one project to another and between users. 2D standards are important, and the standards need to be followed as much as possible.

In 2D, you standardize the results, but when working with 3D CAD, you are instead standardizing the process. Part of the process is the settings. Some settings are important to standardize between users, and some are not.

3D modeling standards tend to be best practice suggestions rather than results oriented. Rules will be very difficult to enforce in the parts that require methods outside the normal to create. Best practice suggestions are more flexible than 2D standards. 2D standards can and should be followed precisely, but you cannot control 3D data in the same way.

Maximizing Your Use of Templates

One of the most important tasks you have as a SolidWorks Administrator is creating and updating templates for all the document types in SolidWorks for your users. Templates are not difficult to create, but the main part of the work is simply building in the settings and custom properties that will make your users' job easy. In this chapter, I pull together discussions from Chapter 8 on standardization of settings and Chapter 13 on custom properties to set up several of the issues that we will deal with in this chapter.

The most difficult types of templates to set up are drawing templates and formats, but by the end of this chapter, you will have several good examples as references.

SolidWorks software generates default templates if you ever find yourself in a situation where your regular templates are lost. You should never find yourself in a situation where you cannot move forward because you do not have a template file. You may not have the particular template that you want to use, but you can always make changes later on.

If you are an IT professional or otherwise don't have a lot of experience running SolidWorks, templates may be difficult for you to set up properly. You may need to enlist the assistance of some SolidWorks users to get their opinions of a few things. You can think of templates being "start files" for SolidWorks. Typically, you can have multiple templates for the three major SolidWorks file types: parts, assemblies, and drawings. Drawings also have an entity called the *sheet format*, which confuses many new users. A sheet format is a type of template for the drawing border.

Any discussion about templates requires a brief history lesson in SolidWorks. When SolidWorks software first came out, you would start a part, assembly, or drawing without selecting a template first, and then change settings from

IN THIS CHAPTER

Designing SolidWorks part templates

Constructing assembly templates

Setting up drawing templates

Setting up standardized sheet formats

Adjusting other types of templates in SolidWorks

Part II: Installing SolidWorks

there. There wasn't the concept we know as "templates" today. There was, however, functionality that used the template name. This is a source of confusion for many users even today. This was in the days when file extensions were still three characters (*.prt, *.asm, and *.drw).

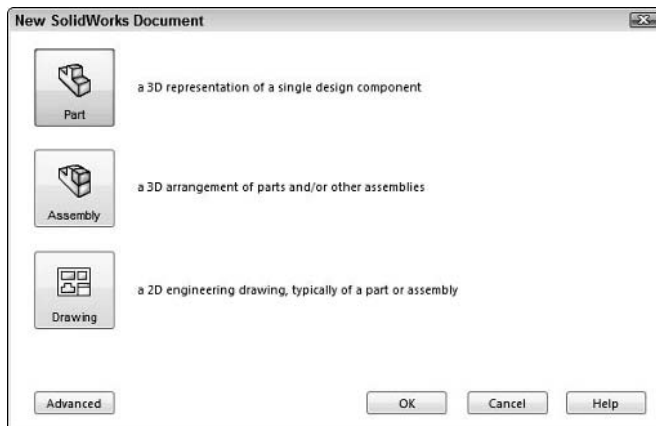
What we know as "sheet format" today used to be called "template." That is why the sheet format file type is called *.slddrt (drt = drawing template). When the change was made to use templates as "start parts," the old template document type became a "format." Formats are now known as the drawing border and associated annotations. So now, parts, assemblies, and drawings all have templates, which serve as start documents. Only drawings have formats, which serve as the drawing border. The template extensions are now *.prtdot, *.asmdot, and *.drwdot, where dot stands for document template. The formats (formerly called templates) are still *.slddrt. Therefore, if the extensions seem to be all confused, you now know the historical reason for it.

You select templates from the New Document interface. You can select the template you want to use, which has the part type, the settings, and some other information already set up inside the files. Inside the New Document interface, you can have multiple templates for each file type set up and ready to go.

The New Document interface has two modes: Advanced and Novice. The Novice interface is shown in Figure 9.1, enabling only one template for each file type. The one template enabled is from the Default templates list discussed in Chapter 8 as part of Default Templates (choose Tools ➤ Options ➤ Default Templates).

FIGURE 9.1

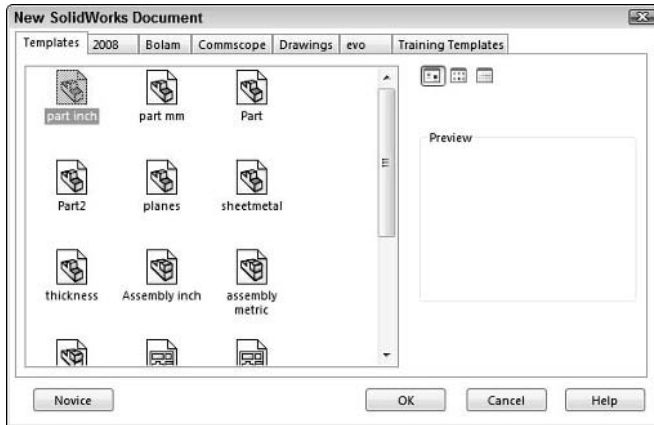
The Novice interface for creating new documents



At some point, you will outgrow a single template for each file type and want or need more options, especially for drawings. Reasons for needing more templates include starting parts already set up for different materials. Figure 9.2 shows the Advanced option of the New Document interface.

FIGURE 9.2

The Advanced interface offers more options.



Caution

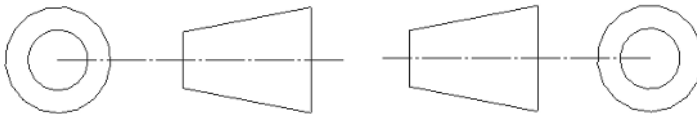
One of the things you may want to watch out for, especially when using the SolidWorks default templates available through the Novice interface, is that SolidWorks seems to prefer using the ISO (International Standardization Organization) Drafting Standard option and, along with this, can come the First Angle projection. First Angle projections are typical in Europe, and Third Angle projections are typical in the United States.

This difference is important because if a First Angle projection drawing is sent to a typical shop where a Third Angle projection is expected, you may have sent them impossible drawings or, in a worst-case scenario, incorrect parts may be produced. ■

Figure 9.3 illustrates the difference between First and Third Angle projections.

FIGURE 9.3

The difference between Third and First Angle projections, respectively



The additional tabs across the top are either additional paths entered into the Document Templates area (choose Tools ⇨ Options ⇨ File Locations ⇨ Document Templates) or subfolders within the paths specified there. You can have as many tabs as you like. In this case, I use the tabs either to separate types of templates or templates for specific customers. Within each of the tabs, I have a selection of different templates for parts, assemblies, and drawings.

Creating Part Templates

A part template is a regular blank SolidWorks part saved as a special file type. The SolidWorks part template file type is *.prtdot (dot stands for document template). To create a template, you have to use an existing template to make a new part or take an existing part and delete all the geometry from it. The types of things you can change in a template include:

- Custom properties
- Standard plane names
- Material properties
- Appearance and scene set up
- Configuration properties
- Document specific settings found under Tools ⇄ Options (such as sketch grid display)

Users frequently ask questions about settings found in part templates. Some of the issues are simple to fix if you only have to change the template, but if you have created many files from the templates, you may have to change all of those as well. For this reason, it is best if you can get your templates set up correctly early on. This will make for fewer adjustments and mistakes later on.

One of the most common questions I see users ask about templates is, “Can you change the template of a part after it has been created?” The straightest answer to that question is simply “No.” You can change all the settings that go into making the template, but you can’t change the template itself. Some tools exist to help you migrate custom properties or settings from one document to another, but these are only available from various sources scattered around the Internet and are not offered by SolidWorks.

You might need a number of different part templates to have starting places for parts with different units, or parts with different default materials. In Figure 9.2, you saw both inch and mm templates. Templates are certainly the easiest way to create starting points for parts in different units. You can also change the units after a part has been started.

Identifying the most important settings

Templates are meant to speed up the creation of new documents by grouping commonly used settings. To create the most effective templates, you must be able to identify accurately the most important settings for your users. One of the best ways to do this is to see which document specific settings users spend much time changing. One example might be units for documents. The standards that you establish need to be supported by the document templates that you provide, along with training. The most popular settings to change, or those new users ask about, should certainly be on your list to incorporate into your new set of part templates.

Drafting Standard

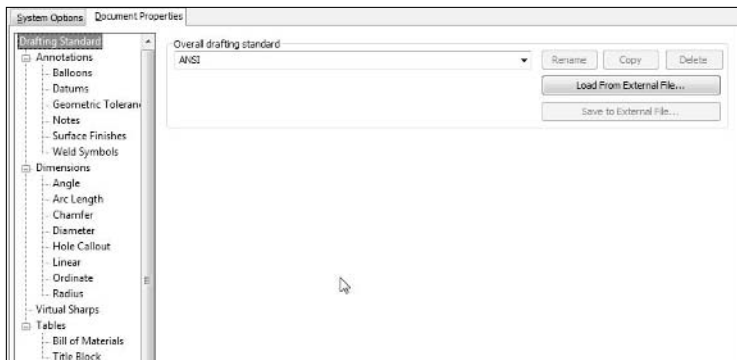
The Drafting Standard in part documents all the Annotations, Dimensions, Virtual Sharps, and Tables settings. For CAD Administrators, this is one of the most important developments in SolidWorks in recent releases.

Out-of-the-box SolidWorks installs with settings for several common international standards. Very few companies truly abide by a standard 100 percent; they usually mix and match aspects of various standards, whether for functional reasons or simply reasons of stylistic preference. SolidWorks Corp has finally realized that this is what people really want — customizability. They have realized that there is nothing they can do that will satisfy all SolidWorks users all the time. Therefore, you now have the ability to customize standards, save out the settings to an external file, and move those settings to other computers and load them there. If you have templates set up, you may not need this, but it is good to have the flexibility to move the standard independent of a set of document templates.

The types of settings that you can control through the Drafting Standard are shown in Figure 9.4. Many of these types of settings may not seem important in the part document, but when you consider the provisions for annotated models in the ASME Y14.41-2003 standard, these options may make more sense.

FIGURE 9.4

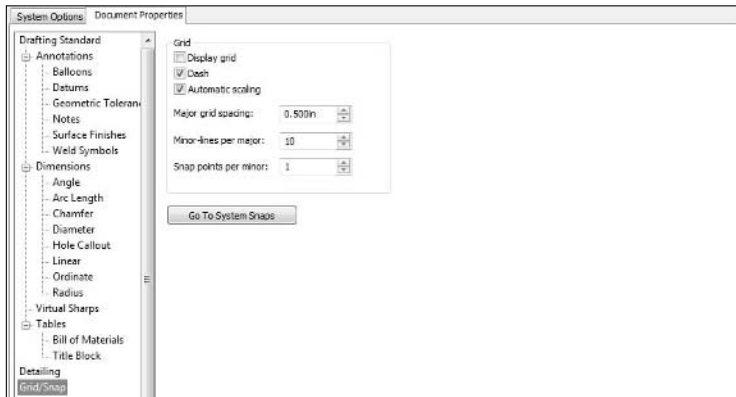
Many settings are controlled through the Drafting Standards settings.



You might wonder why 3D parts and assemblies would have such a large section of settings devoted to Drafting Standards. Part of the answer lies in a new standard from ASME (American Society of Mechanical Engineers), which creates a standard for annotations to be placed directly on the 3D model.

FIGURE 9.6

The Grid/Snap page is where you turn off the grid for document templates.

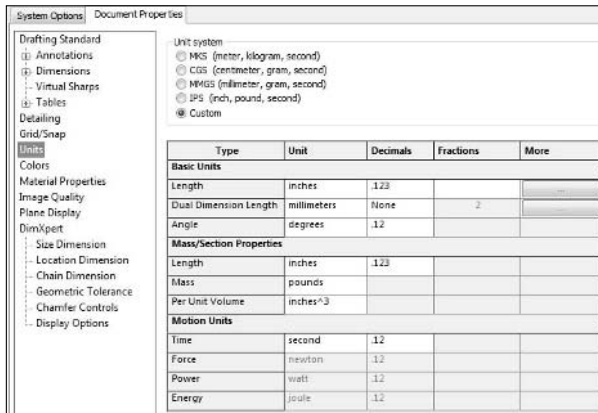


Units

Without a doubt, the most important setting in part templates is the unit setting. While it is very easy to switch back and forth between units for the document, it still pays to have templates with all the basic units that your company generally uses. The interface for the Units page has changed several times over the years, and now it allows you to set up the units for dual dimensions, as well as the units for analysis. You find the units settings by choosing Tools⇨Options⇨Document Properties⇨Units. Figure 9.7 shows this page.

FIGURE 9.7

The Units page offers settings for primary and dual dimension units.



Part II: Installing SolidWorks

Drawing documents can have primary and secondary dual units as well as part and assembly documents. Part and assembly documents can even use dual dimensioning as one of the template default settings. If you have been out of the SolidWorks loop for a few releases, you can now change the position of the dual units to any of the four positions (top, bottom, left, or right) of the main units dimension. You can also show the units text (such as in or mm) for both dimensions. You can find these settings (making dual dimensions default in the part/assembly and changing the position of the dual units) by choosing Tools⇨Options⇨Document Properties⇨Dimensions.

Detailing settings

The detailing settings defaults are generally acceptable for most modeling tasks, but sometimes you need to make some adjustments. The Detailing page of Options (choose Tools⇨Options) is shown in Figure 9.8.

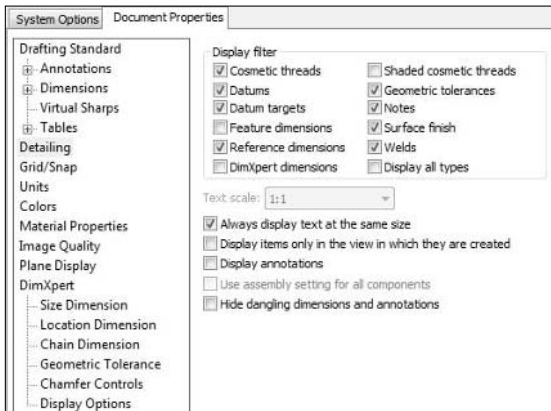
Always display text at the same size is definitely a setting you want to have selected. When it is selected, it prevents dimension and annotation text from changing size as you zoom.

Display items only in the view in which they are created is an interesting setting for developing 3D drawing views, but most people do not have the discipline to use this setting. If you ever apply dimensions while not in a view square to the model, don't use this setting.

Hide dangling dimensions and annotations is a relatively new setting. SolidWorks gets more and more error-phobic as more users from AutoCAD are recruited. I think it is very useful to know when things are broken rather than just have them disappear on you. I recommend you leave this setting deselected unless you have many errors in your model or the boss is looking over your shoulder and he is a SolidWorks user, too.

FIGURE 9.8

The Detailing page controls the display of various annotation types in all document types.



Controlling document specific colors

I recommend changing only one setting on the Colors page for your part template. This is something you may want to consider or try out before you make it final. The Apply same color to wireframe, HLR and shaded setting becomes important in an assembly where you view your parts in wireframe. When using default settings in an assembly, all parts shown in wireframe are black. This doesn't seem like a very helpful default, especially in a program such as SolidWorks that is capable of so much more. With this setting selected, the wireframe color for a part is the same as its shaded color. This is a huge benefit in assembly model, especially in recent versions, where you can also use Display States to combine shaded and wireframe displays.

Some users look at wireframe display as an old-fashioned tool, but I find it helpful when you are primarily interested in edges in the assembly or using it in lieu of hiding a part in an assembly, because wireframe is different from hiding and transparency.

Image quality controls display accuracy

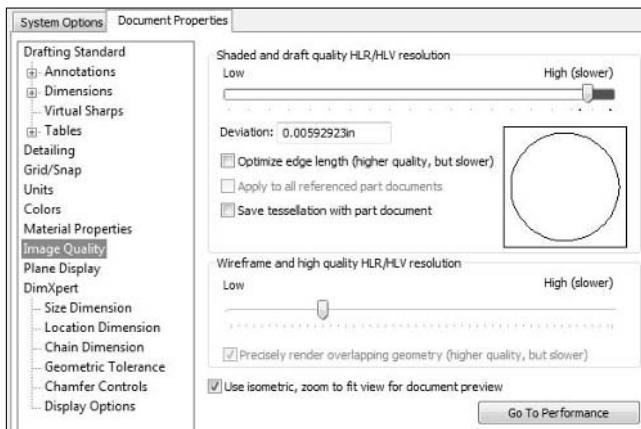
Unless you work with small assemblies or you have unlimited graphics power in your hardware, you should probably be careful of your Image Quality default settings for templates. The shaded quality slider should probably be set about one-third of the way from left to right. You should also not have the Optimize edge length option turned on.

The Save tessellation setting can be important for downstream applications that use SolidWorks files directly, such as eDrawings and PhotoView360. Selecting it does make for a larger file size, but also enables functionality in these other applications.

Figure 9.9 shows the Image Quality page (to find it, choose Tools ⇨ Options).

FIGURE 9.9

The Image Quality page controls the accuracy of the shaded and wireframe displays.



Controlling plane display parameters

The Plane Display page has a couple of obscure options. In the Faces box, it asks you to select front and back colors for the planes. The first thing that is odd here is that it doesn't show the plane color next to the button used to select the colors. Second, what is going on here?

It turns out that other modeling software also enables you to create different colors for opposite sides of planes. Planes and faces have a mathematical identifier known as a “normal.” This vector always points to the front or back of the plane. However, how do you tell the difference between the front and the back of a plane? Well, in SolidWorks, you can't, except for the color. Why is it important to know this kind of information? To most people it might not be important at all, but as CAD Administrator, you will want to know the answer to this question when a curious user comes over to your desk.

Here is an example to try. Sketch a rectangle on the XY plane of a new part and extrude it in the default direction. Now flip the view over and select the face of the block you just extruded that lies on the XY plane. Draw another rectangle and extrude that in the default direction. Notice how the default extrude direction for the sketch on the face is in the opposite direction from the default extrude direction for the sketch on the plane? You can always assume that the default direction for a solid face is to extrude bosses away from the body and cuts toward the body. However, you can't tell which is the default direction for a plane unless all these criteria are satisfied:

- You have plane display turned on in the View menu.
- The plane itself is set to visible.
- The Display Shaded Planes setting (choose Tools ⇨ Options ⇨ System Options ⇨ Display/Selection) is selected.
- You have the Plane Display setting (choose Tools ⇨ Options ⇨ Document Properties ⇨ Plane Display) Front and Back faces turned to different colors. The Front face color is the one that determines which direction sketches will extrude from.

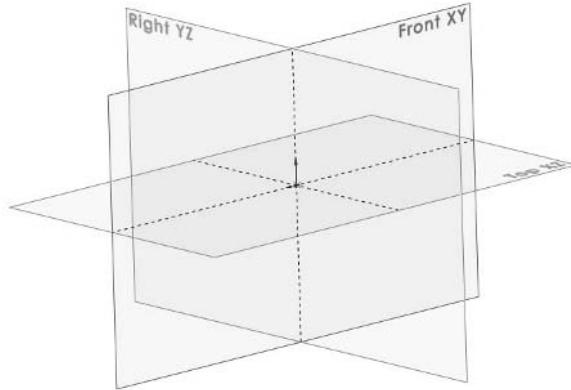
Note

A discussion on which side of faces or planes sketches extrude from by default seems like a trivial or academic matter until you start changing the sketch plane for a particular sketch and the sketch flips on you. The idea of plane and face normals is something you should at least be familiar with. ■

The second setting available in the Plane Display page is the Show Intersections. This may be difficult to see in gray scale shown in Figure 9.10, but the Front plane is shown in red and the other two are shown in green, and intersections between all the planes appear as dotted lines. If you were to make an offset of the Right plane, it would show the same color as Right, and it would show intersections with the Front and Top planes.

FIGURE 9.10

Plane intersection lines are displayed as dotted lines.



This is an interesting setting, and possibly useful in certain situations, but like the sketch grid settings, I would turn it off for doing real work.

Using custom properties

Custom properties are text-based metadata that you can put into your SolidWorks part templates. Getting the right properties set up for use in drawings and file management can be a huge time saver through automation available in linking annotations to properties, Bills of Materials (BOMs), and other functionality that allows access to custom properties. Chapter 13 is entirely on the topic of custom properties, so I won't go into this in much detail here, but setting up custom properties is an important part of template building. Figure 9.11 shows the basic interface for entering custom properties, which you access by choosing File ⇨ Properties.

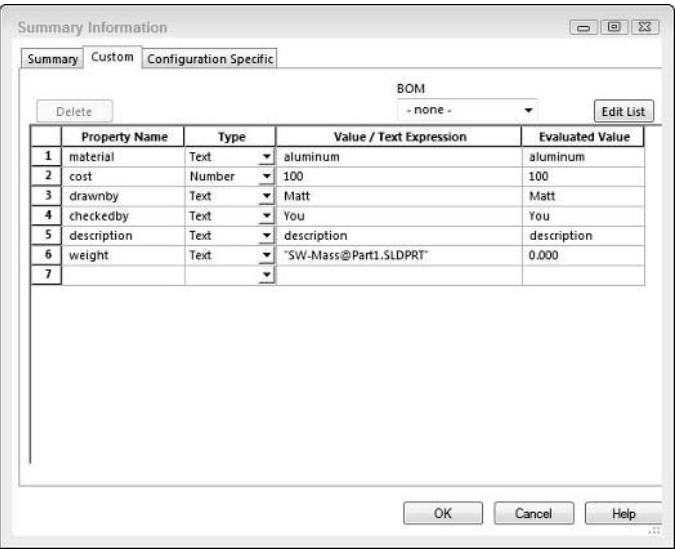
The properties shown are ones that I have used for some time as default settings. The templates I use regularly in my own modeling are ones I created several years ago and bring forward to each new version of the software.

Cross-Ref

In Chapter 11, I cover libraries, and how putting your templates in a library that is central to all users and brought forward from one release to another is a great way to ensure continuity for your users and for your company's documentation practices. ■

FIGURE 9.11

The basic custom property interface offers a grid format for adding properties and values.



Finding hidden settings in part templates

Several settings are not accounted for by any of the items I mentioned previously. These settings may be more obscure, but can be equally helpful.

Link values and global variables

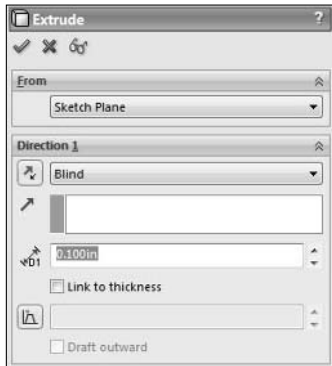
If you create a blank part from a template, make a sketch, add a dimension, then create a link value or a global variable, and then delete the sketch and save the part as a blank template, those variables will be built into all parts made from the template.

For example, if your company makes many cast parts with standard draft of three degrees, you can build that global variable into your template, and then link draft values to the global variable. This is a great example of a best practice that you might consider using at your company. Using link values and global variables can save you a lot of time and energy.

One particular link value that has special significance is the link value “thickness.” If you have a template with a link value of that name, all your Extrude dialog boxes will have a check box for the depth of the extrusion called Link to thickness. This is great for fixed thickness types of parts such as sheet metal, plastics, and castings. Figure 9.12 shows the Extrude PropertyManager of a part started from a template with the thickness link value.

FIGURE 9.12

A PropertyManager with Link to thickness option

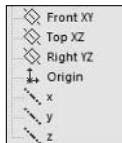


Adding reference geometry

One example of adding reference geometry to a template is to construct three directional axes from the three standard planes. This can be extremely useful in features where you have to specify a direction frequently, such as patterns and many plastics features. Figure 9.13 shows the FeatureManager of a template part where this has been set up.

FIGURE 9.13

Using additional reference geometry in a part template



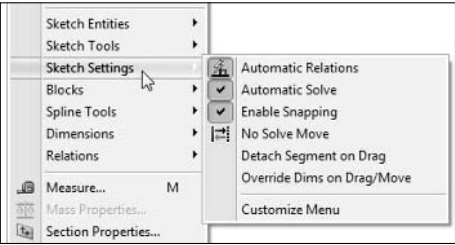
Including sketch settings

You can incorporate some of the more obscure sketch settings into part templates. You can find these settings by choosing Tools ➤ Sketch Settings, as shown in Figure 9.14. You will probably want to keep most of these settings as they are, but for conceptual parts, it might be good to have the Override Dims On Drag option selected and Automatic Relations deselected. They are just additional options.

Part II: Installing SolidWorks

FIGURE 9.14

The Tools⇨Sketch Settings menu selection allows easy access to sketch settings.

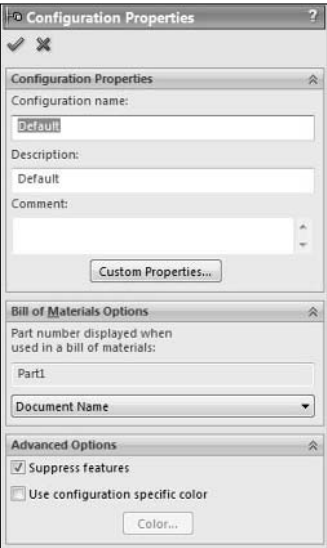


Set configuration properties in your templates

Many users forget about the configuration properties until they have already created a few configurations that are not behaving the way they want them to. You need to consider all these options and add them to templates early on. It is tough to change course midstream with functionality like this. Figure 9.15 shows the Configuration Properties PropertyManager.

FIGURE 9.15

The Configuration Properties PropertyManager controls configuration settings in templates.



When you add a new configuration, SolidWorks uses the Advanced Options from existing configuration in the new configuration.

Creating Assembly Templates

Assembly templates are useful primarily for establishing assembly units, and for other types of data. Assembly units are the most important of these because if the assembly units are not the same as the part units, you will see mixed units on the screen when you edit part dimensions and mate dimensions in the assembly. Other types of settings that you can control in an assembly template are the default reference geometry, custom properties, default configuration properties, and so on. Material settings are not applicable to assemblies, but lighting, annotations, and sensors are.

Assembly templates have fewer options than parts. Assembly templates are probably most important when you are editing parts in the context of the assembly. To create an assembly template, you go through the same basic steps as when creating a part template:

1. **Open an assembly with settings you like.**
2. **Remove all referenced documents from the assembly.**
3. **Adjust settings.**
4. **Save as template.**

Assemblies, like parts, have several common settings that users prefer. In this section, I show you some of the more popular settings and offer recommendations.

Drafting standard

Drafting standards in assemblies may seem even more out of place in assemblies than in parts, but they are used to control the display of annotations, dimensions, and tables within the assembly document. By customizing a standard and sharing it between various document types, you can create and maintain a consistent style between parts, assemblies, and drawings.

In assemblies, the concept of setting up a drafting standard works the same as in parts. If you want to annotate 3D assembly models per ASME Y14.41-2003, you can use an appropriate drafting standard to do so. The ANSI standard is probably your best bet to start. You can save adjusted drafting standards external files and share them with other users, again, just like for parts.

Setting detailing options for assembly templates

Assemblies have most of the same settings as parts; you can refer to the section on part templates for most of the settings. One important exception is the Use Assembly Setting For All Components option. This setting causes assembly settings to be applied to any parts referenced by the assembly. This applies only to the settings in the Display Filter area. A similar setting is found in the assembly document properties Color page, but the two settings are not related.

Important settings on this page are the Always display text at the same size, which keeps the displayed size of annotations and dimensions constant regardless of zoom state, and the cosmetic threads setting, which can allow cosmetic threads to display through solid parts. Feature dimensions option will allow part feature dimensions to be shown in the assembly, which can overwhelm the screen even with a small assembly.

Establishing image quality settings for assembly templates

The only setting I will point out here is the Apply To All Referenced Part Documents option. When editing an assembly, I typically want low quality so the display is faster, but when I work on individual parts, I want a better quality display of details. If you have the Apply To All Referenced Part Documents option selected, then when a part is viewed in an assembly, it will take on the image quality setting of the assembly, which is typically low, and when you go back to your part, you have to adjust the image quality. There is no way to get it to work in low resolution in assembly and high resolution in parts, because the quality slider is deactivated unless the Apply To All Referenced Part Documents option is selected.

Establishing Drawing Templates

For users who create drawings as the deliverable for your product documentation, drawing templates are the most important of the three types. The templates can include automated features such as population of annotations from part or assembly custom property values, automated view creation, placement and sizing, and automatic selection of format size.

Drawing templates (and even more so formats) are critical parts of your SolidWorks implementation setup. A decision on how much automation you intend to build into your templates is up to you. Granted, building templates with many automation features is one thing, while training users to use them is actually an entirely different matter.

Understanding the difference between drawing templates and sheet formats

Because the difference between drawing templates and sheet formats is a frequent user question, I'll cover it again. Templates control drawing settings, and formats control the drawing border. A template can be saved with or without a format already in them. Once a format is in a drawing, the format can be replaced. Once a template has been used to create a drawing, the template *cannot* be replaced.

When you start a new drawing, you start by selecting the template. If the template does not have a format, you are asked to specify one. If it does have a format, SolidWorks skips that step. It is useful to have different drawing templates with different formats. I use drawing templates primarily for sheet size and units.

A more detailed comparison between templates and formats may also be useful in understanding the differences and when to use which type of data.

The drawing template includes:

- All the Document Properties settings for the drawing
- Any predefined or projected views (placed on the sheet)
- All Sheet settings (such as First or Third Angle projections) and sheet size and scale
- Multiple sheets, if applicable
- Any standard annotations that you want to have on the sheet
- Title Block definition
- May or may not contain a sheet format for each sheet
- An Excel table for Excel-based revision table

The sheet format contains:

- A drawing border sketch
- Table anchors for various types of tables
- Annotations linked to custom properties, including annotations used for Title Block functionality

Note

I cover the sheet format formally in the next section of this chapter, but mention it here for completeness. ■

Typically, tables are not stored in templates; they are placed onto the sheets after the drawing is created.

Identifying settings to control with drawing templates

Drawing templates control many settings that users and anyone who reads drawings care very much about, so it is important when establishing drawing standards to get as much agreement as possible from the various internal and external customers of the drawing documents. You will never please everyone all the time, and sometimes it can be difficult to know what the right thing to do is when you have a disagreement. Especially in those situations, you must make a choice based on rational reasons rather than on an emotional reaction.

In this section of this chapter, I cover popular and recommended settings for your drawings.

Drafting standard

Like both parts and assemblies, you can customize, save, and share customized drafting standards. To set this up, choose Tools ⇨ Options ⇨ Document Properties ⇨ Drafting Standard. All the headings under the Drafting Standard page go into making up the standard. If any one changes, then the standard listed on the front-page changes to ANSI [modified], for example. I detail all these settings in Appendix C, but will single out a few here for emphasis.

The headings under the Drafting Standard page include:

- Annotations
- Dimension
- Centerlines/Center Marks
- DimXpert
- Tables
- View Labels
- Virtual Sharps

Establishing template font settings

Fonts are not controlled on a single page. The control is distributed among several other items. If you are a longtime SolidWorks user, you may remember a time when all the fonts' settings were controlled on a single page, but that is no more.

The font for the drawing, including default font size, is controlled on the Annotations page. If you change the font setting on the Annotations page, the font settings for all the pages under Annotations also changes.

The pages listed under the Annotations page include:

- Balloons
- Datums
- Geometric Tolerancing
- Notes
- Surface Finishes
- Weld Symbols

The font for each of these items can also be overridden on the item's page.

When the Annotation font is changed, the font listed on the Dimensions page remains the same, as do all of the items under the Dimensions page.

Under dimensions, you will find the following items:

- Angle
- Arc Length
- Chamfer
- Diameter
- Hole Callout
- Linear
- Ordinate
- Radius

Chapter 9: Maximizing Your Use of Templates

The good news is there are only four locations from which all the fonts in SolidWorks drawings can be controlled. In addition, you can control each type of annotations, dimensions, and tables or view labels where fonts are used individually if you want to do that.

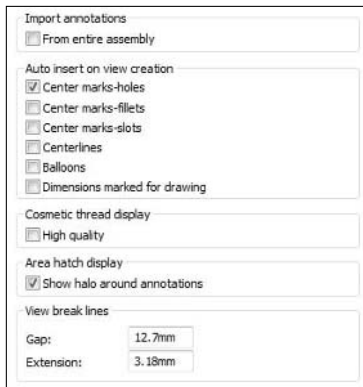
Remember that SolidWorks uses TrueType fonts, not single line fonts to which you must assign a line thickness. You achieve thicker fonts in SolidWorks using bold or filled fonts such as Arial Black. SolidWorks TrueType fonts are driven by the Windows font manager, not a special tool inside of SolidWorks. The printing of fonts is much more like Microsoft Word than AutoCAD.

Setting up detailing options for drawing templates

The Detailing page for SolidWorks drawings has more settings on it than the same page for parts or assemblies. The additional settings are shown in Figure 9.16.

FIGURE 9.16

The additional settings for detailing in drawings



The main settings here that you may want to keep an eye out for are the Show halo around annotations (when annotation is on top of a hatch), and the auto insertion of center marks, centerlines, balloons, and dimensions. Auto insertion, in particular, will be either popular with users or annoying. Either way, you will want to know where the setting is kept.

Setting units for drawing templates

Drawing units are one of the most important settings to apply in your template. In fact, it is probably one you will use to define the template itself, along with the size of the sheet or possibly a customer name.

Dual units on drawings, like parts and assemblies, can be made the default, can be placed on any of the four sides of the primary dimension, and can display units. Remember that if you make a change that deviates from the Drafting Standard you are using, you will need to re-save the drafting standard and redistribute it to anyone you have given your standard to.

Part II: Installing SolidWorks

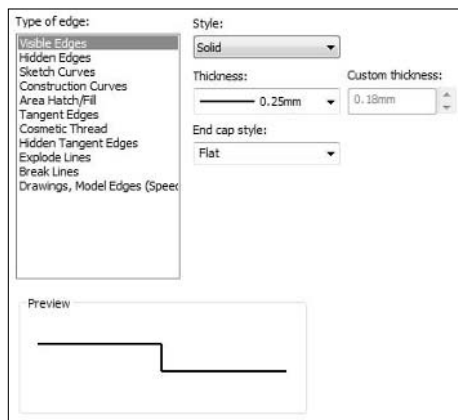
Establishing line font settings for drawing templates

Line font is not something that is typically changed in SolidWorks, although you can do it if you feel the need. This option page enables you to establish which type of line is used for which type of displayed edge. The settings here include line style as well as line thickness.

The line thicknesses are established on a later page of the Document Properties. Figure 9.17 shows the Line Font page of the Document Properties options.

FIGURE 9.17

The Line Font option page enables you to select how line fonts are assigned in template documents.



Note

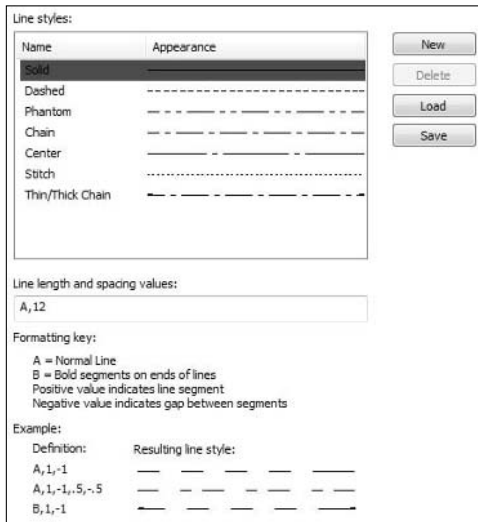
If you make changes to any of these standard settings dealing with the display of annotations, fonts, line types, and others, you should test the changes with all documentation types produced by your company. You should also test the changes with available printers and plotters, as well as electronic printers such as PDF and eDrawings. ■

Establishing line style settings for drawing templates

On the Line Style page, there is not much that you can do other than edit or create new line styles. You can create new line styles by following the formatting key shown in the lower half of the page, and entering in the line length and spacing values in the box in the middle. The syntax is a little archaic, but you should rarely, if ever, actually need it. If you do need to change something, you should only change it once, then save and load the new styles on other computers. Figure 9.18 shows the Line Style page of the Document Properties options.

FIGURE 9.18

The Line Style option page

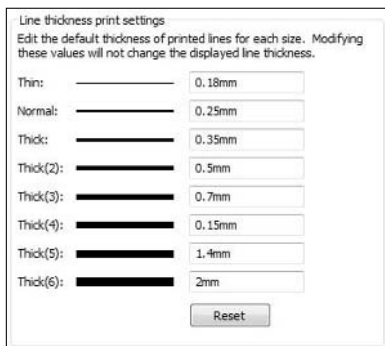


Line thickness

It is probably best to leave the line thickness values at their standard values unless you have a specific need. All changes to these settings must be tested for all company output, including but not limited to output devices: laser printers, plotters, and all export types: DXF/DWG, PDF, TIFF, and so on. Figure 9.19 shows the Line Thickness page of the Document Properties options.

FIGURE 9.19

The Line Thickness option page enables you to establish which styles and widths are assigned to which types of edges or curves in SolidWorks drawings.



Controlling image quality

SolidWorks drawings can contain both shaded and wireframe views. For this reason, the same image quality settings are needed as for parts and assemblies. These settings affect display quality, speed of view refresh, and printing quality.

The setting for Precisely render overlapping geometry is intended for when, for example, you are looking at the end of a cylinder and one circular end is rendered in a lower quality than the other is. With this setting, you would see a nice circle superimposed on a jagged polygon.

Standardizing sheet metal settings for drawing templates

The options on the sheet metal page are meant to help you display the flat pattern drawing for different purposes. For example, you could use a different color for the “bend up” edges than for the “bend down” edges. By default, the flat pattern sketch color is slightly different from the rest of the colors.

This page also enables you to control the position of the bend notes that are automatically placed on the drawing.

Putting views on the template

You can put blank views in a drawing template such that when you create a drawing using that template, they are automatically created. The view types you have to use are any named view, projected view, or a predefined view. Typically, I like to put a predefined view on it, and then use projected views to create the other views, including isometric views. A predefined view is essentially an empty named view.

This is an improvement over the standard three-view functionality, because you have more control over views with which the drawing is automatically populated. You cannot use this method to place detail or section views.

SolidWorks enables you to put drawing views on a drawing in many different ways:

- You can place a single view followed by projected views.
- You can drag and drop a part onto the drawing to get a standard three-view (front, top, and right).
- You can use the Make Drawing from Part/Assembly.

Using predefined views

These methods are all valid, and some are more automated than others are, but none puts views directly on the template. You are not allowed to save a template that references another document, so that limits your choices. You can use the predefined view type for the first view, and project the predefined view to create the rest of the views. You can also use a predefined view to place an isometric view.

Note

You cannot use a predefined view or any other view type on a drawing to create section or detail views. ■

To create a drawing template with empty predefined views, follow these steps:

1. Create a new drawing from a template with the settings you want to use.
2. Create a new predefined view through the menus by choosing **Insert ⇨ Drawing views ⇨ Predefined View**.
3. Assign a default view direction for the new view from the selections at the top of the Drawing view PropertyManager. Front usually works well.

Projecting predefined views

Now that the predefined view is created and located on the drawing, you need to project the predefined view:

1. Select the Projected view icon from the View Layout toolbar.
2. Place the projected views, including a Top and Side.
3. You can also project an isometric view (press and hold Ctrl to break the alignment and then place the view wherever you want it).
4. Press Esc when done.

You could go on and place several predefined views, but if you ever wanted to change the orientation of the part in the drawing, it would be much more difficult: you would have to change the orientation of every view manually. When you want to change the orientation of a drawing created from a template used from those instructions, all you have to do is change the orientation of the original predefined view and all the projected views automatically update to fit. To change the orientation of the view, click in it and select a new view from the top of the PropertyManager again.

Polishing your templates and formats

You could finish your templates right here, save them out, and use them as is, but I recommend you add formats to your templates, and go that extra step with automating drawing creation. When you add formats to your templates, it is not necessary when opening the new drawing. Formats establish the size and border with associated notes for the drawing, so the drawing is not complete until it has a format. For this reason, I save templates with specific formats of a given size and name the templates in a way that includes size, units, and any other important characteristics; for example, if the format was created for a specific company.

In the next section of this chapter, I step you through building a good format. Putting the template together with the format is the best thing you can do to complete the template creation for your users.

Establishing Standardized Sheet Formats

Standardized sheet formats are arguably more important than template files in getting the look and presentation of your drawings to be consistent. Templates are mostly about settings, but formats are about the look. When users refer to “sheet formats” they often just use the term “formats.” These are referring to the same things.

Sheet formats are one of the most frequently asked about drawing topics among new SolidWorks users. It is usually not intuitive to people why something called a sheet format is necessary after you have used a template to create a new drawing. As you have probably picked up by now, the sheet format is essentially the sketch geometry and notes that together make up the drawing border, along with various notes linked to custom properties and some other table management geometry-based anchors.

When working on a new format, you need to perform several tasks which may not be immediately obvious, including how to make a new format, placing table anchors, resizing formats and sheets, and replacing one format with another on a drawing.

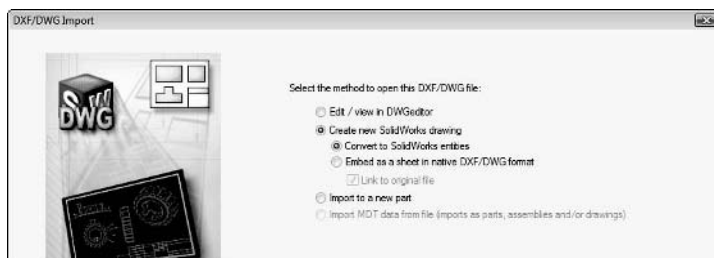
Creating a new format

The most difficult task about creating a new format is getting the sketch geometry correct. It is frustrating to have a high-powered CAD product that cannot handle what most users consider basic 2D functionality. Many users come to the same conclusion when it comes to creating a format from the ground up in SolidWorks — don't do it. SolidWorks doesn't manage this kind of data very well; in fact, it will often suggest that you turn off the sketch relations to speed up the performance of what it looks at as a single large sketch.

For this reason, most people wind up simply importing a drawing border from AutoCAD. The geometry of what you need to create for the drawing border is not complicated, but when you import an AutoCAD file and then need to move or scale the whole thing, it might change things somewhat. Starting with Figure 9.20, I lay out the process of importing a *.DWG to create a format.

FIGURE 9.20

Initiating a DWG import to create a sheet format

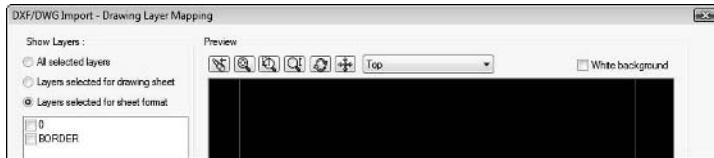


Chapter 9: Maximizing Your Use of Templates

After choosing to import a *.DWG file from the Open dialog box and selecting the desired drawing border file, you will see this interface. If you have to edit the file and you don't have AutoCAD, you can use the DWGeditor. In this case, the file was correct, and it needed to go into a drawing as SolidWorks entities. Figure 9.21 shows the next step.

FIGURE 9.21

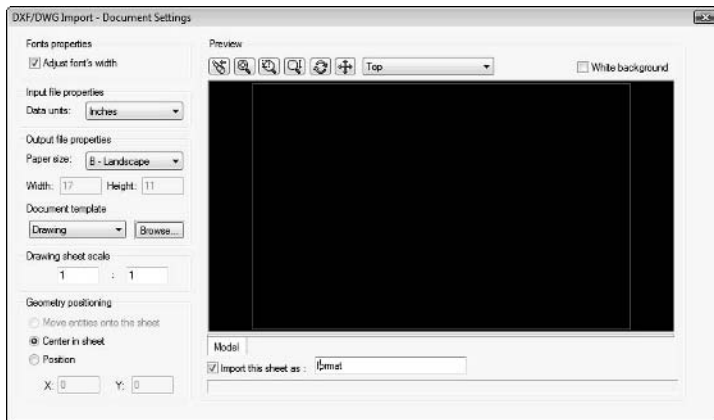
Sending the DWG to the sheet format



In this part of the process, I am not concerned about layers. SolidWorks does use layers, and your sheet format can use layers if you want it to. I actually set up layers later, but you can do it here if it is appropriate. Figure 9.22 is the final step in the process.

FIGURE 9.22

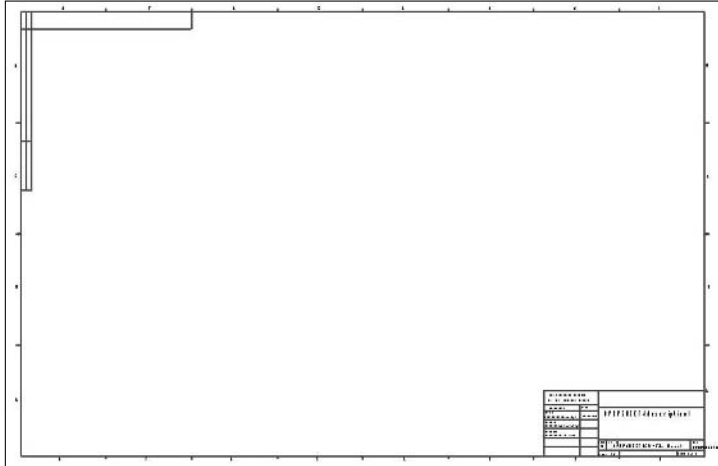
Locating the format on the sheet



The Import wizard tries to make the best decisions based on the input geometry, and in this case, it does a great job. It understands that the border fits onto a B size sheet, and I told it to center it rather than to try to position it. Figure 9.23 shows the final result.

FIGURE 9.23

The result of the import



In this case, the import was perfect, but if you have to move the format, you can use one of several tools:

- The Move Sketch tool
- The Modify Sketch tool
- If you right-click the Draw icon at the head of the FeatureManager in the drawing, you will see a Move option

Of these, the Modify Sketch tool is the most straightforward, but the Move sketch tool is the most accurate. If you need to scale the format to make a B size from an A size, use the Scale sketch tool rather than the Modify Sketch tool. Modify Sketch does not scale the position of the text, but Scale does. With Scale, text size is not scaled, just the position.

Notice that the text does not scale with the sketch geometry. This is a good thing. Standard text size on a B size drawing is not twice the size of text on an A size drawing. Text needs to be large enough to be readable, though not necessarily readable from across the room.

Also, notice that the text does not come into SolidWorks as a font that you would probably want to use on a drawing. It comes in a condensed simplex font. To set the font right, change to a Notes/Balloons selection filter, box select the notes you want to convert, then in the PropertyManager, click the Use Document Font option until it has a check. Once you are sure you don't have to delete and recreate the notes, you can add links to them, just as if they were SolidWorks native annotations.

If you have doubled the size of the format, and need to now double the size of the sheet dimensions, first make sure you are editing the sheet, then right-click in the sheet and select Edit Sheet Properties. This box enables you to select a standard sheet size or input a custom sheet size.

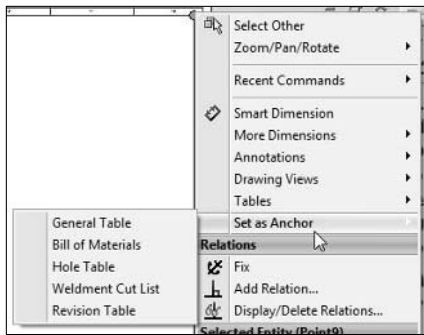
This is also the location where you would reset the next view or datum letter.

Placing table anchors in the format

When you use tables such as Bills of Materials (BOMs), Revision Charts, Hole Tables, and other types of tables, you often have a standard corner for that particular type of table in which to show up. Assigning a table to a corner is as easy as right-clicking and a submenu appears. You must be editing the sheet format when you right-click any sketch endpoint. Figure 9.24 shows the right mouse button menu options for a sketch point in the Sheet Format.

FIGURE 9.24

Right mouse button options for sketch endpoints in the sheet format



Using multiple sheet formats

Just as SolidWorks, drawings can have multiple sheets, SolidWorks templates can also have multiple sheets. Very often companies use a simpler border on sheets after the first sheet. If you set up a second sheet format, it is recommended to add it to the template as a two-sheet template. If you do not add it, you can always add the format to the second sheet when you add the sheet to the drawing. I personally prefer a separate two-sheet template with the simpler format already on the second sheet.

When you want to add formats to drawings, you need SolidWorks to find them, so make sure to use the File Locations settings to tell SolidWorks where your standard and custom formats are located.

Changing the size or format of a drawing

If you need to change the size of a drawing sheet or change the format that you put on it, right-click in the sheet (not in any other entities) and select the new format from the list or click the Browse button. Changing to a larger or smaller format automatically changes the sheet size.

You can also reload a format. You would do this if you have made changes to the standard format; selecting the Reload option updates your drawing.

Standardizing Other Types of Templates in SolidWorks

SolidWorks has other types of templates, aside from parts, assemblies, and drawings. Templates exist for BOMs and other types of tables. Chapter 13 has some examples that use BOMs with edited columns where it makes sense to save the edited BOM as a template. The part, assembly, and drawing templates get all the attention, but you should not forget that you could use these other templates to keep drawing appearance and content standardized.

SolidWorks table types that use templates include:

- Bill of Materials
- Hole tables
- Weldments cut lists
- Revision tables
- General tables

Any time you create a table and customize it, and it looks like one you might have to create again some day, you may be saving yourself a lot of time by saving it as a template. Remember that all these template types can have special storage locations identified in File Locations. Be sure to include these locations in the setup you propagate from machine to machine, and remember to set aside some network drive space for these templates.

Above all, all your hard work to standardize SolidWorks will go for naught if you do not effectively train your users. Training of this sort is not the kind of thing you can buy from a reseller, since it has everything to do with your internal practices. If you have hired a consultant to come in and help you establish a scheme that works, pay a little extra money and have the consultant also train the users. If you do not ingrain new methodologies, things will continue much as they have previously.

Summary

If standardization is a goal for you or your company, optimized document templates are a must. With the settings and options pointed out in this chapter, you should be well on your way to creating templates that are both usable and efficient for use in a number of situations. Remember that not all settings are obvious, or where you expect them to be.

Chapter 9: Maximizing Your Use of Templates

Beyond basic templates, formats are also a necessary element of managing SolidWorks drawing efficiency. When you combine formats with the power of custom properties, drawings, templates, formats, and the associated models that contain the properties, you can turn up the benefits of documentation automation.

By creating templates with all of the necessary settings and standards and making them available to all users via the File Locations path settings, you will be well on your way to establishing the consistency for which a CAD Administrator is responsible.

The final step is to establish templates for all the table types that you will use with SolidWorks. Look through the File Locations folders and make sure that all the template files have a place to go. Using templates of all kinds is certainly a best practice from which everyone benefits.

All CAD administrators, regardless of whether they administer SolidWorks or another CAD package, should seriously consider obtaining all or part of the ASME Y14 series of standards for their company. Even if you do not implement everything that the series lays out, it can provide an excellent and proven starting point for writing your own standard.

Implementing Toolbox

If you as a SolidWorks Administrator are new to SolidWorks Toolbox, you need to read this chapter. In fact, I recommend all potential SolidWorks customers read this chapter before buying the SolidWorks software. Toolbox is an add-in for automating tasks related to inserting and managing commonly used library type parts; and you will get the most out of it if you understand and configure its options early on.

The promise is automation, and the implication is that it works automatically right out of the box. However, the default installation is very far from the configuration in which it actually works best. Both SolidWorks and Toolbox are very complex. This chapter guides you through all of the options and setup parameters available with Toolbox.

Before I discuss how best to implement Toolbox, I cover exactly what Toolbox is, how it works, and why it is as complex as it is. Only then will you be ready to work through various scenarios to see which Toolbox configuration works best for you.

If you only look at Toolbox's individual options, you may never understand everything it does because you are not aware of the big picture. As Administrator, you need to know the capabilities and the limitations of the tool, as well as understand how flexible it is and where its weaknesses lie.

All this goes much deeper than the simple button-push tutorials you will get from most other sources for Toolbox. Understanding the larger picture may make you more tolerant of foibles in the software because you know the payoff can be great. Seeing the overall vision for the software may make you rethink the common wisdom that you should not allow your tools to determine your workflow.

IN THIS CHAPTER

Defining what is Toolbox

Managing Toolbox

Setting up Toolbox

Options to Toolbox

In the past, many people (including myself) have looked at Toolbox as an unmitigated disaster of implementation. Those days appear to be largely behind us, and if you look at Toolbox’s functionality in detail and try to understand why it works the way it does, you will see that if nothing else, the vision driving the creators of the software is very compelling.

Understanding Toolbox

SolidWorks calls Toolbox a library. A library, in CAD terms, is a central place where you can go to get parts that might be shared between multiple products. A more general definition of a library might be that a library is a place you go to get things that exist, as opposed to something like a custom order catalog, where you go to get things that currently don’t exist, but will be made for you.

Another aspect of a library is that it is relatively static. A library has many items from the start. You can add more items to the library, but if you buy a library, you expect it to start out well stocked. Again, Toolbox differs. It doesn’t start out with many items. It starts out with one of every item, but there may be thousands of many items.

It sounds like an academic distinction; can it be important? In the end, you still get your part, right. Well, yes and no. For example, in CAD you share your parts, so that many users can use the same part over and over again. On the other hand, you may send your assembly to someone else, and they are expected to have the same parts in their library that you have in yours.

The main problem is the people who need to know all this information are new customers, and new customers are the last ones to find out that this information exists and realize they need it desperately — *before* they proceed.

Am I dramatizing this? A little, maybe. In the past, it was certainly true that improper Toolbox implementation could result in significant data loss. That is not true so much with versions that are more recent. The main risk now is that you don’t know all of the implementation options, or that functionality such as Smart Fasteners does not live up to its billing.

Because many people are simply not aware of the complete scope of Toolbox. Table 10.1 lists the major bits of functionality that Toolbox offers.

TABLE 10.1

| Toolbox Functions | |
|--|--|
| Toolbox Library | Toolbox Library |
| Administrator lockdown of library settings with a password | Automatic application of custom properties |
| Beam calculator | Bearings |

| Toolbox Library | Toolbox Library |
|---|--|
| Bearing calculator | Brand-name parts, including Helicoil, PEM, SKF, Torrington, Truarc, and Unistrut |
| Bushings | Cams |
| Customized part numbers and descriptions | Export and import Excel tables of part data (create design tables or edit database) |
| Fasteners: screws, bolts, nuts, washers | Grooves (for O-rings and circlips or retaining rings) |
| Hole Wizard. Database of hole sizes to fit standard hardware, including three fit conditions | International standards including ANSI (USA), AS (Australia), BSI (UK), CISC (Canada), DIN (Germany), GB (China), IS (India), ISO (International/Europe), JIS (Japan), KS (Korea), and MIL (US Military) |
| Keys | Limiting selection by standard, type, and size |
| Link hardware and hole sizes | Mass-populate holes with hardware through selection techniques |
| Mold components, including DME, Hasco, PCS, Superior, and Progressive | Option to use configurations or create parts for separate sizes |
| O-rings | Pins |
| Retaining rings, circlips | Sharing Toolbox between multiple users |
| Smart components. Library parts that come with library features; for example, an electrical connector or mounting screws along with cutouts and screw holes | Smart Fasteners. Auto-populates assemblies with fasteners, also uses semi-auto mode |
| A Smart Fastener configuration with fastener types, hole types, and size tolerance | Special recognition as Toolbox parts in Product Data Management (PDM) |
| Specify drive type and thread display for screws | Sprockets, gears, and pulleys |
| Structural members | User-defined standards |

By any measure, this is a lot of functionality in a single SolidWorks add-in, and a lot of functionality to show you how to administer in a single chapter. While this book does not cover how to use the features, it does cover how to administer the functionality, including setup and settings. For more detailed information on how to use the various functions in Toolbox, check out the *SolidWorks 2009 Bible* (Wiley, 2009), or current version.

Understanding how a default Toolbox installation works

Toolbox is not a library. Rather it is a “configurator.” People who work at SolidWorks don’t agree with this assessment, but I’m referring to it by what it does. A *configurator* is an application that custom builds a product for you based on your answers to a set of questions. For example, you might go onto the Web site of an automobile manufacturer and specify that you want the

Part II: Installing SolidWorks

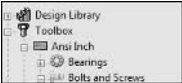
LX model in blue with the upsized engine and MP3 player input. Then this configuration appears on the screen. That's essentially what SolidWorks Toolbox does.

SolidWorks Toolbox has three components: the database, library parts, and program that automate it all. Toolbox takes the input from the user interface and compares it with the database, then uses numbers it retrieves from the database to add a configuration to the part in the library. The default part in the library is usually the biggest or smallest size available from the database. All the numbers for all the sizes are available in the database. In this way, every time you create another size fastener, Toolbox adds a configuration to the part. You can set it up to work differently, but this is how it works by default. This process is shown in Figure 10.1.

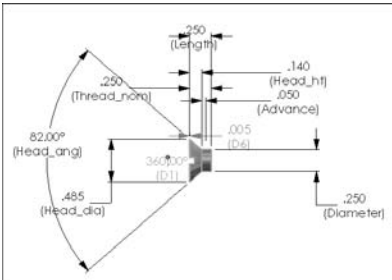
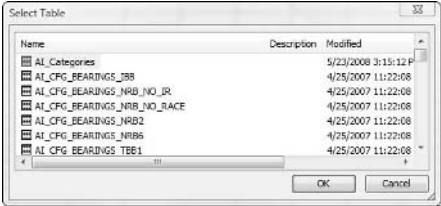
FIGURE 10.1

The basic Toolbox process shows schematically how Toolbox works.

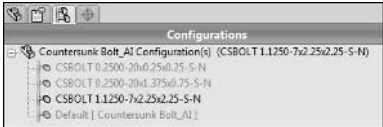
Input size and options
through user interface



Toolbox gets data from database



Database feeds dimensions to SolidWorks



SolidWorks creates a part configuration

I'm sure the reason SolidWorks installs library parts with only a single size is to save space and time during the installation. If you had to create all the sizes of all the fasteners, it would take a great deal of time and a lot of space. Is that a bad thing? This is a question that you will have to answer for yourself. The Toolbox Configuration utility enables you to create all the sizes for a given Toolbox part, whether you are using configurations or individual parts.

The Toolbox default install

When you install Toolbox, and just accept all the defaults, it installs a full database, an empty library, and a set of program files, all on a single machine. Toolbox with all the default settings is

really only meant to be run on a single machine where the user does not share assemblies with other Toolbox users. Ironically, things work okay if assemblies are shared with non-Toolbox users.

I'll cover the database first.

The Toolbox database

The Toolbox database is named SWBrowser.mdb, and for SolidWorks 2009 and 2010 is about 90MB in size; by default, is located at `C:\SolidWorks Data\lang\English\`. As you start using Toolbox and making hardware parts, the size of the database may grow significantly, particularly if you create a "user defined standard," which I discuss later in this chapter.

The end user has no need to do anything in the Toolbox database, and even an Administrator shouldn't take on database customization unless she is well acquainted with Microsoft databases.

The size of the database (about 90MB in default arrangement) can be of significant concern for some situations, such as when sharing a database between users across a network, or even just opening it on machines with low resources.

As with every other kind of data, it is important to back up your Toolbox database if you make any changes to it. Fortunately, if you do make a mistake, it is always easy to get a copy of a default database back by simply performing a default install of SolidWorks. You should keep a full version of the default database available in an archive, and if you customize Toolbox, you should keep a copy of a modified version. Backing up Toolbox data is simple: it requires only a single top-level folder to be copied, the `\SolidWorks Data\` folder. The backup is as simple as copying the folder to another location, and the restore is as simple as overwriting the folder and contents. (It is recommended to rename the old folder before restoring a copy of the folder.)

One thing to remember that can complicate the backup Toolbox database issue is that each version of SolidWorks uses an identifiably different version of the database, and the database version might be tied to versions of the library. If your company uses multiple versions of SolidWorks, maintaining versions of Toolbox can be a significant challenge.

The part library

The Toolbox part library is located in the Browser folder under the SolidWorks Data folder. When the library is initially installed, each part has a single default configuration sized at either the high or the low end of the size range (biggest or smallest available size). These parts have features, some of which are suppressed depending on settings. All parts use Mate References. Mate References are stored faces or edges that are used as default mating entities when inserting Toolbox parts. For more information on how to create or use Mate References in SolidWorks, refer to the *SolidWorks 2009 Bible* (Wiley, 2009).

Program files

The program files for Toolbox are kept in the `\Program Files\SolidWorks\Toolbox` folder, as a set of *.dll files. You will not have any need to access or edit these. These files are updated with new major releases and service packs.

Managing Toolbox data

Managing SolidWorks data is a big issue that you have to understand as a SolidWorks Administrator. When you add Toolbox to the mix, you might expect that SolidWorks Corp has already thought of something clever for managing Toolbox data, but, in that respect, you might be disappointed. In fact, the major conceptual flaw in Toolbox is in how it handles data management issues.

Underlying basic flaw in Toolbox

If there is a basic underlying rule for SolidWorks data management, it is that you need to ensure unique filenames for all documents, and its corollary that you should not have multiple files with the same name accessible by the software, especially if those files with the same names have different content. The designers of the Toolbox software have allowed situations when the rules are very easy to be violated. In truth, it doesn't appear that they considered the file management issues inherent in a configurator application until it was too late. Even now, with all the small fixes they have implemented for all sorts of various small issues, and not so small issues, the file management problem is not one they have addressed with conviction. A real answer to the file management problem is a static library. SolidWorks is a very progressive company, and may see this as unnecessarily timid. Given the problems with the dynamic library, I believe the static library to be prudent.

Beyond the gross file management issue, if you want to include custom property information with your Toolbox parts, and make sure that data is propagated to copies, you may run into difficulties making sure that happens. The model of a static library of parts that you can count on is very difficult to achieve and maintain in Toolbox.

Carrying it all a step further, say you want to add material classifications or even something like point type (such as cup point, nylon point, dog, half-dog, and so on) to screws, or flags for self-tapping threads or nylock thread locker. You can add these types of things, but doing so, and maintaining that information between releases of SolidWorks, can be time-consuming and difficult to figure out initially and troubleshoot.

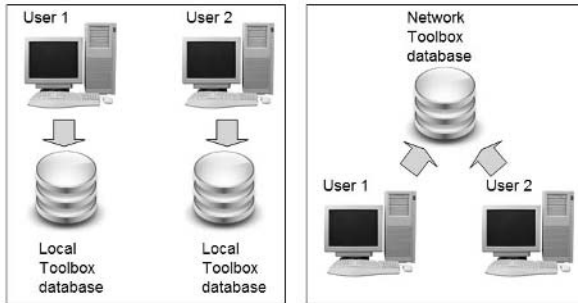
You can work around the file management problems, and the custom property problems and the material/finish/point/thread treatment classifications, but each one of those issues requires a separate solution, and each has limitations, some of which you may need to discover and solve for yourself. Toolbox is certainly capable, and full of functionality, but the way that it is all put together, you have to wonder if the effort to get the solution is worth it.

Configuring a multi-user installation

Many SolidWorks installations consist of multiple users. When multiple users have to work together, they need to avoid the default single-user Toolbox installation. Figure 10.2 shows two situations: one with multiple users using the default single-user Toolbox and another with multiple users sharing a Toolbox installation.

FIGURE 10.2

Multiple users with individual and shared Toolbox installations



Another issue that comes up with this scenario is the speed problem of opening the database across the network, beyond the overhead of the assembly and parts. To counter this, you can put the database local to each user, but you have to make sure that the two users are using the same Toolbox settings before copying the database.

Therefore, that solves some of the problems, but problems with write access on the library parts remain. If you make the libraries local, you're back to the same situation that you started with. The available sizes in one library do not match the available sizes in the other library, so sharing assemblies that might call for missing configurations between the users is not possible.

The obvious solution here would be to fully populate one library, and copy it to both users. This is a great solution, but it can be time-consuming. However, it gives you the option to add custom properties before copying.

Combining libraries

New users often simply install the software trusting it will work without doing much research beforehand. If you do this when Toolbox is involved, and you have multiple users, you will wind up with each user installed as an individual user without shared Toolboxes. If the users use Toolbox when it is installed this way, you will wind up with many individual libraries with different size configurations created in different parts.

Many times users ask if you can combine the configurations from one library with the configurations from another library. The stock answer to this question is no. SolidWorks does not offer a direct way to combine configurations between two different parts.

A method does exist for making this work, but it isn't really intended for this purpose. If you have two parts with configurations and you want to combine the configurations into a single part, the only way I know of to do this is to use the Auto-create Design Table functionality to convert the existing configuration data into a design table, then export the design table and import it into the other part. This will re-create the configurations from the first part into the second part. It is not a very direct method, and if you have configurations from a large number of parts that you need to combine, it can be tedious, but the method works, and as the only method that works, it has advantages.

Administering Toolbox

Toolbox as an add-in requires almost as much of an administrator's attention as the rest of the SolidWorks software combined to properly understand and implement. As an Administrator, Toolbox has plenty of things to keep you busy. The first thing to tackle is how to install Toolbox. I explain how to do it here, as well as how to make changes to existing installations and upgrade Toolbox.

Installing Toolbox

Toolbox is typically installed along with the rest of the SolidWorks software during a standard installation. During a SolidWorks 2010 Installation Manager installation, one of the options in the Summary section of the installation workflow is Toolbox Options. If you do not change the default settings, Toolbox defaults to C:\Solidworks Data\.

Cross-Ref

The standard SolidWorks installation procedure is detailed in Chapter 6. ■

You can install Toolbox two ways:

- **Local Toolbox.** Located where the SolidWorks Data folder is on your local machine.
- **Shared Toolbox.** Located where the SolidWorks Data folder is on the network.

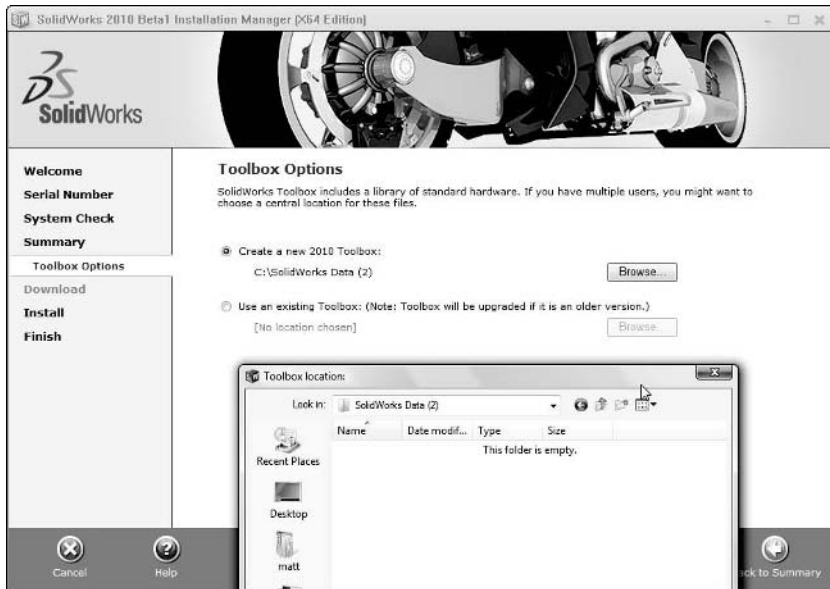
Each method has strengths and weaknesses, and neither is a silver bullet. If you want to create a local Toolbox, follow the default installation workflow.

Installing a shared Toolbox

If you want to install a shared Toolbox between several users, direct the initial installation Toolbox Options path (see Chapter 6) to your network drive. For subsequent installations, use the Use an existing Toolbox option, as shown in Figure 10.3.

FIGURE 10.3

Toolbox installation options enable you to install Toolbox data in a convenient location.



After you have installed the folder to the network location, set the SolidWorks Data folder to Read-only. You may need special user privileges on the server to accomplish this task.

1. Right-click the SolidWorks Data folder (displayed as SolidWorks Data 2010), as shown in Figure 10.4, to open the SolidWorks Data 2010 Properties dialog box.
2. In the SolidWorks Data 2010 Properties dialog box, select the Read-only option.
3. Click Apply and then click OK.

The Confirm Attribute Changes dialog box appears.

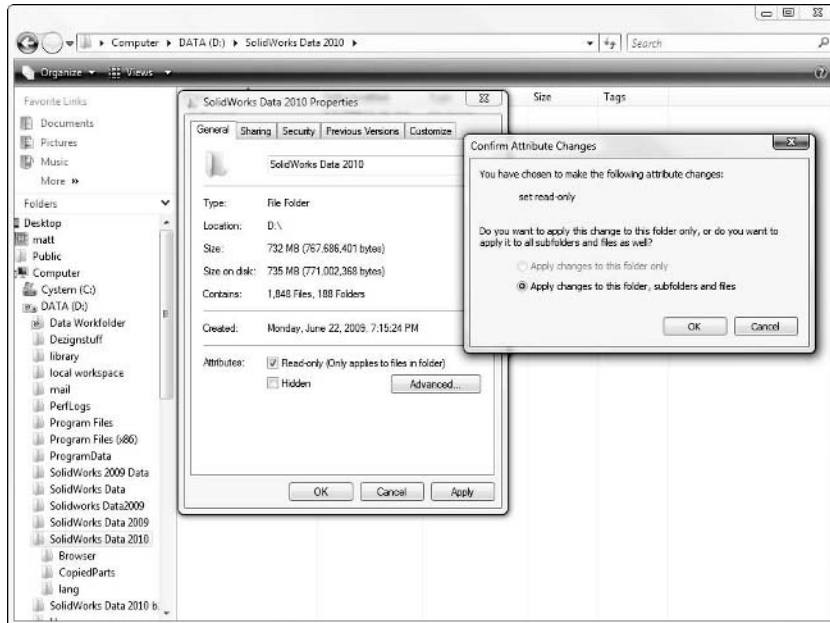
4. Select the Apply changes to this folder, subfolder, and files option, and click OK.

The reason for setting the files and folders to Read-only is to prevent conflicts of file ownership if PDM software is not utilized. Toolbox will manage the write access it needs when the software creates new configurations and wants to save them back to the file.

Part II: Installing SolidWorks

FIGURE 10.4

Setting SolidWorks Data to Read-only



Creating a shared Toolbox from an existing installation

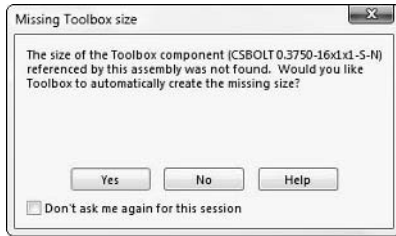
As soon as you install Toolbox and begin using it, making changes to the installation becomes more and more complex. Using Toolbox changes the library, so there is an important difference between a brand-new library and one that has been used even once. If you have an existing Toolbox that was installed locally, and you want to create a shared Toolbox installation from it, you have more work to do.

You may want to create a composite Toolbox from several users who have already been using the software (because few people learn about the settings that they need to know before they do their first installation). You may have users who have created different parts, and, therefore, have different size configurations created in their default library parts. For this reason, if you try to open one user's assembly with the Toolbox library from a different user, you will find that some configurations are missing. Figure 10.5 shows the warning message you get if the assembly calls for a configuration that does not exist in your Toolbox library.

The functionality of Toolbox can only re-create a missing Toolbox size when the assembly was created in version 2007 or later. This does free you from worrying about potential data loss from sharing data with other Toolbox users, but if you choose to use the Copy Parts option (saves copies of parts instead of using configurations), the automatic configuration creation will not benefit you, because it will never be needed. Configurations are a fantastic tool, but missing configurations in library parts does not improve the situation.

FIGURE 10.5

Warning message for a missing Toolbox configuration



Combining configurations from multiple parts

One of the difficulties in combining libraries is that there is no good way to do it. SolidWorks does not have an automatic way to combine the configurations of sets of parts into a single part containing all the combined configurations of both parts. The quickest way to combine configurations is:

1. Use the configured Toolbox part to auto-create a design table.
2. Auto-create design tables from each of the parts with configurations you want to combine.
3. Use the design tables to add configurations into one of the parts.

This method works, but it is not automatic, and if you have many different types of parts to combine or many users from which to combine data, it becomes tedious quickly. For more in-depth information on using design tables and auto-creation, check out the *SolidWorks 2009 Bible* (Wiley, 2009) or the current version.

Setting up the moved Toolbox

Follow these steps to make a shared Toolbox from an existing Toolbox:

1. Select the Toolbox library of the user who has created the most parts using Toolbox, or use the steps described earlier to create new library parts using design tables.
2. Move or copy the \SolidWorks Data\ folder, including the newly combined files in the \SolidWorks Data\Browser\ folder, to the new network drive location.
3. Set the top-level \SolidWorks Data\ folder to Read-only, as described earlier in this chapter.
4. For each SolidWorks installation intended to use the shared Toolbox, choose Tools ⇄ Options ⇄ Hole Wizard/Toolbox and set the folder to the network location of the \SolidWorks Data\ folder.

Part II: Installing SolidWorks

Note

It is preferable to use a UNC (uniform naming convention) path rather than a mapped drive letter because the UNC is more explicit, more consistent, and less vulnerable to change. An example would be `\\servername\AppData\SolidWorks Data\` rather than `P:\AppData\SolidWorks Data\`. Mapped drives can be mapped differently on different computers, but the UNC path to a specific network location is always specific as long as the server it refers to remains on the network. ■

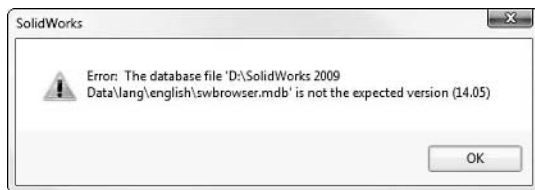
Upgrading Toolbox

If you have a stand-alone installation of SolidWorks and an individual Toolbox, upgrading Toolbox happens in the course of a regular SolidWorks upgrade. This is true whether you are upgrading between SolidWorks versions or just updating a service pack. SolidWorks may provide new executables (Toolbox programming), a new database, and new library parts. The executables should never be a problem, because neither you nor your users should need to edit the executables. However, the database and the library files are different; you are expected to edit the parts at the very least. If you make changes to the database or library and then upgrade, SolidWorks may overwrite your changes in a way that is not recoverable.

Of course, you can use manual methods to deal with this problem, which include backing up your files or performing the upgrade and then overwriting the upgraded files, but even conceptually, this only works for service pack updates. The database could potentially be highly restructured between SolidWorks versions, and if SolidWorks recognizes that the database is an older version, it kicks out an error message like the one shown in Figure 10.6.

FIGURE 10.6

An error message when the Toolbox database is older than the current SolidWorks version



Straight upgrades, where you simply upgrade one version of SolidWorks to another, are not recommended. If you want to upgrade from SolidWorks 2009 to 2010, for example, install 2010 as a separate version, leaving 2009 in place until you don't need it anymore, at which point you can uninstall it. Keeping parallel installs of different major versions does not pose a risk to your computer or SolidWorks installation. In fact, you can even run two different major versions, such as SolidWorks 2009 and 2010, at the same time.

Note

The one thing that is not allowed is to have parallel installations of the same SolidWorks version at different service packs. For example, SolidWorks 2010 SP 0 and SP 1 could not exist on the same computer at the same time. ■

Dealing with version incompatibility in Toolbox

Here is another conundrum you can run into with Toolbox. What happens when one person in your group upgrades SolidWorks versions? If this person upgrades the shared Toolbox along with SolidWorks, other Toolbox users will receive a version compatibility error, at least from the new parts in the library, if not also the database. If the person upgrading does not upgrade his Toolbox, he will receive his own version incompatibility error for the database, but the parts will not give him an error. If he adds a configuration to a part in the library, the part is saved in the new version, and then when users of the older version use that part, they will get the future version error.

Upgrading and keeping both versions

Here's how to deal with the version incompatibility issue in Toolbox. First, correct people who call this "backwards compatibility," because that simply is not the case. It is, in fact, version incompatibility.

Say you intend to upgrade from SolidWorks 2009 to 2010. You plan to have parallel installs, but you realize that if you simply install 2010 with Toolbox, your 2010 Toolbox will be a default Toolbox, and you will be missing all the configurations and customization you added to Toolbox over the past year.

You also realize that you cannot just use the 2009 Toolbox because if SolidWorks 2010 updates any of the parts, SolidWorks 2009 will not be able to read them anymore.

To have a functional 2009 and 2010 Toolbox, both functioning on the same computer without losing any of the work you did to the 2009 Toolbox, follow these steps:

1. **Copy the \SolidWorks Data\ folder to a new location.** You may notice that in some of the screen shots of my setup, I identify the version of the SolidWorks Data folder, such as SolidWorks 2010 Data or SolidWorks Data 2010. Copy and rename the folder with the included data.
2. **When installing SolidWorks 2010, point the Toolbox Options (in the Summary of the installation workflow) to the existing folder \SolidWorks Data 2010\.**

This makes use of your old library with all its included parts, as well as the old database with any setup you have done. The install updates the database, but the parts are not updated until they are used, changed, and saved.

The one drawback to this process is that if you make any changes to the older Toolbox, they do not propagate to the newer Toolbox.

Configuring Toolbox

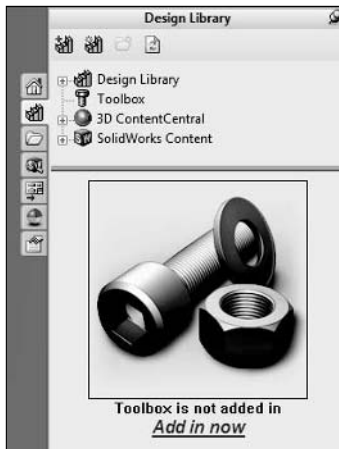
To get the most benefit from Toolbox, you must configure it. Default settings are intended for a single user who never shares files, and these settings do not work well in other situations. To begin to understand all the options you can control to make Toolbox as usable as possible for your organization, you must examine the Toolbox Configuration Wizard. As I mentioned earlier, Toolbox has a dazzling array of options when it comes to settings. This section of this chapter is meant to help you make sense of everything, and understand what is critically important, what needs your attention, and which things are simply optional.

Starting Toolbox

To get started configuring Toolbox, first make sure Toolbox is turned on, and understand how to access it. Toolbox is located either in the pull-down menus or in the Library tab of the Task Pane to the right side of the SolidWorks window. Figure 10.7 shows what you might see if you try to access Toolbox without first turning on the add-in.

FIGURE 10.7

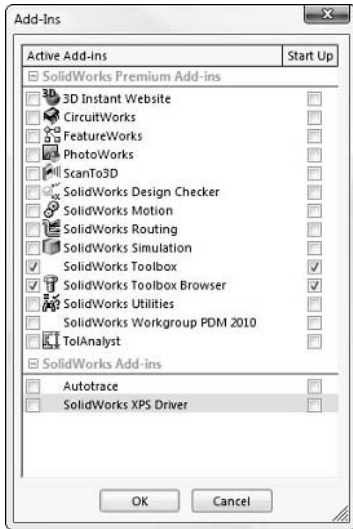
Toolbox is an add-in you must access through the Tools ➤ Add-ins menu.



To activate the Toolbox add-in, you can click the Add in now link or, to have more control over how Toolbox starts up, choose Tools ➤ Add-ins. In the Add-Ins dialog box, notice you have a selection of add-ins and two columns of boxes to select. The column on the right, shown in Figure 10.8, indicates which add-ins will start up whenever SolidWorks starts up. The column on the left enables you to activate add-ins for the current session of SolidWorks only.

FIGURE 10.8

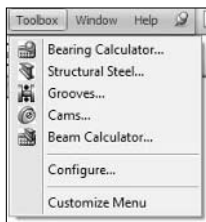
Using the Add-ins menu to activate SolidWorks add-ins



To have access to all the functionality of Toolbox, you need to activate both SolidWorks Toolbox and SolidWorks Toolbox Browser. The difference between these two is that the SolidWorks Toolbox adds the tools accessed through the Toolbox drop-down menu, shown in Figure 10.9. The SolidWorks Toolbox Browser adds the more popular fastener creation tools found in the Task Pane.

FIGURE 10.9

The Toolbox drop-down menu offers access to calculators and other library-feature type tools.



Setting up Toolbox

A wizard helps you set up Toolbox. SolidWorks has not named this utility consistently: you will see it called Configure Toolbox or Toolbox Setup. You access the functionality by choosing **Toolbox** ⇨ **Configure**; first you see **Welcome to Toolbox Setup**, and then Follow the steps below to configure Toolbox.

Part II: Installing SolidWorks

Regardless of what you actually call this wizard, you really need to be aware that the word “configure” in this setting is just an unfortunate use of terminology. It does not have anything to do with SolidWorks configurations; it enables you to use settings to control Toolbox functionality. Some of the functionality that Toolbox Setup controls happens to be related to SolidWorks configurations, so be aware that the word configure is used multiple ways (which can lead to confusion). Figure 10.10 shows the initial interface for the Toolbox Setup Wizard.

FIGURE 10.10

The first window in the Toolbox Setup Wizard



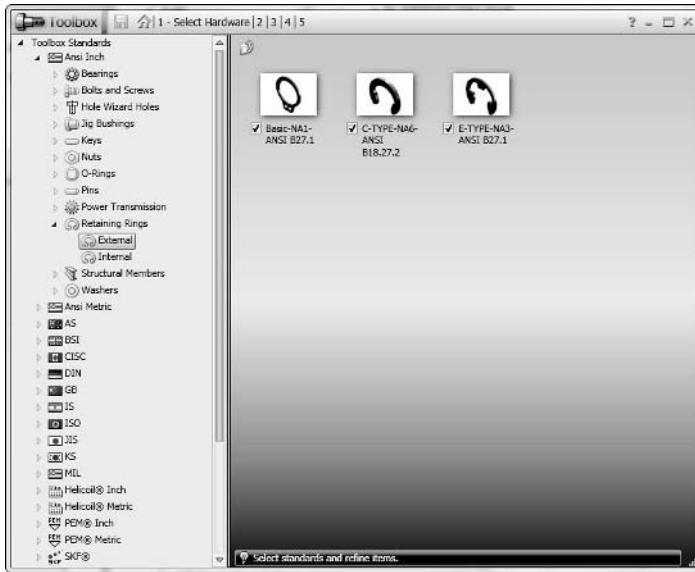
Selecting your hardware — Step 1

The SolidWorks Toolbox has a huge assortment of parts, including many international hardware standards. You might consider this tool collection too much, but someone else might wonder why her favorite part is not included. This is why one of the first things you do when setting up your Toolbox is to limit your Toolbox parts to the parts that you are going to need. This has several benefits, including that users will experience less visual confusion, accessing less data from the database will improve performance, and that users will be less tempted to get creative with hardware selection, limiting themselves to hardware that is more easily attainable.

When you start setting up Toolbox, one thing you need to ask yourself is “Am I really going to use this part?” You have the option to turn off entire standards, hardware classifications, or specific types of hardware, as shown in Figure 10.11.

FIGURE 10.11

Using Toolbox Setup to limit hardware selections



The check boxes next to each item enable you to select and deselect that item. For example, you can turn off the entire ANSI Inch standard, just the Retaining Rings category, or any of the parts: Basic, C-type, or E-type. In another part of the interface, you can turn off specific sizes to limit further the size of the library.

Customizing hardware — Step 2

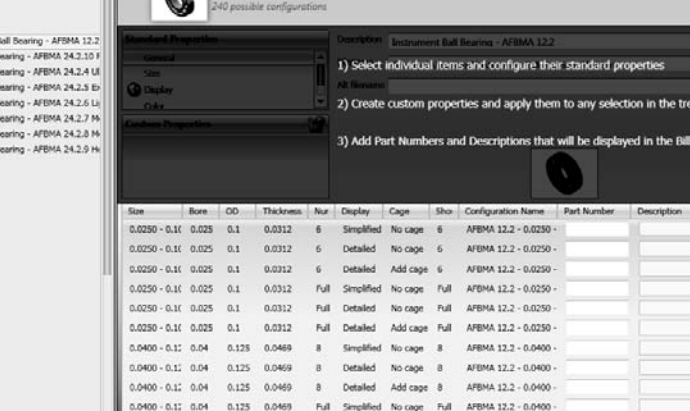
The next step in setting up your Toolbox is to customize the hardware. Many options are available in the Toolbox Setup interface, as shown in Figure 10.12. Much of the work that you do here will be done in Excel, if you choose to go that route.

Figure 10.12 shows the top fields covered by a message prompting you to make changes to the data. If you click in the grayed out area, the message disappears, and you will have access to enable or disable specific sizes of the selected hardware, and add custom properties.

Custom Properties



If you click the Custom Property Definition icon in the Custom Properties area, the interface shown in Figure 10.13 appears and enables you to establish custom properties with a set of preset options that will display in the Toolbox PropertyManager when you place the part.

FIGURE 10.12

100IDBOX 312 - Customize Hardware | 3 | 4 | 5

Standards
1 Inch
Bearings
Ball Bearings
Instrument Ball Bearing - AFBMA 12.2
Thrust Ball Bearing - AFBMA 24.2.10 F
Thrust Ball Bearing - AFBMA 24.2.14 U
Thrust Ball Bearing - AFBMA 24.2.3 B
Thrust Ball bearing - AFBMA 24.2.6 U
Thrust Ball bearing - AFBMA 24.2.7 M
Thrust Ball Bearing - AFBMA 24.2.8 H
Thrust Ball Bearing - AFBMA 24.2.9 H
Roller Bearings
Bolts and Screws
Hole Wizard Holes
3g Bushings
Keys
Nuts
O-rings
Pins
Power Transmission
Retaining Rings
Structural Members
Washers
Metric

Instrument Ball Bearing - AFBMA 12.2
240 possible configurations

Standard Properties
Description: Instrument Ball Bearings - AFBMA 12.2

1) Select individual items and configure their standard properties
2) Create custom properties and apply them to any selection in the tree
3) Add Part Numbers and Descriptions that will be displayed in the Bill of Materials

| Size | Bore | OD | Thickness | Nur | Display | Cage | Sho | Configuration Name | Part Number | Description | Comment |
|---------------|-------|-------|-----------|------|------------|----------|------|-----------------------|-------------|-------------|---------|
| 0.0250 - 0.11 | 0.025 | 0.1 | 0.0312 | 6 | Simplified | No cage | 6 | AFBMA 12.2 - 0.0250 - | | | |
| 0.0250 - 0.11 | 0.025 | 0.1 | 0.0312 | 6 | Detailed | No cage | 6 | AFBMA 12.2 - 0.0250 - | | | |
| 0.0250 - 0.11 | 0.025 | 0.1 | 0.0312 | 6 | Detailed | Add cage | 6 | AFBMA 12.2 - 0.0250 - | | | |
| 0.0250 - 0.11 | 0.025 | 0.1 | 0.0312 | Full | Simplified | No cage | Full | AFBMA 12.2 - 0.0250 - | | | |
| 0.0250 - 0.11 | 0.025 | 0.1 | 0.0312 | Full | Detailed | No cage | Full | AFBMA 12.2 - 0.0250 - | | | |
| 0.0250 - 0.11 | 0.025 | 0.1 | 0.0312 | Full | Detailed | Add cage | Full | AFBMA 12.2 - 0.0250 - | | | |
| 0.0400 - 0.11 | 0.04 | 0.125 | 0.0469 | 8 | Simplified | No cage | 8 | AFBMA 12.2 - 0.0400 - | | | |
| 0.0400 - 0.11 | 0.04 | 0.125 | 0.0469 | 8 | Detailed | No cage | 8 | AFBMA 12.2 - 0.0400 - | | | |
| 0.0400 - 0.11 | 0.04 | 0.125 | 0.0469 | 8 | Detailed | Add cage | 8 | AFBMA 12.2 - 0.0400 - | | | |
| 0.0400 - 0.11 | 0.04 | 0.125 | 0.0469 | Full | Simplified | No cage | Full | AFBMA 12.2 - 0.0400 - | | | |
| 0.0400 - 0.11 | 0.04 | 0.125 | 0.0469 | Full | Detailed | No cage | Full | AFBMA 12.2 - 0.0400 - | | | |

FIGURE 10.13

Custom Property Definition

Property Name:

Type: ☐ Textbox ☒ List

☐ Link to SW material

Options:

- ☒ Add as configuration specific property
- ☒ Each value for this property requires a new configuration name
- ☒ Show in PropertyManager

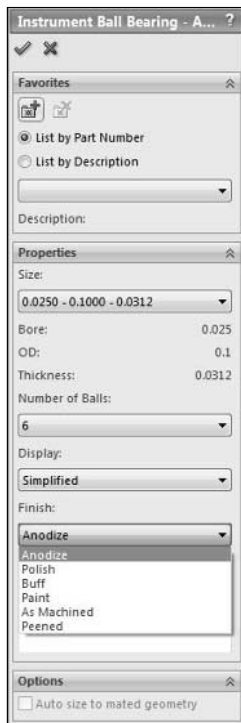
| Value | Suffix | |
|-------------|--------|---|
| Anodize | -a | X |
| Polish | -d | X |
| Buff | -b | X |
| Paint | -p | X |
| As Machined | -m | X |
| Peened | -e | X |
| | | |

OK Cancel

224

FIGURE 10.14

The Toolbox PropertyManager for inserting a part includes a drop-down list of custom property values.



Part number and description

In the Toolbox Setup interface, the Part Number, Description, and Comment fields are blank (refer to Figure 10.12). You can fill in these sections, so when you actually create the parts, those values propagate to your Bill of Materials (BOM). The Part Number is the part number associated with the configuration properties for the BOM; it is not a custom property. The Description is a configuration specific custom property, even when you save sizes out as separate parts.

Notice that next to the bearing image at the top of the screen, a note says the part has 240 possible configurations. Does this mean you are going to enter manually 240 different part numbers and descriptions? No way. This interface has no intelligent capabilities except copy and paste. If you enter both size and type data for the 12 different kinds of hardware, you will need to enter tens of thousands of lines of data, which is obviously impractical just to implement some of the metadata in your hardware library.



To get around the manual data entry, you can export the complete table of data to Excel and build the part numbers and descriptions there. You can export the data using the button shown on the left, which is at the right end of the title row in Figure 10.12 (about halfway down on the right side of the image).

Part II: Installing SolidWorks

If you have a semi-intelligent part numbering system, you can build the part number based on size data exported in the table. Figure 10.15 shows an example of this type of Excel function, the result of using the Concatenate command.

FIGURE 10.15
Using the Excel Concatenate command



| | A | B | C |
|---|-------------------------------|--------------|--------------------------|
| 2 | | \$PARTNUMBER | \$PRP@DESCRIPTION |
| 3 | 0-80 X 125 Button Head-Soc-SS | 54383-0-AAHA | SCR,BHS,0-80X1/8,SOC,SS |
| 4 | 0-80 X 187 Button Head-Soc-SS | 54383-0-ABHA | SCR,BHS,0-80X3/16,SOC,SS |
| 5 | 0-80 X 25 Button Head-Soc-SS | 54383-0-ACHA | SCR,BHS,0-80X1/4,SOC,SS |
| 6 | 0-80 X 312 Button Head-Soc-SS | 54383-0-ADHA | SCR,BHS,0-80X5/16,SOC,SS |
| 7 | 0-80 X 375 Button Head-Soc-SS | 54383-0-AEHA | SCR,BHS,0-80X3/8,SOC,SS |
| 8 | 0-80 X 437 Button Head-Soc-SS | 54383-0-AFHA | SCR,BHS,0-80X7/16,SOC,SS |
| 9 | 0-80 X 5 Button Head-Soc-SS | 54383-0-AGHA | SCR,BHS,0-80X1/2,SOC,SS |

In this example, the base part number 54383-0-AAHA is assembled together with data that Excel pulls from cells E3, M3, and Q3, along with the letter H. You can find more sophisticated examples that use the Vlookup command in Excel.

By populating the part number and description fields this way, you can save a lot of data entry time and avoid mistakes. Once the data is filled in, you can import the Excel data back into the Toolbox Setup Wizard, and it populates the database for you. If you are wondering if this is a bit of a workaround, it is. However, if you want the benefits of having this data automatically entered into your BOMs, without requiring users to enter data manually, this technique, convoluted as it is, will save you a ton of time and mistakes.

Creating parts or configurations

At this point in the Toolbox Setup Wizard, you have not yet been presented with the option to use parts or configurations. It is an absolutely pivotal decision and should probably be the first thing you do in the wizard, but it doesn't show up until Step 3. To preview the difference between using parts or configurations, you have the option for each individual size to be either a configuration within a larger part or a separate part. So your socket head cap screws could all be contained with several hundred configurations within a single part file, or you could have several hundred individual part files where each part contains a single-size socket head cap screw. More on that decision later, but for now, assume you have already made that decision, and for what you do next, it doesn't matter which option you decided upon.



In addition to the export and import Excel files functionality, the icon at the left also enables you to create all the parts or configurations for a given Toolbox part. The Fillister Head Screw has 7,044 possible configurations. Use this tool with care. Do you really need a single part with more than 7,000 configurations? Can you imagine how 7,000 part files in a library just for the fillister screw would affect your users?

The point is that this functionality is available if you want to try it or feel you need it, but it comes at a price. Another option that may be less insane would be to export the table data, reformat it so that it looks like a design table, trim out unnecessary data, then bring it into the library part and create the configurations that you need.

Note

If you look at the data in the Toolbox Setup, Customize Hardware step more closely, you see that the Fillister screw contains duplicate data for cross and slotted heads, and then each of these is tripled by specifying simplified, cosmetic, and schematic thread representations. This means that in essence you have six times as many configurations as is warranted by just the geometry itself. The rest of the configurations are due to metadata. You have to make up your mind about how to deal with this. Does your BOM need information about meta-data, such as drive type, screw tip type, screw material, or finish? If this level of data is handled elsewhere such as in PDM, ERP/MRP (Enterprise Resource Planning/Material Requirements Planning), you may be able to get away with representing the geometry in the SolidWorks model. However, if the information in your SolidWorks BOMs is used to feed your PDM or ERP system, additional metadata will be required. ■

Regardless of how you divide it up, adding this type of data to your Toolbox is going to be a lot of work. The good news is that you'll only need to do it once, and it is done for all your users and for all future releases of the SolidWorks software.

User Settings — Step 3

The main decision you have in Step 3 is Parts or Configurations. Many people have a superstitious fear of configurations in SolidWorks. Others are turned away by the overhead of having a single part file that contains possibly hundreds, or even thousands, of configurations get very large.

Parts or Configurations

One of the huge advantages of configurations over individual parts is that if you have a change that you want to make across all sizes, it is easy to do if you use configurations, but a veritable nightmare if you use individual parts. Granted, you won't be making many changes to library parts, but if you need to derive a new library part from an existing one, it is the same deal, though it is much easier with configurations than with individual parts.

If I could, I would dispel users' fear of configurations. If you plan how you are going to use the configurations, and maybe control them through a visual interface like a design table, they work reliably. For many things, they are simpler to use. Switching configurations of a part is easier than swapping parts. Figure 10.16 shows the User Settings dialog box.

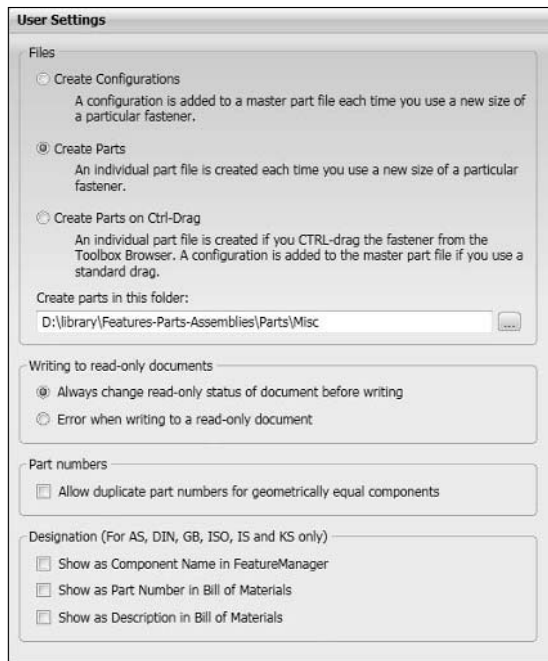
One thing you need to be aware of with this choice between parts and configurations is that if you start doing it one way and then switch, you may cause yourself problems. For example, if you start using Configurations, and switch to Parts, the parts that it copies each have configurations built into them.

With the file management side of things, I recommend not making a definite choice on this parts or configurations dilemma until you have read the section on file management heading in this chapter. Configurations are very attractive, but when you are dealing with managing files and the changing nature of Toolbox parts, you have more to consider than you might originally think.

Part II: Installing SolidWorks

FIGURE 10.16

The User Settings dialog box enables you to select between parts and configurations.



Read-only settings

When you create a multi-user environment for Toolbox, Toolbox must handle part permission to ensure edits are available when they need to be. The settings shown in Figure 10.16 are the defaults, and they should remain that way. I can't think of a reason to change the read-only settings in the User Settings dialog box.

Permissions — Step 4

The Toolbox Setup Wizard is attached to a specific database. To check on which database it is connected, choose Tools ⇨ Options ⇨ Hole Wizard/Toolbox. The path displayed should point to the database that this specific SolidWorks installation is attached. If this is a shared database and you create a password in the Permissions dialog box, you have taken control of the Toolbox installation for all the users who are also linked to that database. In fact, the first user to create a password can take control. As SolidWorks Administrator, I recommend you are the first one to apply a

password. The permissions only affect the settings in the Permissions dialog box, not for the rest of the settings in the Toolbox Setup Wizard. Figure 10.17 shows the Permissions dialog box.

FIGURE 10.17

The Permissions dialog box allows the administrator to control the functions available to the user.



If for some reason you are locked out of the database, replacing the database with a fresh copy should get you back in, but you will lose changes.

Smart Fasteners — Step 5

Smart Fasteners is a tremendous idea, but as it turns out, it is probably one of the weaker areas of functionality in Toolbox. Smart Fasteners has two modes of operation. The first is to simply populate all the holes in an entire assembly automatically and all at the same time. The problem with this is that little irregularities throw off the automation, so you wind up spending more time intervening manually than it saves you. The types of errors it creates will be bolts that go into holes head first, on the ends of shafts, in appropriate sizes, and so on.

The more appropriate way to use Smart Fasteners is to select a face or a set of holes and tell it to populate that smaller selection. A more guided application of this sort is a benefit when compared to manually inserting fasteners, especially the nut and washer stacks. The settings here in the Toolbox Setup Wizard Smart Fasteners page are shown in Figure 10.18.

FIGURE 10.18

Smart Fasteners settings in the Toolbox Setup Wizard



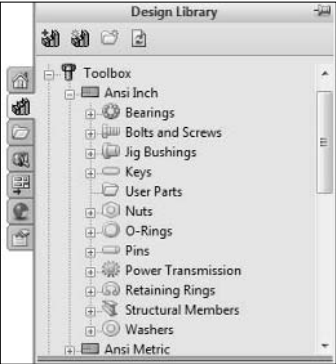
Adding user parts to the Toolbox library

You may run into situations where you want to add your own parts to the Toolbox library to be used similarly to how you use Toolbox parts. Adding parts is easy, but some limitations exist as to what you can do with user-added parts. You cannot use user parts for functionality such as Smart Fasteners.

Figure 10.19 shows the Design Library tab in the Task Pane, where the main Toolbox selection interface resides. Using this interface, you can add your own folders to the existing list, and then save the current part document to that folder.

FIGURE 10.19

The Design Library interface



Note

When adding parts to Toolbox that need to mate to other parts, it is a good idea to use Mate References. The details of applying and using Mate References can be found in the SolidWorks 2009 Bible (Wiley, 2009), which is a user guide. ■

File management and Toolbox

Have you ever noticed that SolidWorks example files never use Toolbox parts directly? There are reasons for that, including that not all customers have Toolbox. In addition, Toolbox is not really set up to be used “that way.” By “that way,” you can infer that file management in Toolbox is not as simple and straightforward as your other SolidWorks part data. This is where Toolbox, as a configurator rather than a library, starts to have a negative impact. Because the files do not exist as a static collection of files, you have to handle file management for Toolbox differently.

In the final four chapters of this book, I discuss SolidWorks Workgroup PDM in detail, and include some information on using Toolbox with Workgroup PDM. In this chapter, I deal with manual file management techniques for Toolbox. You should also be aware that other PDM products exist that have special tools to handle Toolbox data specifically. This includes SolidWorks Enterprise PDMs, which is not covered in this book.

Because of the nature of the SolidWorks Toolbox, you cannot do much with manual file management of Toolbox parts. Toolbox parts need to reside in the library. Sometimes, Toolbox parts may exist in places other than the library, and at these times, you need to be careful, particularly if your Toolbox parts frequently have configurations added to them.

The case of file management when Toolbox parts are accessed from outside of the Toolbox library is much simpler if you are working with sizes as individual parts instead of configurations. If you have a copy of a part with configurations, it could easily happen that any given part in any given user's Toolbox has configurations that another part in another user's Toolbox does not. It is true that since SolidWorks 2007, Toolbox can re-create missing configurations, but in certain situations this may not work (including if Toolbox happens to be turned off). I do not recommend tempting fate by depending on Toolbox in this way.

The best practice recommendation is not to send Toolbox data along with “Pack-and-Go” assemblies if the recipient of your assembly has Toolbox. Find out if people you send assembly data to do not have Toolbox, and then you should send the Toolbox parts. If possible, it is best practice to use the Create Parts setting (rather than configurations) if you send assemblies to other people to avoid completely any difficulties of file management and missing configurations.

If you have multiple copies of Toolbox parts on your computer or network drives, I recommend you delete them, or at least move them to a place that SolidWorks cannot access. The number one rule of file management with SolidWorks is that all SolidWorks documents should have unique filenames. The other side of that is that you should never have different documents with the same names. If one Toolbox part has ten configurations and another one of the same name has only five, you have different documents with the same names.

Caution

The bottom line to this entire discussion is that when you use configurations, Toolbox is guilty of violating the most important file management rule by design. You can certainly make it work, but you have to be careful to avoid multiple copies of Toolbox parts with different configurations. ■

Alternatives to Toolbox

After discovering that some drawbacks exist to the Toolbox functionality, many SolidWorks Administrators ask what the alternatives to Toolbox are. Certainly you can make Toolbox work, but you may not be able to avoid using a series of progressive workarounds. You may be lucky, and your requirements may turn out to be completely compatible with the way Toolbox works, or maybe you can work around the shortcomings in some other way, using tools outside of SolidWorks to cover the data management or metadata issues.

If you're not one of the lucky ones, and you still need a library but can live without the 70 percent functionality that Toolbox provides, the most common solution is to use Toolbox to create a library of static parts. You can still use all the automation tools to add custom properties to the parts before they are saved out. Generally, people who choose this option just use the regular library functionality.

Of course, third-party libraries exist, but you probably won't see any additional value in them. With any library, you are going to need to do some customization work, and Toolbox gives you all the tools you need to do it.

By using an alternative to Toolbox, you are giving up the Smart Fastener integration with the Hole Wizard holes, and other Toolbox specific functionality, such as a nicer method for swapping parts.

Summary

The promise of SolidWorks Toolbox is great. It provides a very valuable list of functionality. Finding situations in which the functionality that it promises actually works as promised is the primary challenge Administrators face when establishing how it will be used at a company. You should not view Toolbox as a plug-and-play hardware library unless you are an individual user and tend not to share assemblies with other Toolbox users. Toolbox requires thought and understanding to realize its full potential and to make it work in the typical multi-user environment with a lot of file sharing.

Combining Toolbox with manual file management techniques requires either a great deal of discipline to avoid multiple copies of parts with different configurations or using copied parts instead of configurations.

Many Administrators choose to create static libraries based on parts made in Toolbox. To do this correctly, the task of creating library parts should be centralized, and a written procedure followed to make sure it is done correctly. Library documents should contain specific information, such as custom properties, tags, configurations, filenames, and so on, that conform to an established standard.

Establishing Libraries

Establishing libraries in SolidWorks is essential to being efficient in production modeling environments. Libraries enable you to reuse data from work you have already done and use work other people have already done, as well as help promote consistency. Libraries are not just for parts; they are also for features, sketches, annotations, assemblies, styles (favorites), blocks, and many other types of data that take time to create in SolidWorks. If you can remember that you have already created certain data, you can save time by not creating it again. The best way to do that is to have standardized places where you store things that you might want to reuse later. Libraries save you time and money if you work on your own, but especially if you work with other people.

In addition to the default libraries installed along with SolidWorks, which are located by default at `C:\ProgramData\SolidWorks\SolidWorks 2010\design library`, you can establish locations for custom library folders at `Tools ➤ Options ➤ File Locations ➤ Design Library`. I recommend for multi-user installations that you set aside a network location and put all your library data in that location, pointing each user to the folder. It is preferable to use a Universal Naming Convention (UNC) path in the Design Library path of `Tools ➤ Options` rather than mapped drives for network drive locations. If you are a one-user shop, make sure your library location is on a drive other than your operating system drive, and that it is backed up.

IN THIS CHAPTER

Creating parts for libraries

Using features from a library

**Getting content from 3D
Content Central**

Using Dissection results

Part II: Installing SolidWorks

Another side of having much stored data is being able to access it. The SolidWorks Search functionality might be an important time-saver in this regard. Browsing for filenames is not always the best way to make data accessible when you need it; consider using custom properties or even tags to utilize other search possibilities.

One of my longtime practices is to put all my favorite SolidWorks data into subfolders of a top-level folder named Library. This helps me in many ways. First, it makes it easy to move library data from one computer to another, and if someone else needs to share the same data I'm using, it is always easy to find. The types of data that go into this folder are basically anything for which a path is identified in the Tools ⇄ Options ⇄ File Locations list. This includes my templates, formats, and all other types of commonly reused data. I have stressed using this practice in other chapters, and it is important for many reasons.

Libraries take on a different meaning if you are trying to incorporate them into a Product Data Management (PDM) scheme. Some libraries must remain outside the vault (such as feature, sketch, block, or weldments profiles), and some can be used inside or outside the vault (such as parts or assemblies).

Creating Library Parts

Creating library parts is something you will do continuously throughout your SolidWorks career, but most frequently early on. You might consider assigning one person to be responsible for creating and maintaining the library with a consistent list of custom properties, mate references as appropriate, and configured sizes as appropriate.

On a basic level, library parts are easy to create: you just save them to a particular folder. However, if you are going to get the most benefit out of the parts, you need to know how to do more than just save them to a directory.

Adding parts to the Design Library

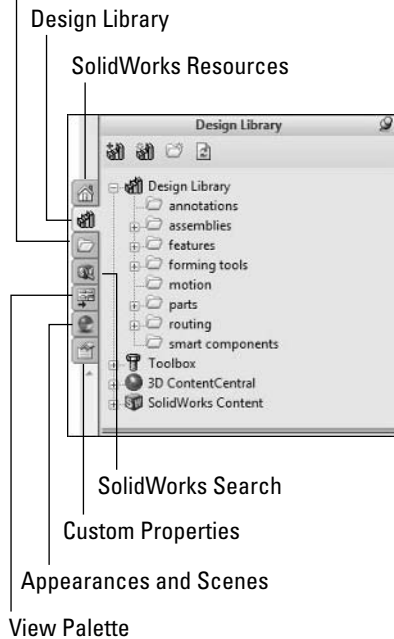
The Design Library is represented by the stack of books tab icon in the Task Pane, typically found to the right of the graphics window in SolidWorks. The location for the library on your hard drive is specified in the Tools ⇄ Options ⇄ File Locations ⇄ Design Library location. Any subfolders you place under that location appear as folders in the Design Library. You can also use the icons in the row just under the Design Library name to add a file location and add a new folder to the active location. These icons are identified in Figure 11.1.

You can add the current part to the library using the icon at the left end of the row. This opens a PropertyManager interface, and enables you to specify several aspects of the active item to add to the library. For example, you could select only a logo sketch or a particular feature that you will need to create frequently. Figure 11.2 shows the Add to Library PropertyManager interface.

FIGURE 11.1

The Design Library control interface is a tab on the Task Pane.

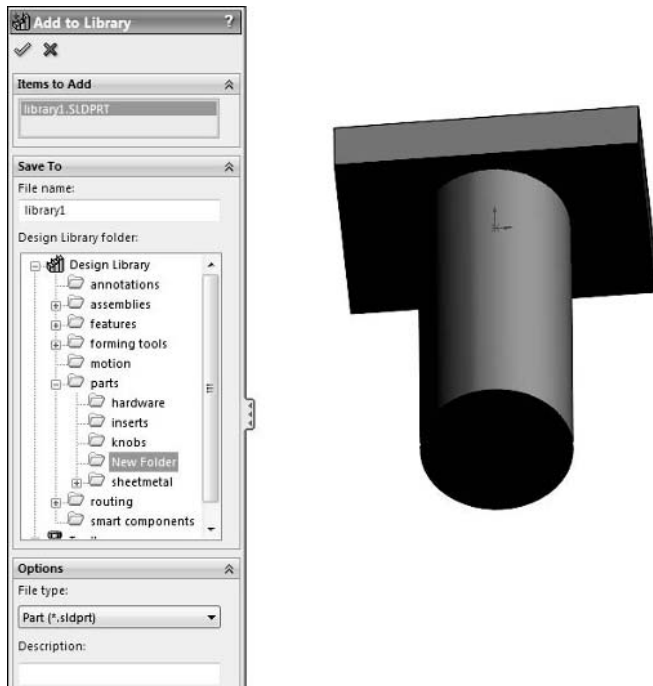
File Explorer
(Workgroup PDMI)



The Design Library is not just for parts; it is also for many other kinds of geometrical data, including sketches, blocks, annotations, assemblies, features, forming tools, Smart Components, and many other types of items.

FIGURE 11.2

Adding a part to the Design Library



Using Mate References

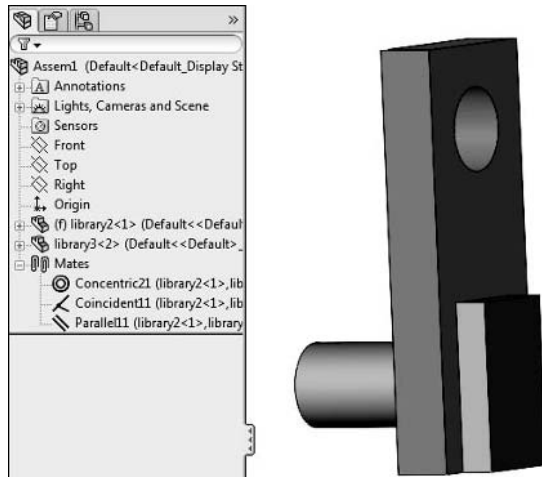
The reason I include Mate References in a book for Administrators is that they must be established before they are used in order to work in a library. Mate References are dependent on setup; they must be saved with Mate References attached to the appropriate geometry so that when a user drags them into an assembly from the Library window, the parts simply snap into place without depending in the user having much specialized knowledge or needing any special effort. This is both a positive and a negative aspect of Mate References. The positive side is that it removes the need for users to learn another technique. The negative side is that users cannot intervene (short of using manual mating techniques) to help Mate References work if they are having difficulty.

A *Mate Reference* is a face, edge, or set of faces or edges that looks for similar entities it can mate to when dragged into an assembly from the Design Library. Most commonly, cylindrical faces mate concentrically with other cylindrical faces, and flat faces mate coincident to other flat faces. If you identify the circular edge of a hole as a Mate Reference on, for example, a rivet part in a library, when the rivet is pulled into an assembly, it automatically looks for other cylindrical edges to which it can mate. A situation like that produces both concentric and coincident mates with pairs of cylindrical and planar faces.

Other types of features can actually be used for alignment, and in certain situations three mates can be assigned automatically. A pair of cylindrical faces can get the concentric mate, the coincident mate to make the ends flush, and a parallel mate to force alignment, as shown in Figure 11.3.

FIGURE 11.3

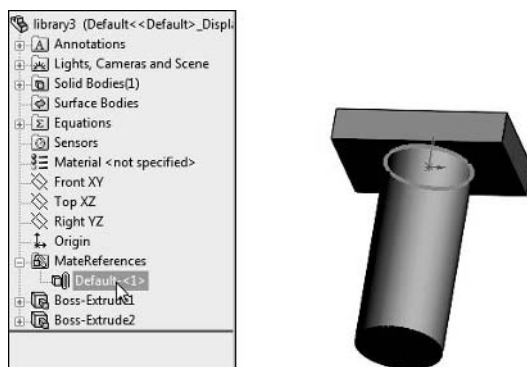
Mating a T-bolt into a hole with three mates



Look at how this works behind the scenes. First, look at the individual parts. The T-bolt, along with its FeatureManager, is shown in Figure 11.4. Notice the Mate Reference entry in the FeatureManager.

FIGURE 11.4

The FeatureManager of the T-bolt shows the Mate Reference as a circular edge.



Part II: Installing SolidWorks

Because Mate References are shown in a folder, you are allowed to have multiple mate references in a single part. You would use multiple Mate References in a single part if:

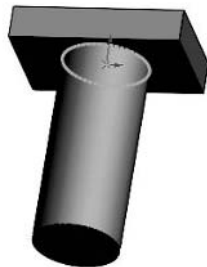
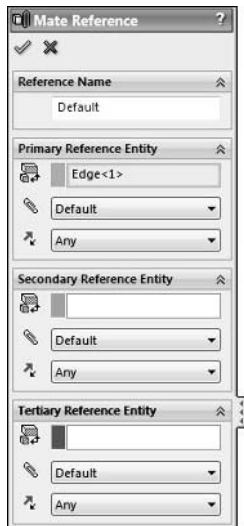
- The part with the mate references was used frequently, and thus stored in the Design Library
- The part with the mate references was mated to another part in a predictable way, and a third part was mated to it

An example of something like this might be a shaft coupler, where the coupler would be mated to a shaft on each end of the coupler. This would require two Mate References to achieve, and would have to be done in two steps: first applying the coupler to an existing shaft, and second applying a second shaft to the newly placed coupler.

Going back to the T-bolt, Figure 11.5 shows a simple setup with the Mate Reference. The circular edge between the head and the body of the bolt acts as a reference for both a concentric mate between the cylindrical face of the bolt and the cylindrical face of the mating hole, as well as the flat underside of the head of the T-bolt mating to the face of a plate.

FIGURE 11.5

Using the Mate Reference PropertyManager to set up a simple Mate Reference

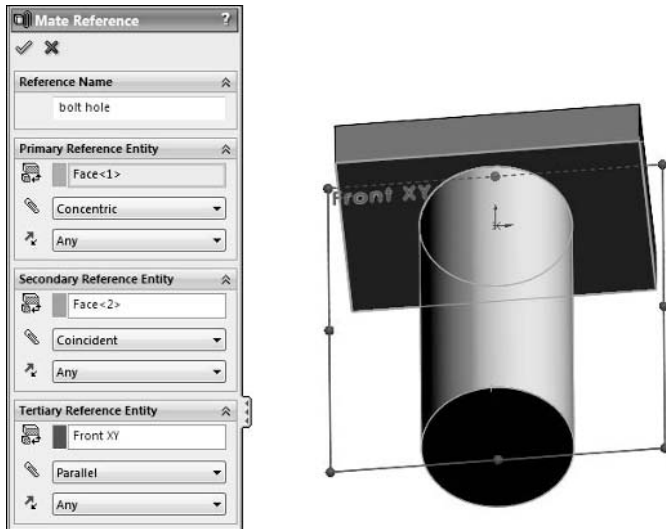


Notice in this case that the Reference Name is left at Default, as well as the mate type and alignment. When a part does not have a specific place to go, it is best left rather generic. If the part can go into any hole with a flat surface outside, then you should not over specify the Mate Reference. If you attempt to put too many conditions on the Mate Reference, the Mate Reference fails most of the time.

If the part has a specific type of geometry that it mates to, you can automate the Mate Reference a little more to include placement in the hole as well as alignment. For typical screws, alignment is usually not necessary. For a T-bolt like this part, it is more useful. Look at how the T-bolt might be set up to automatically align when placed into an assembly. Figure 11.6 shows a more-thorough specification of the various Mate Reference parameters.

FIGURE 11.6

Setting up a named three-entity Mate Reference on a T-bolt

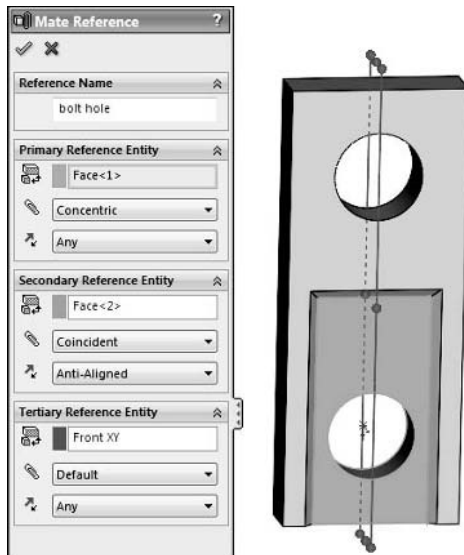


The Primary Reference Entity in the Mate Reference definition is the cylindrical face of the body of the bolt. The Secondary Reference is the flat underside of the head. The Tertiary Reference is the front plane. In this example, I gave the Mate Reference a name so that SolidWorks can very quickly identify if the Mate Reference can be applied to a part in the assembly. If any of the references do not match, the Mate Reference will not be applied. Therefore, the more definition your Mate Reference has, the more specific the situation needs to be in order for it to be applied.

Figure 11.7 shows a part that will accept the Mate Reference in the T-bolt. Notice it has the same name and the same types of references selected for the Primary, Secondary, and Tertiary Reference entity.

FIGURE 11.7

Setting up a corresponding Mate Reference on a T-bolt seat with the same name as the Mate Reference on the bolt



A Mate Reference that does not contain and match the references will not be mated automatically to this hole. The T-bolt setup (refer to Figure 11.5) will mate to either hole (the one in the inset or the one flush to the outer face), but the downside is that it will not be aligned if it mates to a hole without using the mate reference. You need to drag the bolt over the hole (instead of simply over the part) for the Mate Reference to activate. When you have a Mate Reference on the part being dragged from the library and not on the part that is already in the assembly, the Mate Reference acts like SmartMates when you Alt+drag an edge or face on one part to another. To obtain more in-depth information on mates and SmartMates, check out the *SolidWorks 2009 Bible* (Wiley, 2009) or the current version.

Including metadata in libraries

SolidWorks enables you to include various types of metadata, or non-geometrical text-based information, into many types of geometrical data. For example, you can use custom properties with part files, along with tags, comments, and other tools. SolidWorks Search and the FeatureManager filter are great ways to make use of metadata to find or sort documents.

Custom properties and tags

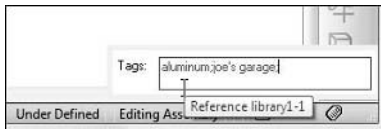
Custom properties and tags are text metadata that can be used for search, Bills of Materials (BOMs), links to annotations, and the FeatureManager filter. Custom properties consist of a property and a value, while tags are stand-alone keywords.

You should already be familiar with the use of custom properties in SolidWorks documents — a property name is accompanied by a value, such as Vendor with a value of Joe's Garage. Tags are a more recent addition to SolidWorks, and many users and even Administrators may not be familiar with how to use tags. They are essentially stand-alone keywords you can associate with any document. They are different from custom properties because they are just a single word without a property name. A tag such as "aluminum" could be assigned. The fact that aluminum is a material is assumed. Tags are great for searching, but not so great for use in a chart such as a BOM. With a BOM, you need to classify the information without any assumptions for the headers in the BOM to put the right information in the right column.

You can apply a tag at the lower right-hand corner of the SolidWorks graphics window. A small, yellow icon serves as the entry point for the interface shown in Figure 11.8.

FIGURE 11.8

Tagging a part in an assembly



In the part document, you can tag features and sketches. You have to add tag information to a part from the assembly level, then select the part, and then enter the tag. Tag search results show up in SolidWorks Search, SolidWorks Explorer Search, and the FeatureManager filter. The tag box in the bottom-right corner of the SolidWorks window activates when you click the Tag icon and stays open until you click the Tag icon again. When it is grayed out, it will not accept tag data. When it is ready to accept tag data, the Tag box displays current tags for the selection. If there are no tags for the selection, the text adds a tag.

Note

SolidWorks Labs (<http://labs.solidworks.com>) has an application called **Tagger** that simplifies adding tags outside of the SolidWorks environment. ■

Using Library Features

Library features are tools that I consider underused. SolidWorks has provided the capability to store features that can be attached to other parts in a very flexible way. If you create parts that share certain types of geometry frequently, this type of capability is what you need to be using. For example, if you design digital cameras, you frequently include a tripod mount, battery compartments, data cable hookups, Secure Digital (SD) card slots, and other standard features. Each of these features might have several actual SolidWorks features that are associated with them. If you design each one from the ground up every time, you may be missing an opportunity to save some time and reuse work that you only need to do once.

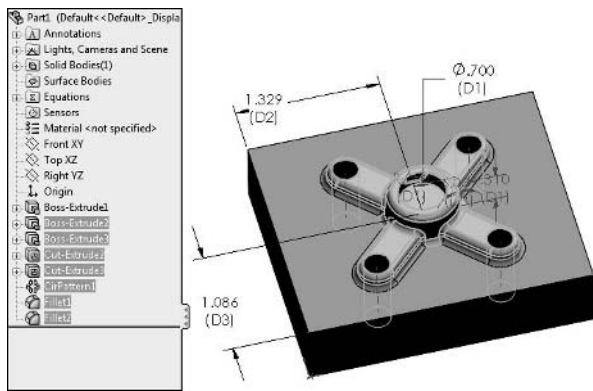
Part II: Installing SolidWorks

Library features can be simple commonly used holes, or complex combinations of features. The idea with a library feature in SolidWorks is that you build the feature, or set of features, with relations to an existing part, and then put the feature on a new part. This makes the library feature a flexible, automated way to copy geometry between parts. Because it is so flexible, there are also many ways in which it can fail. For this reason, you need to set up relations with an eye toward simplicity and the reliability of reattaching the relations.

For example, say you have the set of features shown in Figure 11.9. This includes two extrudes (besides the base feature), two cuts, a pattern, and two fillet features. The extrusions are blind, and the cut is through all. When setting up a feature like this, you want to be careful about what you select for dimension and sketch relation references because you will need to reattach everything to the new model. It is best if you can reduce the number of external references as much as possible.

FIGURE 11.9

A set of features to make a Library Feature



With the features in the FeatureManager selected, choose File ⇨ Save as from the menu. From the Files of Type pull-down menu, select Library Feature Part file. This gives the file an extension of *.sldlfp, places a green L on each feature of the Library feature in the FeatureManager, and makes the icon for the file a stack of books. All of this is shown in Figure 11.10.

You should save the file into a path included in the Tools ⇨ Options ⇨ File Locations ⇨ Design Library paths. When this is done, the path shows up in the Design Library list of folders, and the library feature file shows up in the list.

Figure 11.11 shows the interface that appears when you drag and drop the library feature onto the part to which you are adding it. In this case, three external references were created. One was the face that the feature is sketched on (listed as the Placement Plane), and the next two were edges that the dimensions were driven from. Notice the References list in Figure 11.11. Edges will highlight in the upper image, and you have to select an appropriate equivalent edge on the new model.

FIGURE 11.10

The FeatureManager of the Library feature file shows the library icon and the L on the feature icons.

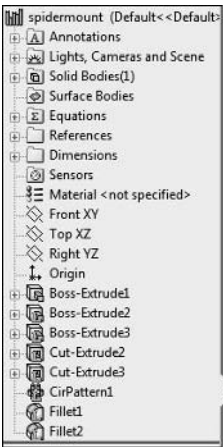
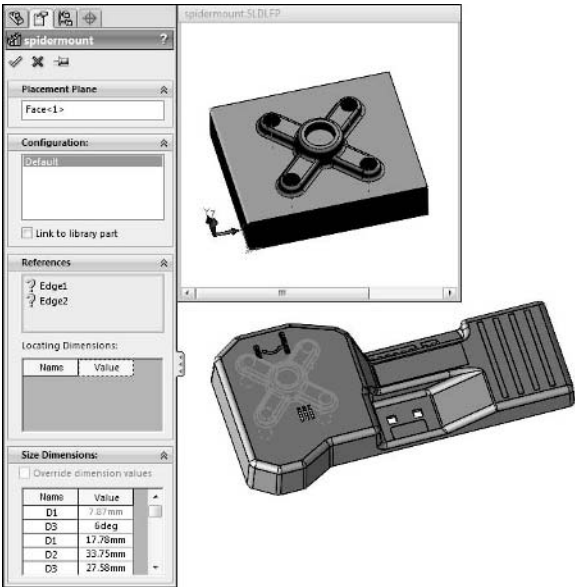


FIGURE 11.11

Placing the feature on the part



This quick demonstration of the functionality is intended as an administrative overview of the Library feature functionality. For more details on the usage, refer to the *SolidWorks 2009 Bible* (Wiley, 2009). As the SolidWorks Administrator for your company, you need to be able to recognize the type of data that can be reused and the best way to automate its application. Library features are certainly a worthy functionality to look into to save yourself and your company a significant amount of time.

The major difference is the ability to use design tables (configurations) with library features, while palette features do not support configurations. The decision to use one or the other can best be explained as follows:

- Use Library Features when a set of known values can be defined and reasonably managed for a range of sizes. For example, there are less than 20 sizes for SAE J1926 Ports; therefore, they are definable and manageable. In this case, select a size then plug-and-play.
- Use Palette Features when the number of sizes may be known but defining and managing all of them is not reasonable. Example: O-Ring Grooves can be defined for standard and custom sizes. Therefore, when you consider male & female glands and static & dynamic applications, definition of all possibilities is close to impossible. In this case, you apply a generic feature and then define or look up the required values to be used.

Accessing SolidWorks Content and 3D Content Central

SolidWorks provides some sample content for several libraries, including electrical and pneumatic schematic blocks, Routing libraries, and Weldment profile libraries. In addition to these, you can also access supplier and user-created content from the 3D Content Central Web site. All of this is available through the Design Library.

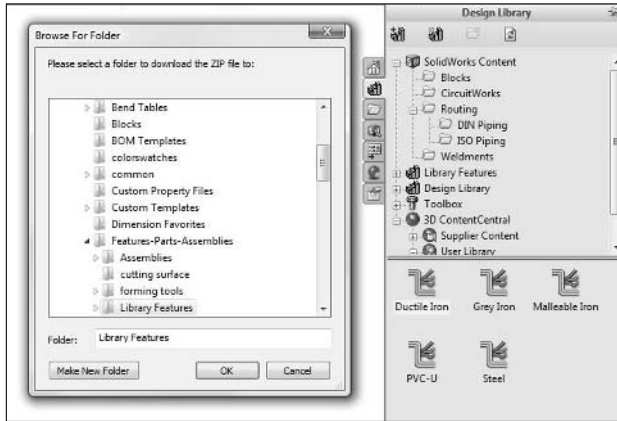
To access, click the + sign next to the SolidWorks Content entry in the Design Library to expand available download content. Figure 11.12 shows the content of the DIN (Deutsches Institut für Normung) standard piping libraries. The information shown in the lower panel is actually a zipped library. To download the library, Ctrl+click any of the icons in the lower panel. Download the zip file to an archive location, and then unpack the zip data to a library location.

There is no indication where the downloaded files are hosted, but it seems clear that you will need an Internet connection to make the download happen. If your workstations are not connected to the Internet by default, then you should be able to download the files from a computer with SolidWorks installed that is connected and then extract the files to each machine individually or to a central network library location.

When downloading data from the internet, there is no guarantee that the parts you download are correct in any way, or that they have all the detail that exists on the actual parts. This is true regardless if the source of the data is a corporation or a private individual.

FIGURE 11.12

Sample SolidWorks Library data



For each type of available library data — sketch blocks, CircuitWorks parts, Routing, and Weldments — you can create your own library data manually, place it in the appropriate folders, and use the content in your parts and assemblies.

Examine a Routing library component

The Routing library contains a great example of library content. Routing is a complex application and for the most part is well beyond the scope of this book, but because its use of libraries is so sophisticated, I include it here for that purpose alone.

Figure 11.13 shows selections from one of the parts in the downloaded Routing libraries. The features, configurations, and configuration specific custom properties come together to make a successful library part with all the necessary sizes and a design table to enable the user to create more sizes very easily if necessary.

If you aspire to automate design at your company and use standard off-the-shelf components in various sizes, you should consider emulating a part like this. The features are simple, the detail is sufficient, the associativity is complete, the capacity for adding more sizes is all there, and even the metadata is flexible and fits the entire range of sizes.

FIGURE 11.13

A sample part from the downloadable Routing library



Adding to the Weldments library

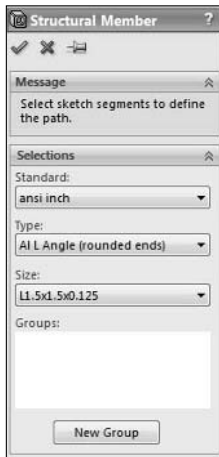
The Weldments library is a downloadable library consisting of profile sketches for various standard structural shapes. Weldment profiles have their own entry in the File Locations paths list, and the default is C:\Program Files\SolidWorks Corp\SolidWorks\data\weldment profiles. Again, you should relocate it so that it is not on the same drive as your operating system and not in same folder structure as your SolidWorks installation.

The Weldments library is not accessed in the same way that you access the other libraries discussed here. It is more like the Toolbox in that you select the size and shape profile from a set of drop-down menus in the PropertyManager.

In addition to the downloadable data, you can create your own profiles for the Weldments environment. The folders are organized by Standard, Type, and Size, with the filename of a library feature file (*.sldlfp) giving the size, as shown in Figure 11.14.

FIGURE 11.14

The Weldments profile library consists of a Standard, a Type, and a Size.



Note

SolidWorks has created static libraries using configurations (the routing part from earlier in this chapter) and libraries using individual files (the weldment profiles just discussed). In addition to the dynamic configurator of Toolbox, I think it covers the complete range. To me, the most effective of the three styles is the routing part because of the design-table-driven configurations. Sketches and library features are certainly configurable, and the Weldment profiles could have been done differently. Mainly, it goes to show that you can create successful libraries using any available technique, regardless of individual opinions. ■

Using Dissection Results

SolidWorks 2008 added a function called Dissection. Dissection takes apart all your part, drawing, and DWG/DXF (2D format) documents and stores the individual extrude (boss and cut) features, sketches, blocks, tables, and views in independently searchable items. It uses the dissected items as what SolidWorks calls Design Clipart.

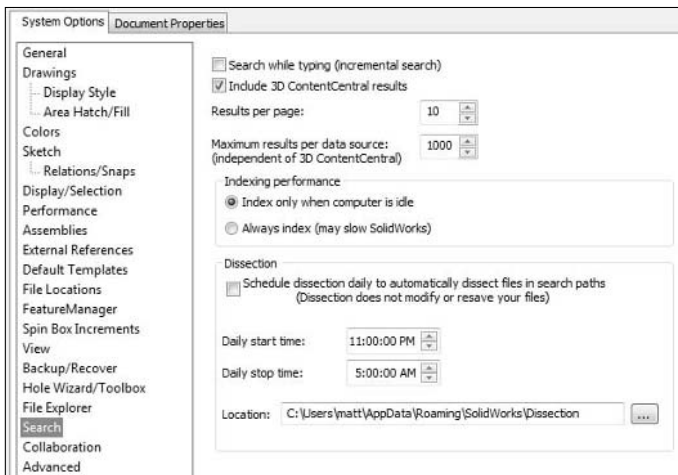
If you do certain types of design work that requires interfaces with many standard components, you might find this kind of thing useful. If you do a lot of custom plastic parts or styling, you may have difficulty finding a use for Dissection.

SolidWorks Search

Dissection starts as a process run by the SolidWorks Task Scheduler that is run automatically on a new installation of SolidWorks. It is scheduled automatically out of the box to run at 11 p.m. every day and to look through all your SolidWorks documents. If you are still at your computer at that time of the evening, this can be a frightening experience. If you have run into this function and want to turn it off, choose Tools ⇨ Options ⇨ Search ⇨ Dissection, and simply deselect the Schedule dissection daily option. Figure 11.15 shows the settings associated with SolidWorks Search, including the Dissection settings.

FIGURE 11.15

The SolidWorks Search settings



You can also access Dissection settings through the SolidWorks Task Scheduler; choose Start Menu ⇨ Programs ⇨ SolidWorks. The result of the Dissection process is a list of features, sketches, annotations, and more that you can access through the SolidWorks Search tab in the Task Pane, as shown in Figure 11.16.

In Figure 11.16, the keyword “tab” returns many 3D Content Central data. 3D Content Central is an online repository for vendor-created, name-brand engineering components used for many general purpose modeling situations.

FIGURE 11.16

SolidWorks Search looks for the keyword "tab."



Windows Desktop Search

Windows Desktop Search is behind the SolidWorks Search functionality. You can add or remove it at any time. The advantage of using this over any of the other desktop search utilities is that the Windows Desktop Search is linked to SolidWorks Search, which works from within SolidWorks and displays results in the Task Pane. Indexing is required to use SolidWorks Search but can be limited to certain folders, or to include 3D Content Central.

Summary

SolidWorks has extensive capabilities when it comes to library creation and management, especially if you include the Toolbox functionality. Library features offer a lot of flexibility. Mate References include time-saving automation. Smart Components use equal parts library parts, library features, and auto-sizing automation.

Only you know your company's needs for design automation and data reuse. Libraries of various types as well as the prudent use of metadata should enable you to quickly find and apply data that is used frequently, but you need to check carefully any data you obtain from sources that cannot vouch for the accuracy or completeness of the data.

Part III

Maintaining SolidWorks

This part brings you information on topics such as support questions, installation of service packs, and new versions. Change management is a huge topic that CAD administrators come up against quite often, and here you find a chapter devoted to that topic, meant to help you get through some of the tough changes that you must drive as a CAD administrator.

IN THIS PART

Chapter 12
Writing a SolidWorks Standard

Chapter 13
Using Custom Properties

Chapter 14
Establishing Best Practice Rules

Chapter 15
Automating Tasks with the API

Chapter 16
Supporting SolidWorks Users

Chapter 17
Collaboration

Chapter 18
Managing Change

Chapter 19
Rolling out New Service Packs

Chapter 20
Rolling out New Versions

Writing a SolidWorks Standard

If you are trying to find SolidWorks standards information, you may not be meeting with much success. Standards information for general drawings is plentiful enough, and for AutoCAD you can find a lot, although it is usually proprietary. However, if you are looking for SolidWorks standards and more specifically SolidWorks 3D standards, you are probably drawing a blank. One reason companies do not share this information is that it is often proprietary. Beyond that, most companies probably have not written a 3D portion to their SolidWorks standard because it is not clear what it would contain in the first place.

The first step in figuring out how to write a standard for SolidWorks is figuring out what a SolidWorks standard should standardize. Chapter 14 covers this in terms of exploring best practices, but it turns out standards and best practice concepts have a fair amount in common.

However, 2D standards are readily used. They tend to address layers, colors, arrow sizes and styles, and line weights and types; how dimensions, annotations, and charts should be laid out; and maybe specify a drawing border. In addition, they might detail sheet order for complex projects like architecture or piping. This is expected. A SolidWorks standard has to cover all this as well, because for most people, 2D drawings are still the way to communicate with manufacturing, or part of the legal document that specifies the design.

When it comes to 3D, what needs to be standardized is much less clear. The answer may lie in fundamental questions about why 2D standards are used in the first place. No one disputes the need for 2D standards.

2D drawing standards are used to make your drawings consistent from one user to another and between companies, mainly to ensure they mean the same thing to everyone. The interpretation of the drawings should be the same regardless of who has created the drawing or who picks it up to read it.

IN THIS CHAPTER

Identifying a SolidWorks standard

Implementing standards

The 2D side of SolidWorks is well-covered territory. In AutoCAD, it doesn't matter how the 2D geometry is created. In fact, the standards rarely address how lines on a drawing are created, or whether they are drawn directly, copied, offset, trimmed, extended, or created using any other technique. In SolidWorks, you would never dream of regulating the geometry itself, but you would be more interested in how the geometry is made.

Given this, a SolidWorks standard should address how a model is created and should ensure that when people have to use its data, it is accessible by every one. It should specify modeling techniques, make many recommendations, and establish boundaries not to be crossed. If you think a SolidWorks modeling standard looks a lot like a set of best practice rules or suggestions, you're correct.

Defining a SolidWorks Standard

A SolidWorks standard needs two distinctly different sections: one on 2D results and one on 3D methods. The difference between making drawings and making models in SolidWorks is significant. 2D drawings, regardless of how they are made, are made with the goal of communicating manufacturing, assembly, or inspection parameters to people responsible for those tasks. Drawings can be printed on paper, in PDF format, or as native SolidWorks documents without an impact on the people reading them.

To some people, the goal of making a 3D model is to create a 2D drawing. That's fine. It's certainly a part of the reason, but more frequently, the goal of a 3D model is to model the actual manufactured part accurately. The 3D data may be used in a variety of formats for a variety of tasks, including making 2D drawings, stress analysis, visualization through rendering, fit or tolerance analysis, evaluation for manufacturing through mold flow analysis software, programming CNC (Computer Numerical Control) tool paths directly from the 3D data, and a number of other downstream functions.

In some of these downstream applications, other users may need to edit or reconstruct the model to some extent. Because of SolidWorks' history-based FeatureManager, the way that the original modeler of a part creates the features affects how people accessing the data later will be able to work with it. The same way a 2D standard ensures other people in the 2D workflow can use the 2D data, your 3D SolidWorks standard needs to ensure that other people using your SolidWorks model can work with the data you send them.

In the end, a SolidWorks 3D standard must standardize methodology and technique to some extent. SolidWorks offers a nearly limitless number of ways to create even the simplest parts. A situation you are trying to avoid is one where an advanced user uses unnecessarily complex methods to create relatively simple parts, and when a user with relatively basic skill sets tries to modify the part, he is completely lost and unproductive. To take it a step further, the standard has to require that the user employ the simplest, most straightforward method to create a part. In some cases it is easy to see that the phrase "simplest, most straightforward method" can be debated, but this is what your standard is there for, to define simplest where it matters most.

Say one user likes to create draft as a part of the feature, and another argues that you retain more control if you apply draft as a separate feature. It is certainly simplest to draft as part of an extrude, but no one will list that as a best practice. Many situations like that exist within SolidWorks modeling practice.

Much of your work in creating standards can be done in the form of templates, but there are areas where the process must be controlled more than the actual output of the process. 2D drawing standards are relatively easy to establish because there are so many of them in existence already, and the ASME Y14 Series of standards lays down a good framework. In SolidWorks, the 2D data doesn't depend much on process aside from the question as to whether you populate the drawing with model items or reference dimensions or DimXpert dimensions.

The 3D data output from SolidWorks is principally governed by the design requirements and the manufacturing process. You might be able to specify that users should avoid certain types of features such as Move Face or Surfacing features for certain types of products, but you can't really write a standard for 3D data in the same way that you would write a standard for 2D data. For 3D, you really have to focus on regulating the process rather than the result or the tools. In a 3D standard, it might be valid to say that all valve cavities must be created using existing and approved library features from a particular folder in the library. You should be able to specify that no splines or loft features be used to create machine design parts, but be careful about using the word "never," because cams are generated using splines, and you may have to use some advanced features to create threads, auger blades, or any one of a number of everyday complex shapes.

Whether a standard is 2D or 3D, the ultimate goal of the standard is to make the company more profitable. It must be understood that CAD standards need to make technical sense, not just enable control freaks or people afraid of change. The road to productivity isn't always the most conservative approach available.

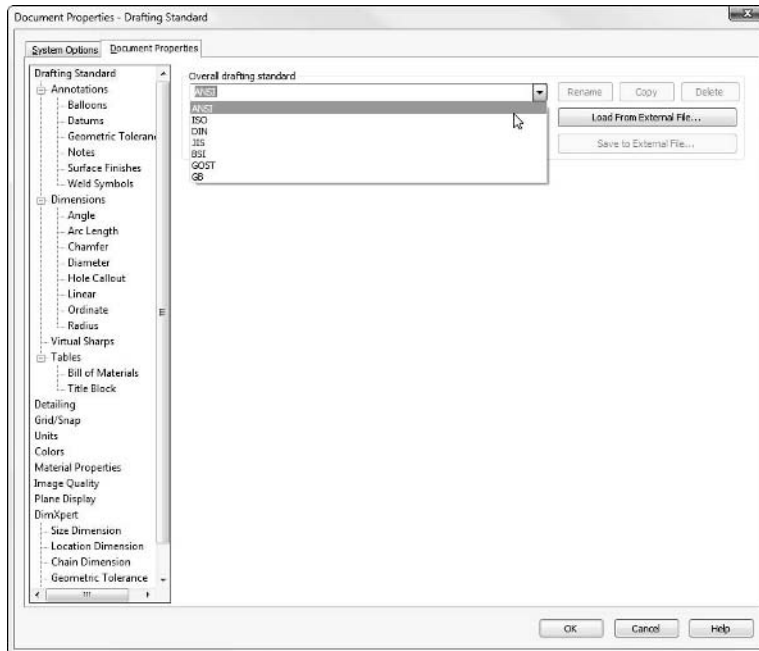
Putting together a 2D standard

2D standards have been around for decades, and they are as tedious to set up and follow now as ever. One thing that has been proved over time is that to get people to follow standards, you have to build it into the process as much as possible; this way, following it becomes automatic rather than a conscious decision. This is where the SolidWorks templates and settings come into play. You don't have to follow a specific set of rules because they have already been followed for you. You're okay as long as you don't un-follow them.

You can specify much of the SolidWorks 2D standard directly in the software settings, without a lot of writing. The Document Properties tab in Tools ⇨ Options contains a Drafting Standard page, shown in Figure 12.1. The Drafting Standard is a combination of all the settings for Annotations, Dimensions, Virtual Sharps, and Tables. A more general standard that you would publish to your CAD department would include these settings, along with the rest of the Document Properties, properly saved inside a document template, and instructions on how to lay out the items on a drawing.

FIGURE 12.1

The SolidWorks Drafting Standard dialog box setup



Using templates and libraries goes a long way toward establishing a real standard based on proper implementation, and with minimal verbal harangue.

A drawing standard needs to be clear enough to communicate your intentions to anyone who reads drawings produced from it, but also general enough that it is easier to follow the rules than to break them. You can get sample CAD standards from the National CAD Standard (NCS) organization (www.buildingsmartalliance.org/ncs/), or from the American Society of Mechanical Engineers (ASME). If you use geometric dimensioning and tolerancing (GD&T), the appropriate ASME Y14.5M has complete standard information that you need. If you use this type of tolerancing, you will not want to deviate from the existing standard. Acceptance of GD&T is spotty, but much lower if you start customizing your own GD&T standard.

Between these sources and other company specific standards you may have access to, you should be able to come up with a clear and usable 2D standard, along with the template material.

Putting together a 3D standard

The 3D standard is much more interesting, and something that has not been seen much. The SolidWorks user community, in general, is maturing; because the software has been available for

14 years, it is common to find users with more than 10 years of experience. However, it is less common to find a company that has implemented a 3D standard or a CAD administrator who has some idea of what shape a 3D standard might take.

Your 3D SolidWorks standard should be a best practice list, or rule of thumb. Most of the list should be characterized as suggestions, because almost every rule has an exception. Examples of items you might find in a 3D standard might be:

- Use the Hole Wizard to create holes when possible.
- Use the 2D placement sketch with the Hole Wizard when possible.
- Use fully defined sketches (except for splines, which should be avoided when possible).
- Use sheet metal functionality with the Base Flange method rather than the Insert Bends method when possible.
- At the bottom of the FeatureManager, parts should contain only a single solid body and no surface bodies (use Delete Bodies to remove excess bodies).
- Part templates should have an Origin, three standard planes, and three standard axes saved into the file.
- Avoid using Regions to select extrusion profiles.
- Use closed loop sketches except where an open loop sketch is clearly a better choice.
- Avoid sketches with more than 30 sketch entities.
- Avoid sketches with more than a few sketch fillets.
- Use feature patterns rather than sketch patterns.
- Name important features in the part.
- If you export a file from SolidWorks for a vendor, always re-import (round-trip) to make sure the data you are sending can be read by SolidWorks.
- When mating parts, mate to reference geometry such as planes or axes.
- Use subassemblies instead of using multiple copies of individual parts or groups of parts.
- Avoid virtual components when possible.
- Do not create in-context relationships between parts that move relative to one another.
- Do not create in-context relationships from or within library parts.
- If you need to rename an assembly that contains in-context relations, use SolidWorks or SolidWorks Explorer to do it; do not use Windows Explorer.

These are just sample entries for parts and assemblies. A more complete set of written guidelines should also be accompanied by a set of document templates. I recommend you break your standard up into sections such that each section has a dozen or so of these statements. You should not be emulating Thomas Jefferson when writing a CAD standard. Flowery and legalistic language leads to more confusion than anything else does. If you are going to channel a writer, channel Ernest Hemmingway, who wrote sparse and to the point sentences. Remember that if you put images in your CAD standard, communicating what you're talking about will be much easier.

In addition, people need to understand the spirit of the law more than the letter of the law. If you were to write a standard that must be followed to the letter, you would need a lawyer to write and interpret it for you. You don't have time for this, and neither do your users. Everyone needs to understand what the standard is trying to achieve. As long as everyone is on board with the spirit of the standard, writing it will be much simpler, and enforcing it should involve fewer arguments.

Writing a 3D standard should be to some extent a group effort and include the most experienced SolidWorks users at your company, documentation people if available, and certainly you as the SolidWorks Administrator. You might also consider having a mentor or guru figure have a look at your standard before you enact it. This person might be a consultant, a lead technical person at a reseller, someone from your local user group, maybe a SolidWorks employee you met at SolidWorks World. It never hurts to get some outside perspective on what you are trying to do. You might even consider asking senior management at your own company to review the document. If you can get their sign-off on it, you will have fewer problems enforcing it later.

When developing standards with a group of people, it may be difficult to avoid the “designed by committee” feeling, so the standard should be written with a single unifying philosophy if possible to prevent it from becoming self-contradictory.

Looking at excerpts from a working CAD standard

What follows are some excerpts from a much longer SolidWorks Reference Guide written by Ed Cyganik, while he served as CAD Administrator for a mixed CAD environment that used SmarTeam (an enterprise level PDM product). The point is that this was a working document assembled by an experienced professional CAD Administrator. This was used as a guide and checklist for users to make sure they were submitting work that was done properly.

SolidWorks Reference Guide

This document should be utilized during creation and as a final review of all SolidWorks objects, Parts, Assemblies, and Drawings. For ease of use, the checklist below contains links to expand or example information found throughout this document and to related external documents. For additional information, refer to the SolidWorks User Guide, On-Line-Help, and the Engineering Data Index that contains links to a wide variety of related documents.

3. SKETCHES: Go to SKETCHES Section

- 3.1. Have the default planes been used wherever possible?
- 3.2. Are Dimension & Geometric Relations attached to the correct entities?
- 3.3. Have you minimized requirements for sketching and reduced the probability of errors?
- 3.4. Has design intent been captured?
- 3.5. Are sketches geometrically correct and properly dimensioned?
- 3.6. Is the sketch proper for feature to be created?

3.7. Have construction aids been used to define sketches?

3.8. Have base features been utilized for sketches?

4. ALL MODELS: (Parts & Assemblies) Go to ALL MODELS Section

4.1. Is more than one component fixed?

4.2. Does the FMT contain minus signs, plus signs, and/or question marks?

4.3. If configurations exist, were they created properly? Are they correct? Have Design Tables been incorporated?

4.4. Are exploded views and/or 3D views available?

4.5. Has the model been reviewed?

4.6. Has the model been visually inspected?

4.7. Have external references been reviewed?

4.8. Would a temporary drawing aid in the design process?

5. PART MODELS & FEATURES: Back to top of checklist or Go to PART MODELS & FEATURES Section

5.1. Has the part been created to suit the manufacturing process? Also, see: ENI_DFM.doc

5.2. Has the part been created to suit assembly needs? See: ENI_DFA.doc

5.3. Have features been aptly named? ()

5.4. Is an "Alter Item" or "As Manufactured" part model required? See AlteredItems_BaseParts.doc & AM&A_MODELS.DOC

5.5. Has the part been constructed smartly?

5.6. Has the Hole Wizard been utilized where possible?

5.7. Have Library & Palette Features been employed?

5.8. Have the following items been created or set up in the part mode ...

5.8.1. ... Material Density?

5.8.2. ... Reference Dimensions?

5.8.3. ... Tolerancing?

5.8.4. ... Annotations? (Notes)

5.8.5. ... Cosmetic Thread Callouts?

5.8.6. ... Surface Finishes?

5.9. Are there suppressed or hidden features in the FMT?

5.10. Is the display correct for all sketches, planes, axis, points, etc.?

5.11. Have requirements for Part Configuration been considered?

continued

continued

- 5.12. Do you have a robust Part Model?
- 5.13. Have Text Features been minimized?

6. ASSEMBLY MODELS: Go to ASSEMBLY MODELS Section

- 6.1. Is the assembly modeled properly?
 - 6.1.1. ... Proper Base or First Component?
 - 6.1.2. ... Use of Neutral/Symmetry Plane?
 - 6.1.3. ... Proper use of Subassemblies?
 - 6.1.4. ... Required Assembly Features?
 - 6.1.4.1. Is an “Alter Item” or “As Manufactured” assembly model required?
- 6.2. If required at drawing level, have the following items, been created in assembly mode for later use...
 - 6.2.1. ... Isometric or 3D Views?
 - 6.2.2. ... Exploded Views?
 - 6.2.3. ... Weld Symbols?
 - 6.2.4. ... Surface Finishes?
 - 6.2.5. ... Reference Dimensions?
 - 6.2.6. ... Tolerancing?
- 6.3. Are there suppressed, hidden or lightweight, “out-of-date” components in the FMT?
- 6.4. Are there any suppressed mating conditions?
- 6.5. Do the terms or appear as suffixes on any components in the FMT?
- 6.6. Have all requirements for Assembly Configurations been satisfied?

PARTS MODELS & FEATURES

- Map out part modeling strategy prior to start. Do this by determining...
 - ... what manufacturing process will be (lathe, mill, cast, forged, molded, sheet metal, and so on). If a part is created on a lathe, start with the proper bar stock. An extrusion should start out with the proper extruded shape & size. The Overall Finished Dimensions are all that is necessary to simulate stock size. Extra material shall not be added.
 - ... the base (first) feature type (extrude, revolve, swept, blend, advanced, solid, thin, and so on). It should provide for the most bang for the buck, it should be the most stable.
 - ... if a neutral plane or axis feature would be helpful.
 - As a minimum, key features should be given meaningful names for quick and easy identification.

- When necessary, take advantage of creating construction features such as planes, axes and points and/or use sketch entities to aid in a design process. Any construction aids should lend themselves to capturing design intent. As new features are created & added, dimension & align them to surfaces, axis, planes and edges, which make sense. Perform these tasks while in an isometric view and/or by using select others.
- If possible, create all holes using the Hole Wizard for additional benefits at the drawing level (Hole Callouts).
- Library & Palette Features should be used whenever possible. To decrease modeling time, consistently dimension common features and reduce the possibility of errors, see LibFeat.doc & LibraryFeatures.xls
- Determine & then create or setup the following part model items:
 - Material Density (For precise weight calculations.) See the “Density Tab” in ENI_STD_NOTES.xls
 - Reference Dimensions should be created in the part model where required but should be kept to a minimum. Excessive use of reference dimensions is a sign of poor modeling practices. Consider changing the dimensioning scheme of sketches to reduce the number of reference dimensions.
 - Add all tolerances in the part model. This is where the definition of form, fit, design intent, and manufacturing requirements are determined. Set up all tolerance types, nominal, plus/minus, limits, symmetric, GD&T, datums, basic dimensions, and so on, in the part model.
 - Annotations or Notes can be created at the part level also. If holes are created using a variety of features (versus the Hole wizard) then use the Annotations command to capture the required information so that parametric notes can be used downstream in drawings.
- During the process of creating Cosmetic Threads, be sure to fill in the Thread callout box to provide for the required notes on drawings.
- Like all other annotations, Surface Finishes should be added at the part level.
- All suppressed (gray) or hidden (wireframe) features in the FMT must be investigated and resolved. Suppressed and/or hidden features are only acceptable if there are configurations present, and the state of the features is controlled in a design table.
- Investigate and set the proper display for all sketches, planes, axis, points, and so on. This is a simple matter of determining whether an item is to be Shown or Hidden.
- Part Configurations should consider Create Design Table with column headings for ease of use.

Enforcing Standards

Everyone hopes that you can avoid standard enforcement. Enforcement is no fun for you, no fun for the person who gets files back with a Redo label on them, and no fun for management, who sees you waste your time and SolidWorks users duplicating their own efforts. Training and encouragement are always preferable to enforcement, but it is just not avoidable.

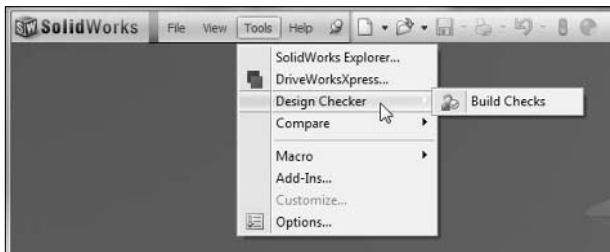
Part III: Maintaining SolidWorks

For enforcement to work, you have to have an impartial judge and some consequences if work is returned to redo. Many companies have a review board that looks at finished drawings and approves them. More rarely, you will find a dedicated drawing checker or even a peer review system such that every drawing must be signed by another CAD user who should also understand the CAD standard intimately.

In addition to performing a manual review, you can use the SolidWorks Design Checker tool. You access it through the Tools menu, as shown in Figure 12.2.

FIGURE 12.2

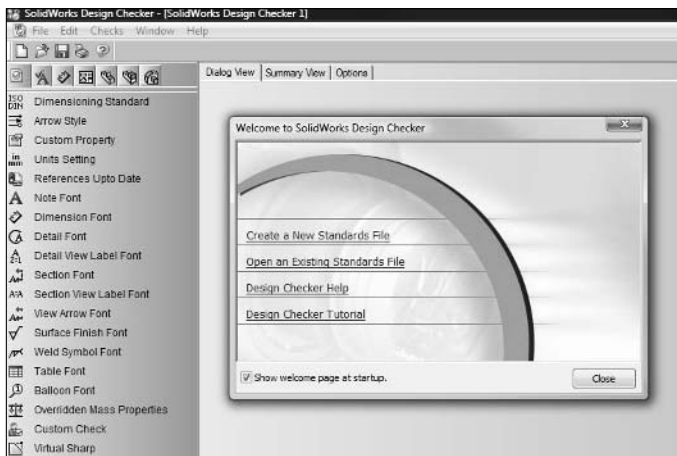
Starting the Design Checker



You can also find Design Checker in the Add-ins list. The Design Checker makes sure that all the entities, such as annotations, dimensions, tables, custom properties, and so on, follow the settings established in a standard file. The first interface is shown in Figure 12.3.

FIGURE 12.3

Setting up the Design Checker



The checker application is primarily intended for checking drawings, but you can also use it to check that the standards are followed in parts and assemblies. It is only available in SolidWorks Professional and Premium. The checker doesn't catch the kinds of things that humans catch easily, but it does catch things that humans can easily miss, such as a font name change.

Summary

SolidWorks standards consist of two parts: a 2D standard for drawings, and a 3D standard for models (parts and assemblies). Most of your 2D standard can be established as drawing templates and settings within the templates. This is the best kind of standard because it is self-applying. The 3D standard is a list of best practice suggestions that can be applied to parts and assemblies. Remember that best practice ideas can rarely be held rigidly. Being unnecessarily rigid about 3D standards is probably not the best choice. The main goal of all standards is to make sure anyone can create a valid document, anyone can interpret it, and everyone is able to work time-efficiently.

If you are going to have standards, you have to be prepared to enforce them. Manual or even group checking of drawings can be tedious. The SolidWorks Design Checker removes some of the tedium from the work and provides customizable search features.

Using Custom Properties

When it comes to standardizing SolidWorks usage, you will find fewer topics more important than custom properties. Custom properties, including configuration specific properties, are meta-data — non-geometrical textual information that tells you something about the document to which the property belongs. You can control custom properties in many ways, including through the basic custom property interface (choose File ⇨ Properties) or the Custom Property tabs used to help standardize custom property entry. You also do not want to forget about tools such as design tables, which can be great for simplifying entry of configuration specific custom properties for large numbers of configurations.

Custom properties can also be controlled through the Update Custom Properties function of the SolidWorks Task Scheduler, through SolidWorks Explorer, and through the Workgroup Product Data Management (PDM) interface, as well as, to some extent, the Windows Explorer interface.

In addition to applications within SolidWorks, custom and configuration specific file properties need to be reviewed and considered for use in PDM/PLM and MRP/ERP applications. In some cases, you may have applications that have one-way or bidirectional capabilities. This is where your involvement with other departments and individuals, along with your thorough knowledge of the SolidWorks software and how to apply good ideas, is important to a successful implementation of SolidWorks.

Tags are another type of metadata that you can use to identify SolidWorks documents and are somewhat simpler than custom properties because they act as keywords instead of properties with values. Tags can be a fast and easy way to gain some indexing capabilities that you may not have in your custom properties scheme.

IN THIS CHAPTER

Why do you need custom properties?

How to build Custom Property tabs

Crafting forms

Make use of configurations and custom properties

Establishing the Need for Custom Properties

If you are trying to save time by automating data entry tasks in drawings or charts, or allow information about your SolidWorks parts or assemblies to propagate to your PDM tool, you need custom properties. Custom properties enable you to enter the data once, in your part or assembly files, and then to reuse the data wherever that type of data needs to be reused. Many PDM packages automatically read the custom property information and use it to fill out data cards on the parts or assemblies.

The custom properties can come from templates; material assignment, mass property, and dimensional data can be linked to properties. It can be manually entered through the SolidWorks interface, through Windows Explorer, SolidWorks Explorer, by macros, or by the new Custom Property tab functionality found in the Task Pane.

Isn't the file naming enough information? Remember the discussion on intelligent and descriptive file names in Chapter 3? Filenames should be used to uniquely identify a file, not to describe it. The description of the contents of the file should come in the custom properties. Examples of commonly used custom properties are "description," "material," "finish," or "DrawnBy." Each of these properties will also have a value associated with it.

Once custom properties are added to your SolidWorks document, where do you use them. It turns out that there are many places. One of the most obvious places is in drawing Bills of Material (BOM). You can also use them when filling out the title block information in a drawing. Very commonly, it is used in Workgroup PDM searches. In fact, it may be more common to search on custom property data than on filenames.

Using custom properties in the BOM

Putting custom properties in the BOM is standard practice for many, if not most, SolidWorks users who make assembly drawings. It is a convenient, fast, and accurate way to get component information onto the drawing.

Additional uses for BOMs should also be explored and considered, such as a "Preliminary BOM," which could be distributed after a design review but prior to the initial release of data. This BOM could identify items such as "purchased" or "manufactured" parts and "Long Lead" items. This BOM would be given to planners and purchasing personnel, providing them with a heads-up they never had before.

Figure 13.1 shows part of a drawing with a BOM populated with custom property data, including description and material.

Building a BOM like this is easy; the next step is to save out what you build as a BOM template, and then share the template so no one has to custom build it again. You can learn the other details of the BOM from other places; for example, I cover it in the *SolidWorks 2009 Bible* (Wiley, 2009). This chapter focuses on the custom properties.

FIGURE 13.1

A BOM built from custom property data

| ITEM NO. | PART NUMBER | DESCRIPTION | MATERIAL | QTY. |
|----------|----------------|-----------------------|----------|------|
| 1 | Chapter 13.001 | Custom Aluminum Frame | aluminum | 1 |
| 2 | Chapter 13.012 | Bottom Bracket | aluminum | 1 |
| 3 | Chapter 13.003 | Headset | aluminum | 1 |
| 3.1 | Chapter 13.014 | Right Crankarm | aluminum | 1 |
| 3.2 | Chapter 13.015 | Left Crankarm | aluminum | 1 |
| 3.3 | Chapter 13.016 | Spokes | aluminum | 5 |
| 3.4 | Chapter 13.017 | Seat Post | aluminum | 1 |
| 4 | Chapter 13.013 | Left Crankarm | aluminum | 1 |
| 5 | Chapter 13.004 | Seat Post Assembly | aluminum | 1 |
| 5.1 | Chapter 13.003 | Aluminum Seat Post | aluminum | 1 |
| 5.2 | Chapter 13.004 | Seat Post Clamp | aluminum | 1 |
| 5.3 | Chapter 13.003 | Spoke | aluminum | 1 |
| 6 | Chapter 13.005 | Front Wheel Assembly | aluminum | 1 |
| 6.1 | Chapter 13.015 | Rim | aluminum | 1 |
| 6.2 | Chapter 13.019 | Front Hub | aluminum | 1 |
| 6.3 | Chapter 13.020 | Spokes | aluminum | 12 |
| 6.4 | Chapter 13.021 | Tire | aluminum | 1 |

| ITEM NO. | PART NUMBER | DESCRIPTION | MATERIAL | QTY. |
|----------|----------------|----------------------|----------|------|
| 7 | Chapter 13.002 | Rear Wheel Assembly | aluminum | 1 |
| 7.1 | Chapter 13.019 | Rim | aluminum | 1 |
| 7.2 | Chapter 13.022 | Rear Hub | aluminum | 1 |
| 7.3 | Chapter 13.023 | Spokes | aluminum | 12 |
| 7.4 | Chapter 13.021 | Tire | aluminum | 1 |
| 7.5 | Chapter 13.023 | Spokeset | aluminum | 1 |
| 8 | Chapter 13.006 | Front Fork Assembly | aluminum | 1 |
| 8.1 | Chapter 13.009 | Stem and Headset | aluminum | 1 |
| 8.2 | Chapter 13.008 | Handlebar | aluminum | 1 |
| 8.3 | Chapter 13.010 | Upper Link | aluminum | 2 |
| 8.4 | Chapter 13.014 | Lower Link | aluminum | 2 |
| 8.5 | Chapter 13.011 | Fork | aluminum | 1 |
| 8.6 | Chapter 13.005 | Drop | aluminum | 2 |
| 8.7 | Chapter 13.007 | Front Lower Assembly | aluminum | 1 |
| 8.7.1 | Chapter 13.008 | Spoke Lower Mount | aluminum | 1 |
| 8.7.2 | Chapter 13.007 | Spoke Lower | aluminum | 1 |
| 9 | Chapter 13.005 | Chainstay Frame | aluminum | 1 |

The BOM in Figure 13.1 is essentially a default BOM, with the addition of the material column, and a couple of setting changes. To add the column, right-click in an adjacent column, select Insert, then choose either Column Right or Left, depending on your needs (see Figure 13.2).

FIGURE 13.2

Adding a column to your BOM

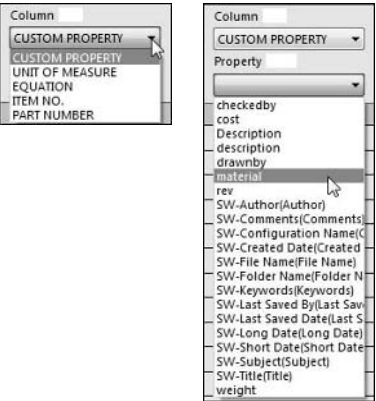
| ITEM NO. | PART NUMBER | DESCRIPTION | QTY. |
|----------|----------------|-----------------------|------|
| 1 | Chapter 13.001 | Custom Aluminum Frame | 1 |
| 2 | Chapter 13.012 | Bottom Bracket | 1 |
| 3 | Chapter 13.003 | Headset | 1 |
| 3.1 | Chapter 13.014 | Right Crankarm | 1 |
| 3.2 | Chapter 13.015 | Left Crankarm | 1 |
| 3.3 | Chapter 13.016 | Spokes | 5 |
| 3.4 | Chapter 13.017 | Seat Post | 1 |
| 4 | Chapter 13.013 | Left Crankarm | 1 |
| 5 | Chapter 13.004 | Seat Post Assembly | 1 |
| 5.1 | Chapter 13.003 | Aluminum Seat Post | 1 |
| 5.2 | Chapter 13.004 | Seat Post Clamp | 1 |
| 5.3 | Chapter 13.003 | Spoke | 1 |
| 6 | Chapter 13.005 | Front Wheel Assembly | 1 |

Part III: Maintaining SolidWorks

Once the column is added, you first select what type of data to put in the column automatically. In this case, I chose custom property. From there, you can select which custom property you want to populate the column. Figure 13.3 shows the interface for accomplishing these tasks.

FIGURE 13.3

The interface for assigning properties to BOM columns

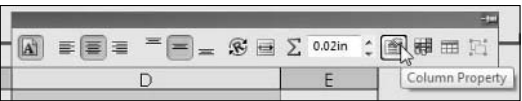


If the property you want is not in the list, you must first add the property to one of the parts. Make sure the property has a value assigned.

To return to the interface where you assign the property to the column, click in the space above the column, and the toolbar shown in Figure 13.4 appears. Click the Column Properties icon, and you are back to the interface shown in Figure 13.3.

FIGURE 13.4

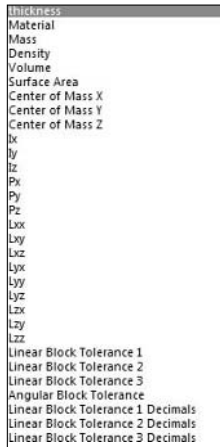
Getting back to the interface in Figure 13.3



Remember that your custom properties can include many types of data. The types of information that you can automatically populate your properties with are shown in Figure 13.5. The Thickness item at the top of the list is not a standard data type, but a link value. All link values and global variables can be inserted into a custom property this way.

FIGURE 13.5

Data types that can be automatically inserted into a custom property



Using custom properties in the title block

A common goal for SolidWorks Administrators is to avoid having users actually needing to edit sheet formats. This can be a difficult to achieve when a lot of information is stored in a format that cannot simply be put on the drawing sheet. The title block in a drawing, for example, is on the format, and you must edit the format to change that information.

Establishing linked notes

Two tools can help you keep users from tangling with the sheet format. One is to have all the title block information come from custom properties, and the other is to use the new functionality introduced in SolidWorks 2009 called, appropriately enough, title block. In this part of the chapter, you will learn how to accomplish both of these setups.

Note

To get started, you will need a drawing with a sheet format. If you are unfamiliar with the terminology, the sheet format is the drawing border and associated data that you can see but not select on a drawing. If you don't have a drawing template with this already set up, I highly recommend using the one provided with the data for Chapter 13. The file is called `inch B.drwdot`. You will have to move it to the folder you specified in File Locations, as discussed in Chapter 8, and use the Advanced option in the New Document dialog box. ■

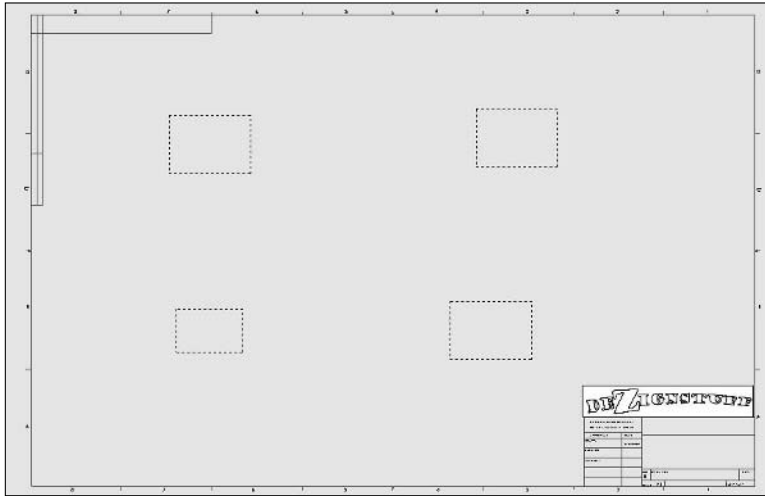
The first step is to explain how to set up notes in the drawing format to pull in custom property data from either the model or drawing document. When data is pulled from the model, it means the data is coming from an external source. When it comes from the drawing document, the source of the data is in the same file.

Part III: Maintaining SolidWorks

Start out with a drawing that has a sheet format in it. Use the template described in the previous Note if you do not already have a template that fits this description. When you open a new drawing using that template, it should look like Figure 13.6.

FIGURE 13.6

Starting from a template with a sheet format



Note

The dashed rectangles are the borders of empty predefined and projected views. When a model is associated with this drawing, it populates the views automatically. ■

Notice that you cannot select any of the drawing border geometry on the new drawing. The items that you can see but cannot select are on the sheet format. You can access the sheet format by right-clicking on the sheet and selecting Edit Sheet Format. You could also expand the Sheet icon in the FeatureManager and right-click the Sheet Format there, as shown in Figure 13.7.

Once you have accessed the sheet format, notice that some notes are already in the lower right-hand corner. These notes use special syntax to link to properties. The notes that display \$PRPSHEET:{property} are linked to a property of the model associated to the sheet. These are blank at this time because there is not currently any model associated with the drawing sheet.

Other notes have values already. These may be either static notes or notes linked to properties within the drawing file itself. The syntax for notes linked to the drawing instead of to the model is \$PRP:"property". In Figure 13.8, notice the property being edited.

FIGURE 13.7

Accessing the sheet format

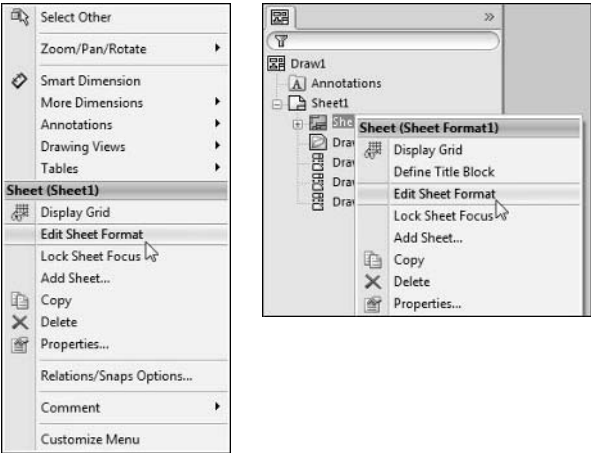


FIGURE 13.8

Syntax for linking properties to model and drawing

| | | | |
|--|---------------------------|--------------------------|--|
| CAD GENERATED DRAWING, DO NOT MANUALLY UPDATE | | | |
| APPROVALS | | DATE | |
| DRAWN | \$PRPSHEET:{drawnby} | 3/24/2002 | |
| CHECKED | \$PRPSHEET:{checkedby} | | |
| MATERIAL | \$PRPSHEET:{material} | | |
| | | \$PRPSHEET:{description} | |
| | | \$PRP:"date" | |
| SIZE | DWG. NO. | REV. | |
| B | \$PRPSHEET:{SW-File Name} | \$PRPSHEET:{rev} | |
| SCALE 1:1 | | SHEET 1 of 1 | |
| 2 | | 1 | |

To create a note with a link, follow these steps:

A

1. Start to create a note by either choosing Insert⇨Annotations⇨Note or using the Note icon on the Annotation toolbar.
2. Place the annotation by clicking in a blank space.
3. Click the Link To Property icon in the Text Format panel of the Note PropertyManager.
4. Select from custom properties for the current document or the model in view specified in sheet properties.



Part III: Maintaining SolidWorks

When you have linked your notes to your properties, unless they were local properties (properties of the drawing document), they will be empty or display the syntax until you put a part or assembly on the drawing. At that point, the property value will display in the place of the note.

If you put a model on the drawing, but the model does not have a property that you have set up a linked note for, you may see an error displayed on the drawing. For example, Figure 13.9 shows the properties from Figure 13.8 filled in, except one which has an error.

FIGURE 13.9

Filled in notes linked to properties

| | | | | |
|--|-----------|---|----------------|-------------------|
| CAD GENERATED DRAWING, DO NOT MANUALLY UPDATE | | <div>description</div> <div>3/24/2002</div> | | |
| APPROVALS | DATE | | | |
| DRAWN Matt | 3/24/2002 | | | |
| CHECKED You | | | | |
| MATERIAL aluminum | | | | |
| | | SIZE B | DWG. NO. BB | REV. ERROR!rev |
| | | SCALE 1:1 | SHEET 1 of 1 | |

In the lower right of Figure 13.9 notice that in the REV. box is a note that shows “ERROR!rev”. This means two things. First, it means that the part on the drawing does not have a property called rev, and it means that the option at View ⇄ Annotation Link Errors is selected. You can avoid seeing these errors simply by deselecting that option.

Establishing the Title Block

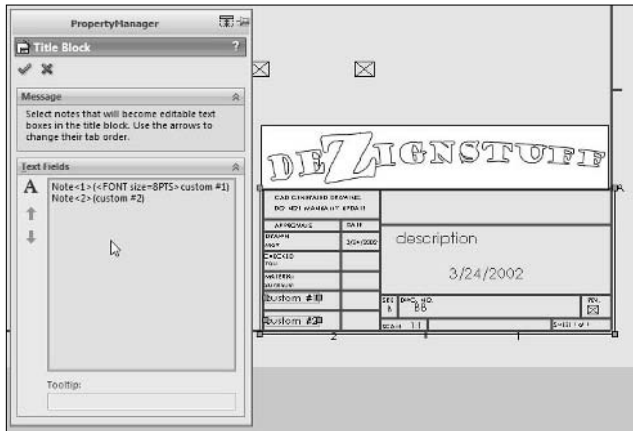
The Title Block functionality was introduced in SolidWorks 2009. Don’t confuse the capitalized Title Block functionality with the generic title block that has existed for decades on all types of drawings long before SolidWorks existed.

A Title Block in SolidWorks is an area you can establish where any notes that lie on the sheet format within the area can be edited as if they are just on the sheet. The way Title Blocks work is that the Administrator establishes a rectangular area on the sheet format and then selects a number of notes from the sheet format and places the notes in the order in which the user should edit the values. Then the Admin saves the drawing as a template. When the user uses the template, she can click in the rectangular area and edit the notes in the established order by typing text, then tabbing through the notes. The Administrator can establish tooltips for each note as reminders for the user while entering text.

Figure 13.10 shows the interface for establishing the Title Block.

FIGURE 13.10

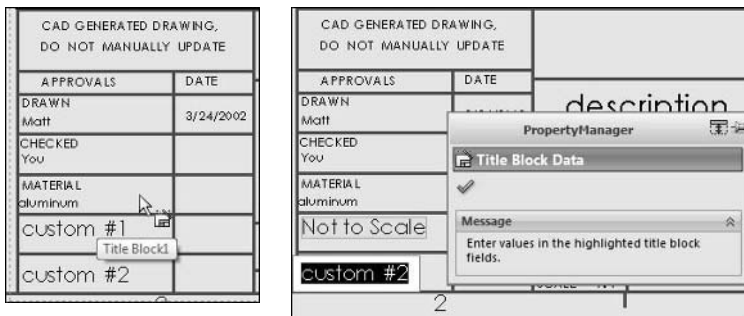
Interface for establishing the Title Block



Notice the tooltip in Figure 13.11. Figure 13.11 shows the drawing in Edit Sheet mode (rather than Edit Sheet Format), with the cursor over the area I established as the Title Block area. Double-clicking the Title Block area activates it, and then you start editing notes by single-clicking one of them. As you complete one note, just hit Tab to move to the next.

FIGURE 13.11

Entering data into a Title Block



Establishing a Title Block on templates that also contain sheet formats is a great idea for several reasons:

- When creating a new drawing, the user does not have to also select a sheet format.
- The size of the drawing is established right from the template.
- Between custom properties and user-entered Title Block text, the user will never have to access the sheet format.

The biggest reason to help users avoid the sheet format is that many new users, and even many experienced users that do not use drawings often, do not understand the function or the necessity for the sheet format. The terminology SolidWorks has chosen doesn't help clear up the mystery at all, either. If you have provided much technical support for your users, you may already know this. "What is the difference between a template and a format?" is one of the most common questions from users getting started on drawings. The Template/Format/Title Block combination erases the user's need to know most of that information, but still gives you the control to standardize.

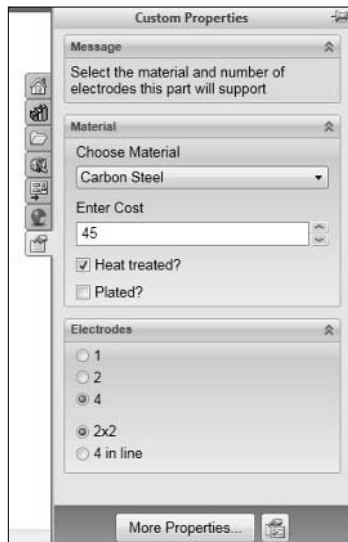
Building Custom Property Tabs

The ability to build custom property tabs was new in SolidWorks 2009. Custom Property tabs reside in the Task Pane and enable you to enter in custom properties using a customizable interface. The SolidWorks Administrator builds the tab and then deploys it to users by placing the file created by the Property Tab Builder into a folder found by choosing Tools ⇨ Options ⇨ File Locations ⇨ Custom Property Files. The builder application creates three different types of files, *.prtprp (part properties), *.asmprp (assembly properties), and *.drwprp (drawing properties).

The interface created by the builder application makes custom property data entry more seamless, more consistent, and faster and easier. The Property Tab Builder looks vaguely like the interface for a visual programming language but is greatly simplified and relatively straightforward. Figure 13.12 shows a Custom Property tab in use.

FIGURE 13.12

Using a Custom Property tab



If you click the Custom Properties tab on the bottom of the series of Task Pane tabs, it reveals the interface for the file type you are currently editing. You can create separate tabs for parts, assemblies, and drawings. You can even have multiple tabs for each file type. You could have one for sheet metal, one for weldments, one for plastic parts, and so on.



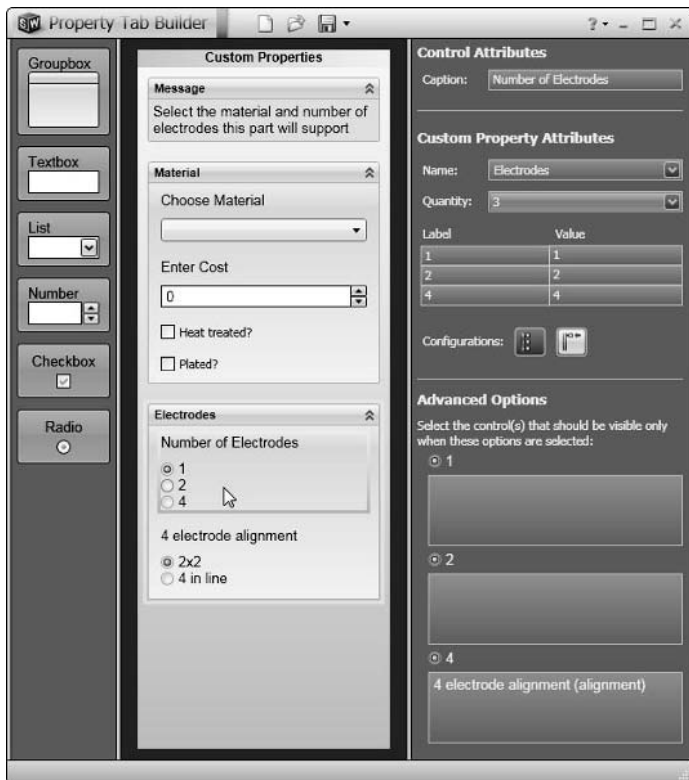
To change to a different tab interface, click the icon at the bottom of the pane and select another template from the list. The More Properties button brings up the old familiar Custom Properties dialog box.

Building the tabs is easy, but be warned: SolidWorks does not make any distinction between regular users and Administrators, so all users will have access to the same tools.

You can find the Property Tab Builder by clicking on the Start menu and choosing Start → SolidWorks 2009 → SolidWorks Tools → Property Tab Builder. The interface for building the interface shown in Figure 13.12 is shown in Figure 13.13.

FIGURE 13.13

The Property Tab Builder interface



Creating the Forms

Creating the property forms is easy, and you may even find it kind of fun if you have any background in programming. You will need to decide how many forms to create rather than trying to put all the possible options on a single form. Typically, I use different forms for different types of components. Certainly, any data that needs to go into the same document should be collected on the same form, but if you have parts made through different processes or for different projects that require vastly different data, the best option will be to create separate forms for each type.

- **Using groupboxes.** Every element of a Custom Property tab must be on a groupbox. You may have multiple groupboxes. In Figure 13.13, the Material and Electrodes headers are both on top of a groupbox. Groupboxes should be used to group together multiple properties related to the same type of properties.

When you create a groupbox, you have two things you need to determine. First, is the caption applied to the groupbox, and second, is the groupbox expanded or collapsed by default? Unless your Custom Properties tab is very long, I recommend leaving every thing expanded by default. Collapsed groupboxes are easy to miss and pointless to expand unless doing so automatically requires scrolling the pane area.

- **Using textboxes.** Textboxes enable the user to enter any type of text data. When you design the form you can leave the box blank, put a default text value in it, or use a link to a custom property. Remember that the caption and the name are two different values; the name becomes the name of the custom property that the field drives.
- **Using lists.** The list control enables you to pre-establish a list from which the user can select a value. This is helpful when you want to standardize options. It helps keep spelling in line and prevents users from entering options that may not make sense. The list can come from a list that you hand-enter into the Builder window, an external text file, an Excel file, or even an Access database file. By default, the box allows four values. If you press Enter after the last value, it makes room for another value.
- **Using number boxes.** When a number box is placed on the property tab, it can accept any number value, but the scroll arrows on the right side of the box only allow increments of 1. You can start the box with a default value if you like.
- **Using check boxes.** Think of the check box as being a toggle between two values, selected and unselected. You can specify what the value will be for both states. It could be Yes and No, 1 and 0, Blue and Red, or any other combination you need to use. You can use a single check box by itself, unlike the radio button which must be used in groups.
- **Using radio buttons.** Radio buttons can only be used in groups of two or three, and enable the user to select one of the available options. Only one button can be selected at a time.

Using Configurations with Custom Properties

When you use configurations in SolidWorks, you need to be aware of the effect on custom properties. Custom properties come in two types: custom properties and configuration specific custom properties. The properties created by the Property Tab Builder can be either, but by default, they are configuration-specific properties. The difference between the two types is that the regular custom properties are common to all parts, regardless if there are multiple configurations or not. You can also use the configuration specific properties whether there is just a single configuration or many, but they enable you to have different properties for each configuration.

Once you start creating one type or the other (regular or configuration specific), the type cannot be changed, and there is no automatic feature available that will move properties between the two types. Some applications such as Workgroup PDM do not handle the two types equally. Regular custom properties work better with Workgroup PDM. You can do searches on configuration specific properties, but you cannot map SolidWorks configuration specific custom properties to Workgroup PDM properties.

With BOMs, you need to be careful which type of custom properties you are using. If you have a configured part that uses regular custom properties, the same properties will be displayed for all configurations.

Summary

Custom properties come in two types, regular custom properties and configuration specific custom properties. These types are not interchangeable. You should use regular custom properties unless you have properties that you want to be driven by configurations.

Properties can be driven or entered by a number of sources, including manual entry, properties driven by mass properties or dimensions, configurations, design tables, and the Custom Property tabs (which must be built by the administrator, and enable the user to enter property data in the Task Pane). The Custom Property tabs are a great tool to help administrators simplify the initial data entry. Administrators can set it up to allow only values from a list to keep the selection cleaner.

Properties and tags can also be used in a number of places, such as on a BOM, in notes on the drawing, FeatureManager filter, SolidWorks Search, and SolidWorks Explorer

BOMs can be used for a wide variety of disciplines; design engineering would utilize a traditional BOM with item no., qty., part number, description, and material, while manufacturing engineering may require additional items such as stock size, material code number, and so on. In addition, as previously mentioned, purchasing would appreciate the extra time necessary to procure long lead items.

Establishing Best Practice Rules

As you know by now, establishing best practice rules is a good starting place for modeling standards. This may explain in part why modeling standards are so difficult to write, because best practice rules tend to be self-contradictory and change from situation to situation.

One could argue that the entire SolidWorks software came into being at the cusp of the conflict between fast-and-loose modeling methods and robust conservative methods employed by Pro/ENGINEER. Pro/ENGINEER was always recognized as powerful software, but it required a lot of training, was not particularly easy to use, and required the user to conform to the Pro/E process. The ease-of-use concept was promoted by the less expensive, less training intensive, mid-range CAD software that came into existence in the mid-1990s. Of course, ease-of-use won that particular war. Best practice promises to bring SolidWorks back full circle, recognizing that there was indeed some value in the more methodical way of doing things. If you have ever made a model and had it fall apart the first time you made a change to it that wasn't necessarily part of the plan, you are a victim of this fast-and-loose legacy.

In this chapter, I present a higher-level discussion of several best practice topics but do not get into the details of every topic. If you require more detailed how-to information on any topic I discuss here, please refer to another resource such as the *SolidWorks 2009 Bible* (Wiley, 2009), which is a general desk reference for end-user types of issues.

IN THIS CHAPTER

Understanding best practice

Setting up best practice for sketches

Finding out best practice for features

Using best practice in assemblies

Regulating in-context best practice

Understanding the Concept of Best Practice

The term *best practice* is widely used but is perhaps a bit of a misnomer. The word “best” suggests it may be somewhat of a judgment call, and not a completely objective science. Not only that, but best practice for sheet metal may be drastically different from best practice for weldments. Further, best practice usually represents a conflict between two or more factors, and naming one “best” is only true from a particular point of view.

Defining what best practice is

The conflict is usually between speed or ease of use and building a robust model that can withstand all sorts of changes. It’s not always true that whatever you add to one side detracts from the other, but that’s how it works out most of the time. Best practice is generally taken to mean the most conservative set of options that give precedence to quality over speed.

This definition is somewhat of my own making, but I’ve been working with the best practice concept for years, and it seems most accurate. The tough part of trying to define best practice is that you can change the meaning of the term by simply changing the goal, or the definition of “best.”

For example, on a consulting visit, I was visiting with two companies that worked together. One company developed concepts quickly, and the other fleshed out the concepts into production-ready data. Both companies wanted me to help them work faster and produce data that are more detailed.

As you might expect, the situation required two different answers. For the business that did conceptual work, I steered them toward under defined sketches, sketch blocks, and even working in 2D outside of SolidWorks, or even outside of the computer. Despite all our progress, computers are not very helpful for early stage concept development for mechanisms, shapes, or larger systems that must be understood schematically. The design itself happens in your head, and you only put it down on paper or on the computer to communicate it to other people. Once you have added some definition to the project, you can start drawing the outline in the computer, making individual parts, putting meat on the concept. Best practice for people doing initial conceptual work often leads somewhere other than a CAD program.

The other business that wanted advice needed to think more organizationally. What its people were doing was much more amenable to a CAD program. The idea already existed, and they were busy creating the details and making it all happen. They understood the product as an assembly of parts that had relationships to one another, not as functions that needed to happen in a specific location.

CAD companies are fond of telling you how great their software is for “design.” I don’t believe that CAD companies have much idea of what “design” really is. You don’t do design in CAD at all, at least I never have. “Design” happens inside your head, and you capture, or document, it in CAD. With 2D CAD, you also have the burden of visualizing the 3D in your head, because your CAD can’t do it for you. 3D CAD is more process or technique based, so in that way, it is more difficult, but it removes the burden of needing to calculate all that 3D visualization in your head.

Because 3D CAD is so much more process based than 2D CAD, you have to have some guidelines to govern the process toward the best result, and that may begin with a more developed definition of what best practice is when it comes to SolidWorks.

Identifying what best practice is not

You may often hear best practice called a set of rules. When people say that, they probably mean it more metaphorically than anything. You cannot write a set of rules that can accurately govern design or modeling work. If that were possible, there is no doubt that computers would be able to do the work without the operator by now. A computer can be programmed to complete any task that you can reduce to a set of “if-then” rules. Modeling is certainly not that.

One time I tried to put together a set of rules to govern the relationships of the fillet, shell, and draft features in a typical SolidWorks part. The rules primarily covered the order of the features and the various types of geometrical decisions you had to make in order to proceed or create the desired shape. I had rules for small fillets on edges moving perpendicular to the draft direction, and then large fillets in the same direction. I had rules for which fillets or draft had to go before or after the shell feature, and then if you were building a mold or a molded part, how you would account for the plastic shrink and yet still be able to make a drawing of the part in its finished state.

In the end I was left with a convoluted tangle of rules that almost worked, but always required some sort of “except when” clause.

Best practice is also not a magic recipe by which you can get anyone who knows how to run the SolidWorks software to build the same part the same exact way every time. You need people to make decisions, and everyone makes decisions differently. Best practice can suggest general concepts, and it can rule out cheap shortcuts, but that still leaves a lot of room for individualism.

CAD is not necessarily supposed to be a means of individual expression, but because it requires human interaction and humans are not robotic machines, but driven by flexible organic matter, a task will never be 100 percent completely repeatable.

Adjusting your expectations to reality

If you write a 3D CAD standard based on best practice type of requirements, you can count on your plan being imperfect right out of the gate. Your CAD users are not going to follow you with zombie-like single-mindedness, but if your writing is simple enough, they will get the point and be able to follow the gist of what you are trying to communicate.

In fact, beyond the whole concept of writing a standard as best practice, the other most important trait your standard will need to live up to is simplicity and clarity. It has to be easy to understand and to follow. It might be an exaggeration to say that your standard needs to be easier to follow than to not follow, but an element of that may exist. It is doubtful you could write a perfect standard such that random events would fulfill it better than your effort and expertise, but oh, if you could....

The more realistic you are about your standard, your best practices, and your users, the more likely you are to write an effective standard that combats enemies such as lack of training and user apathy.

Also, be aware that you may run into some resistance about a 3D CAD standard in the first place. Users understand a 2D standard, because those have been around as long as there have been two people making drawings from which to make things, but 3D CAD standards are not so well established. Creating a standard may mean to some people that you are trying to take away their value. By standardizing modeling practice, you really are trying to get people to do things the same way, and the most proficient users are going to resist that the most. These are also the users you want to alienate the least, because they will be the most productive and most help with other users. You may want to include them in discussions about writing the standard and, especially, about the need for a standard in the first place.

Establishing Best Practice for Sketching

The SolidWorks sketcher is one thing that you almost cannot avoid using when using the software. The sketcher has its strengths and its weaknesses, and everyone uses it differently, but you can definitely identify some practices that generally cause trouble and some that generally help you avoid trouble. Creating a set of best practices around sketching can help your company's data be more consistent and editable by everyone throughout your design and modeling organization.

Some modeling practice may need to ban the use of advanced sketch elements like splines, ellipses, or parabolas mainly out of concerns for manufacturing those types of shapes. Other industries may encourage the use of splines to help ensure curvature continuity on shapely products.

Getting people to avoid the more complex functionality is usually easy, but you may find some situations where they're needed, and some people who are simply drawn to the more complex solutions. With the former, have a second look. It could be that something more complex really is the best solution. If you can learn a little bit and benefit the project a lot, the extra effort is certainly worth it. On the other hand, if you are just drawn to complex solutions out of curiosity or boredom with drawing lines and arcs, you may want to consider motivating yourself in some other way. Always stick to the best solution for the situation, don't try to invent uses for cool new stuff that you want to have an excuse to play with.

Handling sketch relations

Sketch relations are one of the most ubiquitous yet finicky aspects of the SolidWorks software. For any valid sketch, dozens of valid combinations of sketch elements and relations might go together to produce a valid result. Again, like the example of the fillet, draft, shell from earlier in this chapter, I would like to suggest a hierarchy of relations or techniques; users choose some whenever they're available and others only when they are the only option.

In sketches, it's easy to get too lazy to do things correctly. For example, look at the rectangle tool. The rectangle makes things so easy in most situations, but in a situation where a rectangle needs to be rotated at an angle, there is much reworking of the sketch and sketch relations to do. You might

consider it best practice always to make the 3-point rectangle instead of the 2-point rectangle. The 3-point rectangle uses parallel and perpendicular sketch relations instead of horizontal and vertical ones. It is only one more click to make, but it eliminates a lot of frustration in those rare circumstances that you have to rotate the sketch somewhat.

The difference between the 2-point and 3-point rectangles is representative of the rest of the best practice argument. Sometimes you really do just need to get it down now, and the cheap and easy way turns out to be the most effort or time effective. However, in other situations, when you need to be more methodical and precise, you can't afford the cheap and easy way. You will certainly come back later and pay full price for the shortcuts you have taken.

Choosing between brevity and longevity

I know many people who believe that everything should be easy all the time, and there should never be a penalty for taking a shortcut. While I acknowledge that in certain types of modeling, that attitude will serve you well, overall it is more likely to get you into trouble. Your task is to understand that there is a time for shortcuts and a time for detail, and to be able to tell the difference between them the first time, without needing to go back and rework something for longevity that was designed for brevity.

My philosophy here is that for many types of work, you should develop two completely separate sets of models. One set is fast for conceptual work. You get it down as quickly as you can, and maybe you have to use your imagination to fill in the details a little bit. This is the *concept model*, and it might be full of “hack and whack” type modeling, but it conveys the essential idea, and it gets it down fast.

The other set of models is your *production model*. This one has to be something of pride. It has to be accurate, precise, well behaved, and something you could submit to scrutiny. It's true it takes time to develop two sets of models, but if you develop your concept model trying to follow the rules for your production model, you will be late with your concept and maybe even forget what you set out to do before you complete it. Similarly, if you set out to do your production models with a concept model mentality, you are sure to turn in sloppy work and need to redo it multiple times. It also may not hold up to the rigors of changes that production models have to go through.

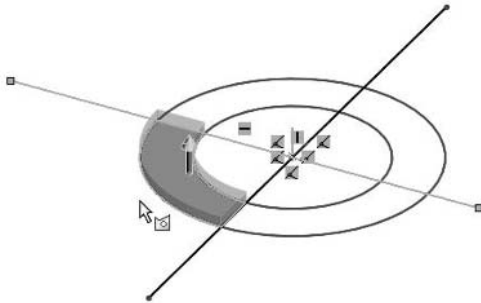
So where does this leave you with sketching? It depends. How early is the conceptual stage? The earlier it is, the less careful you need to be. What type of modeling are you doing? The more prismatic or analytical (simple shapes like lines and arcs), the less you need to worry.

Using sketch Contours

One sure sign of concept modeling in sketching is the use of contours. Contours occur where you have several overlapping and crossing sketch elements, and together they give you several options for enclosed areas to select for an extrude or revolve. Contours are fast, but they are also sloppy and not well controlled. Figure 14.1 shows a sketch where contours are used to create an extrudable area.

FIGURE 14.1

Contours in a sketch



Using under defined sketches

Under defined sketches are a topic I go back and forth on for different types of modeling. In general, your production model sketches should be fully defined where possible. By “where possible,” I’m leaving open the loophole of splines. It is typically not practical to define splines fully. Besides, with all the interior spline points, you also have the handles direction and magnitude, some of which is difficult or impossible to define truly with dimensions.

For this reason, some SolidWorks users insist on not using splines at all, even in modeling work that really requires their use. This is an example of a time when you need to be flexible with your standards to get what you technically need. I doubt you can articulate a real reason for actually needing to define fully a spline other than the dogma you hear very frequently reiterated by every smart person you know that all sketches should be fully defined.

I have heard people say that if a sketch is under defined, it might move or change shape without warning. I’ve done a lot of modeling, and a lot of it using under defined sketches. I have never found this statement to be true, and I chalk it up to an urban myth.

Most sketching, however, does not make use of splines and does not have a real pressing need for under defined sketches.

So is there really a reason for this best practice rule? Pro/ENGINEER forced it on you, and so can SolidWorks. Is it just another bit of outdated modeling dogma? I believe it is, but I also believe good reasons do exist for sticking to it. The main reason is reproducibility. Let’s say that you have a part in SolidWorks 2010, and you want to send it to someone with SolidWorks 2009. If all you need to do is re-create a couple of sketches to re-create the part, then you can do that from a screen capture of the dimensions and sketch relations shown. If all you have is lines in a sketch, you may need a DXF (Data eXchange Format) transfer to convey the geometry.

Under defined sketches seem so uncontrolled. If you accidentally move one, you will never get it back where it came from. On the other hand, there is actually an option that enables you to move fully defined sketches as if they had no dimensions on them. You find this option by choosing Tools ⇨ Sketch Settings ⇨ Override Dimensions on Drag. This might scare you a little bit, dragging fully defined sketch elements and watching your dimensions update, but it can be a useful way of managing some of the spline difficulties. Say you just used “Fully Define Sketch” to define completely all the points on a spline with dimensions, something you probably wouldn’t do by manually dimensioning all those points. One of the arguments against this is that with all those points dimensioned, you cannot freely edit the spline. However, the beautiful thing is that with the Override Dimensions on Drag option enabled, you can easily edit the spline freely, even with the dimensions attached.

Choosing between features and sketches

Users often are caught up in the idea that modeling fewer features is a best practice. One of the biggest dangers in this is that the best practice can be taken out of context and misapplied. This happens very frequently when users are concerned about performance, and they realize that fewer features may also mean faster rebuilds. If you can put a whole lot of features into a single sketch, you can save a lot of time, right?

Maybe, but probably not. Here is the problem. When you put features like fillets and holes into sketches and then do something like extrude or revolve the sketch, you do save yourself a few features. However, you also give yourself less control over your model. One of the advantages of having separate features in the FeatureManager is so you can delete, suppress, troubleshoot, or edit the features individually. For example, if you wanted to make a simplified version of a part, one of the ways you might do that would be to remove the cosmetic fillets. If the cosmetic fillets are drawn in sketches, you cannot just turn them off and then turn them back on later.

Sketches can also be a problem for recovering AutoCAD users. AutoCAD users may equate SolidWorks sketches with drawing lines in AutoCAD. The best thing to do is to start thinking in 3D rather than continuing to lean on that 2D crutch. For an AutoCAD user, 2D may be the most intuitive approach, but it is not necessarily the best way to do things in SolidWorks. Fillets, when possible, should be created as separate features rather than as sketched entities. The same goes for holes and chamfers.

If nothing else has convinced you that fillets need to be separate features, try making a sketch with a lot of sketch fillets or even simple tangent arcs in it. Make one where the lines aren’t just horizontal and vertical, and the fillets are applied as you sketch, not after everything is in place. Tangent arcs in under defined sketches are nearly unmanageable in SolidWorks.

It’s best to use fillet, chamfer, and hole features rather than depend on sketches.

Something else you need to consider is the rebuild speed. Many users assume that sketches rebuild faster than features, but in the end, it just isn’t true. In the *SolidWorks 2009 Bible* (Wiley, 2009), I did the experiment where I made a sketch pattern and a feature pattern, and the rebuild time for the sketch pattern is much higher than the feature pattern.

Using layout sketches

Because SolidWorks recently co-opted the Layout Sketch terminology, I need to be careful about how I use this term. I am not talking about the formal feature named Layout Sketch (necessarily); I'm talking about the generic technique of using an initial sketch to drive features throughout the part. Layout sketches, in both the generic and formal types, are very useful in both conceptual and production modeling. Laying out an entire assembly, a small product with single or multiple sketches, or a 2D or 3D sketch can be a very powerful technique that you may need to use for your products.

Layout sketch

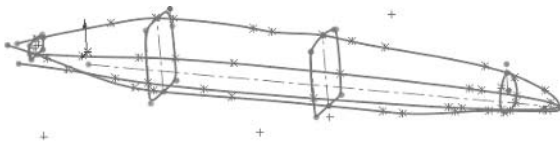
You can use a layout sketch several ways. You can mate parts to a sketch in an assembly, or use Convert Entities to copy sketch elements in a part. You can use several layout sketches on multiple planes to locate major features of a part, and then use sketch relations to make feature sketches driven from the layout sketch. Layout sketches can also be a single shared sketch, of which several features use a few elements.

The main point of a layout sketch is to have a central location from which to drive the rest of the model or assembly. This technique takes a little planning, but it works well, and especially for assemblies, gives a central location to control data.

Figure 14.2 shows a layout sketch being used to plan a model of an F1 car.

FIGURE 14.2

Planning an F1 car body with a layout sketch



Understanding Horizontal Modeling

A “best practice” modeling technique exists called *Horizontal Modeling*. The technique is actually patented by Delphi to encompass a range of techniques that you can employ on a wide range of products from a single part to an entire complex project such as an aircraft.

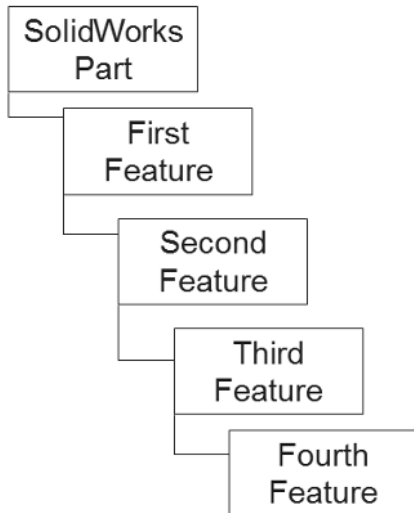
You can use Horizontal Modeling with the layout sketch technique to minimize the types of references that have a tendency to fail when making changes to the model.

Horizontal modeling was named for a diagram of parent/child relations within a part. If you look at the part in Figure 14.3, notice that the traditional SolidWorks model will completely fail if something happens to the first feature.

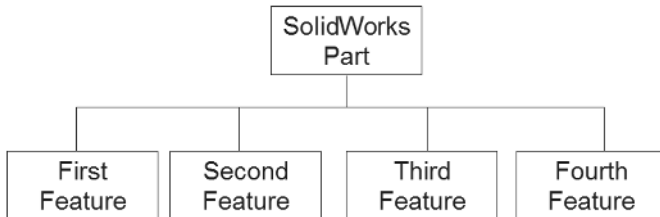
FIGURE 14.3

The Horizontal Model parent/child versus the traditional parametric model

Traditional SolidWorks Modeling



Horizontal Modeling



If you use the Horizontal Modeling scheme, however, and the first feature fails for some reason, the rest of the model keeps on going. The troubleshooting benefits of this technique are enormous, especially with complex models where you make many changes. Of course, the cost of this type of modeling is high, because it is a different way of thinking, and you have to train yourself to think this way. Once you get used to it, however, it is a very powerful and valuable tool.

There are many interpretations of how it should be used, but my interpretation goes something like this. All references within a part should be made to the origin or the standard planes. You can also refer to layout sketches on those planes or on additional reference planes. The whole scheme is based on avoiding parent-child relationships to types of geometry that might get lost in changes

later. This means you can make relations to sketches, but not to solid geometry. You can use reference geometry such as planes, axes, and such as long as it is only created from other reference geometry or stable sketches and not dependent on solid features.

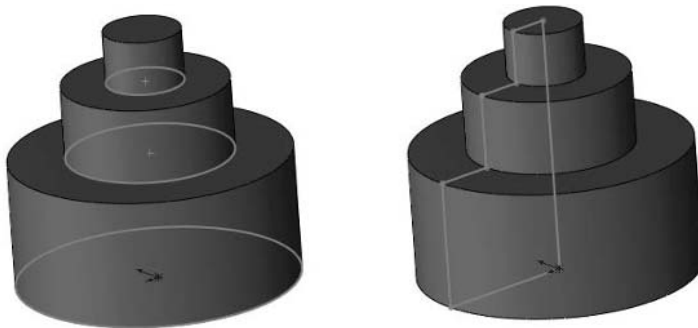
There is no question that taken to its extreme, Horizontal Modeling seems to be intended to hamstring the power of parametric modeling, but what it really does is make the parametric bomb-proof. This is why it is considered a best practice technique. It is extremely conservative, and few users adhere to it 100 percent of the time, but it also enables you to make changes without causing as many failures as what we have come to know as traditional modeling practice in SolidWorks.

Determining Best Practice for Features

When you are selecting which features to use in a part, it is best to be conservative. Use the simplest feature that will unquestionably do what you need it to do. Should you use a stack of extrudes or a single revolve with a stepped sketch? I recommend a revolve. All the data is in one place, all the dimensions are controlled by a single sketch, and it is simpler. Figure 14.4 shows the difference between a revolve and a series of extrudes for a “wedding cake” type of turned part.

FIGURE 14.4

A wedding cake part made from extrudes and a revolve



Selecting feature types wisely

Earlier in this chapter, I discussed when it was appropriate to use features instead of sketches. In this section, I compare various types of features that can be used in similar ways. If users are learning the software on their own, and using it in a vacuum, it can be difficult to tell which feature to

use when two features do the same thing. In this section, I try to shed some light on the differences between the most commonly conflicting features and put some urban myths to rest.

Using Dome versus Shape versus Freeform

The Shape feature has been removed by SolidWorks from the 2010 release. The feature was not used frequently or with much success by users, and SolidWorks Corporation felt that instead of maintaining or upgrading the feature, they should just remove it. The useful parts of Shape functionality were largely duplicated by Dome and Freeform, anyway, both of which are more controllable. If you see a Shape feature in the FeatureManager of SolidWorks 2009, it will display with a warning icon. In 2010, after a rebuild, all Shape features will fail and display the message shown in Figure 14.5.

The icons for the Dome, Shape, and Freeform tools in SolidWorks look very similar. If the icons look similar, and they can all achieve similar geometrical results, it's easy to see why users can get confused about which one to use.

If you are trying to make a bit of a bump on the face of a part, say a button, or trying to create a supportive arch on a flat face, the short answer here is to use the Dome feature. Dome will give you cleaner results and is far easier to control than the others. Figure 14.5 shows some situations where the Dome feature really excels, as well as the error message in SolidWorks 2010 regarding the discontinued support of the Shape feature.

FIGURE 14.5

Using the Dome feature



Selecting Rib feature versus Thin feature

For me this is not even a choice. I know many people get frustrated with the Rib feature, but I think it is one of the best features in SolidWorks. I would always choose a Rib over a Thin Feature Extrude unless the feature needed to work like a standing plan view rib, but did not extend all the way to the wall. I believe Thin features are underutilized, but I don't believe that most of the time they are a suitable replacement for the Rib feature.

Pattern/mirror features, faces or bodies

When deciding between patterning (or mirroring) bodies or features, it comes down to the functionality you need. If you only need copies of existing geometry, patterning bodies might be the best thing, because it is fast, and treats the body like imported geometry, without regard for feature intelligence. If you are looking for a speed improvement, patterning bodies is definitely the thing to do. If you need to pattern the intelligence of the end condition of a feature (for example, extrude up to surface), then patterning features is necessary, and you must pattern without the “geometry pattern” option.

Also, remember that the assumption that the geometry pattern speeds up patterns is a myth. It is primarily a distraction used by tech support to divert users’ attention while they are waiting for very large feature patterns to rebuild.

Patterning or mirroring faces is less commonly used, but can work on native SolidWorks models or even on imported models. It treats the selected faces like a surface body and then integrates the faces into the solid model. Next time you need to pattern faces that belong to multiple features, but you don’t need to pattern all of the faces from those features, try to pattern or mirror just the faces.

Sketch on a plane or on a face

If you have the option to sketch on a plane or on a face, always select the plane. I know most users would select the face because it is easier to pick, but your model will be more resilient to change if you sketch from reference geometry rather than directly from model geometry. It is so easy to select that face, and SolidWorks training focuses on picking the easy way when possible, but remember that if something happens to that face, it could affect the viability of the feature created from it. Planes are far more stable.

Edit or delete

You might be surprised at how many people just delete things that they would otherwise just have to change. It is usually better to edit to make a change rather than deleting and re-creating features. When you delete something, you should consider some things. First, anything dependent on what you just deleted is going to be deleted, possibly fail, or at least show an error. Then what do you do to that, edit or delete?

Getting in the habit of editing rather than deleting will teach you a lot about the software. There are a few sketch relations that cannot be reattached, and if they go dangling, you will need to delete them, such as Pierce, OnEdge, or At the Intersection. Aside from things like this, where you really don’t have any choice, you should choose to edit when you can.

Selecting a draft technique

For people who make plastic part models, this can be a difficult choice. My preferred way of doing this is to use separate draft features. That is because when you are editing a plastic part, it can be difficult to know where in the tree a draft feature came from. If they are all at the end of the tree after the fillets, it becomes much easier.

The question about sketching draft or applying draft as a feature is similar to the question about sketching fillets. Sometimes it is truly the only way to do it, but typically, it is not. If you make many plastic parts, you may develop an instinct for when to use which approach, but I use separate draft features unless there are tangency issues with the basic shape. If you have a U-shaped part, and there is no flat on the bottom of the U, you need to plan for draft right from the beginning. It may be a more complex choice than the three options I have proposed so far; you may also want to consider options such as ruled surfaces, lofted surfaces, or splines with a tangent direction at the ends. When I need to use a surface approach to drafting, I like to use the ruled surface because it has so many options. It is also somewhat finicky at times, but it gives you many options for creating drafted faces.

Deciding to use many or few features

Users sometimes become obsessed with the number of features. This could be due to a number of things, ranging from contests to see how few features you can model something in to the figures the Feature Statistics tool shows. Whether people are serious about this obsession or understand it is just one measure, when modeling, you need to focus on correct geometry and feature stability. Feature count doesn't have any direct bearing on anything. It is sometimes used as a general measure of the complexity of a model, but it shouldn't be used as a goal to maximize or minimize.

Employing Best Practice in Assemblies

To this point, I have been mainly talking about best practice in parts. For most users, parts are only a portion of the story; they must also create assemblies and drawings. Drawings are less important from a best practice point of view because the visual result is its own standard. In parts and assemblies, it is important not only what the end result looks like, but also how you got there. Because parts and assemblies are so process based, and that process is recorded, it is important how you got to the result.

Using layout sketches in assemblies

The formal feature in an assembly called the Layout Sketch is a 3D sketch that you can use to locate parts, size parts, and locate features on parts. SolidWorks took the name of a widely used technique and started using it as the name of a formal feature. I think in the end, this will wind up confusing users, especially because the formal Layout Sketch feature, as a 3D sketch, is not as easy to use as 2D sketches. If you can get past the extra burden of sketching in a 3D sketch, there are some advantages to being able to sketch on any plane at any time, but for some users, that advantage will never overcome the clumsiness of some of the still missing functionality in 3D sketches. Over the years, 3D sketches have improved significantly, but those changes have been slow, and still incomplete.

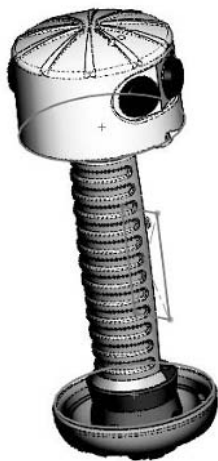
Part III: Maintaining SolidWorks

However, you achieve them, layouts in your assemblies can be extremely useful. If you prefer the informal 2D sketches to the formal 3D Layout Sketch, you just open a new assembly and start sketching on one of the standard planes. Your sketches can represent parts or places where parts mate or features on parts or subassemblies. You can mate parts to the layout or build parts in-context from the layout.

The formal Layout Sketch feature enables you to extrude parts right in the assembly directly from the layout. This enables you to make parts quickly for concept work. Figure 14.6 shows parts created in this way directly in-context from a formal 3D Layout Sketch.

FIGURE 14.6

Using a Layout Sketch to build parts in-context



Organizing assemblies

In Chapter 3, I discussed some assembly organization options, and in this chapter, I do it again from a different perspective: best practice. You can organize assemblies for manufacturing, documentation, assembly instructions, BOM, manufacturing resource planning (MRP), motion, in-context references, subassembly performance, or one of a number of other purposes. All are valid for determining the organization of an assembly, and you may be in a situation where you can't select one that is the only driving factor. You may need assemblies organized on multiple factors.

One of the things you need to keep in mind here is that you are not necessarily limited to only making a single assembly. In some situations, it may make sense to create multiple assemblies for different purposes. If you really need an assembly for assembly instructions, and it needs to be built differently from an assembly built for motion analysis, then there is no getting around it. You need to build separate assemblies. It may be inconvenient, it may be extra maintenance, it may be more work, but there also may be no other way to achieve what you need to do correctly.

Using virtual components

One thing you will need to be careful about, especially if you are using the in-context part creation mentioned in relation to the formal Layout Sketch is the virtual component functionality. Virtual components are parts that you begin to design in-context, and they are not saved to your hard drive as a separate part immediately, but instead reside inside the assembly file.

Best practice with virtual components is to avoid them. Save them to your hard drive as distinct files as soon as you can. The argument in favor of virtual components as you might expect is the old “ease of use” excuse. File management is so much easier when there are no files to manage. Part of the problem is virtual parts are still problematic, being only a release or two old at this point. It often takes SolidWorks several releases to get the kinks worked out of new functionality. On the other hand, making parts without the overhead of file management is one of those things that simply sound too good to be true.

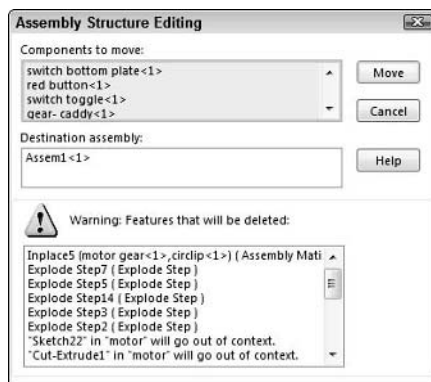
Reorganizing assemblies with in-context references

When you have an assembly or subassembly with in-context references, you need to be careful about how you reorganize components. Some types of changes will cause in-context references to become broken, and the software will usually try to warn you before doing that.

Tools for reorganization include simply dragging (or Alt+dragging) parts and subassemblies either into or out of other subassemblies using the Form New Subassembly Here or Dissolve Subassembly options from the RMB (right mouse button) menu. When you reorganize the assembly, you may see a dialog box similar to the one shown in Figure 14.7.

FIGURE 14.7

Losing some data when you reorganize the assembly



These warnings can tell of the loss of anything from mates or explode steps to in-context references and assembly features. Some of the items can be moved to the new location, such as mates, but typically, the other items are lost.

Mating with best practice

Users use mates many different ways in SolidWorks. Again, there is no single method or philosophy that rules mating, and that will give you the best results under all situations. Mates do not necessarily reflect the reality of assembled parts, but rather some mathematical ideal. For example, when mating to plastic parts with draft or cast parts with rough surface finish, or even with press or loose fit parts, you have to make some approximations in the CAD system. Your goal may be simply to document assembly steps, create a marketing rendering, show the product in use, or enable stress or motion analysis. Each of these situations requires something different.

Add to your mating concerns the fact that assembly mates can be one of the most unstable areas of the SolidWorks software, and you have a topic that is ripe for serious debate on a very wide range of best practice concerns.

Mating for position

Most of the time when SolidWorks users create mates, they do not intend the mated part to move. Even when the designed assembly is a mechanism with motion, generally mates are meant to position parts relative to other parts or other references in the assembly. When mating parts for position only, it is best if you can mate to the most stable references available, which generally means mating to reference geometry, specifically planes. If you can mate to assembly planes, that is best. Sometimes you must mate to part planes. Mating to planes is completely in keeping with the conservative best practice associated with Horizontal Modeling, again where the basic concept is that you only refer to the most stable entities available.

The most stable mate is a plane-to-plane mate, but there are times when you will need to mate between entities other than planes to ensure that the assembly updates with the part geometry.

Analyzing degrees of freedom

When mates run into trouble and become over defined, it is usually the case that the available degrees of freedom have been exceeded. Understanding degrees of freedom is necessary for efficient troubleshooting of assembly mates in all situations.

Every part in SolidWorks can translate and rotate (move by sliding and by turning) in three different directions: Translation in X, Y, and Z and rotation about X, Y, and Z. So all together, each part has six degrees of freedom.

If you mate a free plane to a fixed plane, the free plane goes from having all six degrees of freedom to losing one translational degree of freedom. (It can no longer move in the direction perpendicular to the fixed plane.) It also gives up two rotational degrees of freedom; meaning, it can only spin on an axis perpendicular to the fixed plane, not on axes parallel to the fixed plane.

Therefore, if you think about making three face-to-face mates, the way most people locate a rectangular part with respect to another rectangular part, it ties down three translational degrees of freedom and six rotational degrees of freedom. This is a problem. How can you tie down six rotational

degrees of freedom when only three exist to begin with? The answer is you can't. This is a very common problem, and users routinely over define models and just expect them to work anyway. SolidWorks has been programmed to allow for some duplication of degrees of freedom.

In rectangular parts that remain stationary, this over definition isn't normally a problem, but as an assembly becomes more complex, and the mates may be between many different parts, the story can get complicated quickly. If you have a mechanism with motion and possibly a loop of interacting parts, and some of these parts are technically over defined with respect to one another, you can expect some quirky things happening with motion due to either dragging the parts or using the SolidWorks Motion Manager to make an animation.

The best practice is to be as efficient with the mates as possible. This means that you have to keep track of the degrees of freedom that various mate types constrain. If you are working with two blocks, you have already established that face-to-face mates tie down one translation and two rotations, and that three face-to-face mates actually over define the situation. If the first mate is face to face, the second mate should be line to face (one translational and one rotational for a total of two trans and three rot) and then the final mate should be a point to face mate, which only ties down a single translational degree of freedom. This is something you need to try for yourself. Put two blocks in an assembly, fix one of them and mate the second one, keeping close tally on the degrees of freedom. Once you understand that the standard operating procedure most SolidWorks users follow for mating something as simple as two blocks actually over defines the system by 33 percent, it may change your perspective on mate problems, troubleshooting, and what you tell your users when they bring complicated mating problems to you to solve.

Is this kind of scrutiny practical all the time? Probably not, but as I've said before, 3D CAD standards are best practice suggestions, and best practice is generally conservative, maybe even radically conservative. Possibly, you will want to save this degree of freedom analysis for troubleshooting rather than everyday modeling.

Mating for motion

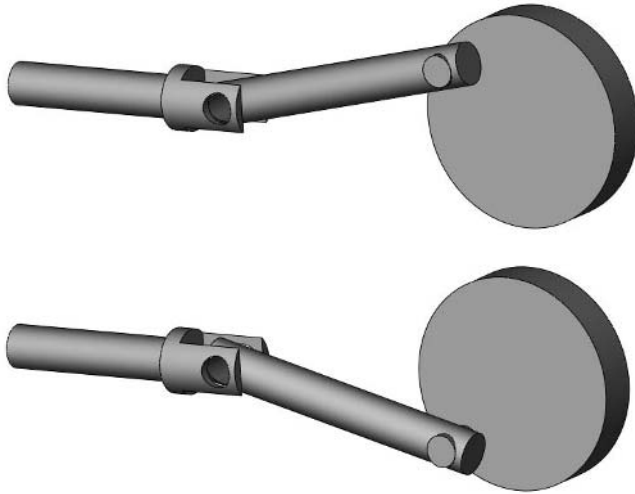
When mating parts in an assembly with the goal of using dynamic assembly motion (dragging parts with the cursor) or some form of animation or motion analysis, it is best practice if you can leave only a single degree of freedom undefined in the assembly. This ensures there is only a single path that the mechanism can take, and significantly cuts down on the possible errors.

Another factor to keep your eye on is the situation in which some parts move linearly and some move circularly. For example, think of the drive linkage on a locomotive. The piston moves back and forth, but the link on one end goes around. This means that for a given position of the piston, there are two possible locations for the link and wheel. This is illustrated in Figure 14.8.

What this means to you is that if you are manually dragging the parts to produce the motion, and you drag the piston, how does SolidWorks know which of the two possible positions the link and wheel should be in? You may actually see the link and wheel jump or jerk between the two positions as you drag the piston.

FIGURE 14.8

A locomotive type linkage in two positions



If you drag the wheel, however, the motion should be more predictable, because you are defining the position of the underdetermined part of the system. How do you write something like this into a CAD specification? I'm not sure you can, but this is certainly something you and your users need to be aware of when troubleshooting mated mechanisms. In addition, the ability to visualize the degrees of freedom will be crucial to modeling more stable assemblies in SolidWorks the next time you set out to do it.

Further, when mating for motion, do not forget that several purpose-specific mates exist. Mates like the Screw, Linear Coupler, Universal Joint, Gear, Cam, Hinge, Rack and Pinion mates are meant to make your mechanisms work more easily. Remember that the Advanced and Mechanical mates are there and make life easier. However, some of these mates may have reliability problems, such as the Path and Cam.

General mating standards suggestions

Users need to know several things before they start putting together assemblies. Unfortunately, this information often comes too late, after they have done several projects, experimented a little, developed some bad habits, and made a few assumptions. Each new license of SolidWorks should come with a standard book called *Stuff You Should Know*, but unfortunately, it doesn't. Software vendors are very eager to talk about the upside of their software, and somewhat reluctant to talk about the unpleasant but necessary information people need in order to stay out of trouble.

Following are some suggestions to help keep your users out of trouble right from the start:

- Mate to standard assembly planes when possible.
- Mate to assembly layout sketches next.
- Mate to part reference geometry next.
- Mate to part layout sketch next.
- Mate to part faces or edges last.
- Mate to a base part in the assembly (1-2, 1-3, 1-4) rather than daisy-chaining mates (1-2, 2-3, 3-4).
- Use mate references when possible.
- Avoid mating to parts within subassemblies; preference should be given to top-level parts and reference geometry, or top subassembly-level reference geometry.
- When several parts are concentric, try to mate all the parts to the same face rather than some to an inner cylindrical face and some to an outer cylindrical face.
- Do not mate to a feature that was created in-context.
- Do not mate to assembly features (cuts, plane, axis) that in turn reference part geometry (faces, edges).
- Do not mate to instances of component patterns.
- Do not mate design parts to library parts to locate the design part.

Standardizing Best Practice for In-Context Modeling

In-context modeling is probably one of the areas of SolidWorks that is most plagued by myths. There is a lot of fear surrounding this technique, and your users need to have a good understanding of what it is and what it does so that they can get the most out of the functionality while avoiding the dangers that while usually overstated, certainly are real. The problem is that most users are afraid of the wrong things, and wind up in trouble for something they weren't aware of, while avoiding perfectly good functionality.

Understanding how in-context and external references work

In-context references and external references are always denoted by the symbol \rightarrow . This simply indicates that the part requires data from another file to be complete. In the case of external references such as mirrored parts or inserted parts (base or derived parts), there is only one file referenced. One part file simply references another part file directly.

In-context references are somewhat more complex, but still manageable. The difference between a simple external reference and an in-context reference is that while the external reference references

Part III: Maintaining SolidWorks

another part file directly, the in-context reference references another part in the context of an assembly. Therefore, the in-context part has to take into account the referenced part's position as it is mated with respect to the referencing part and all other parts in the assembly.

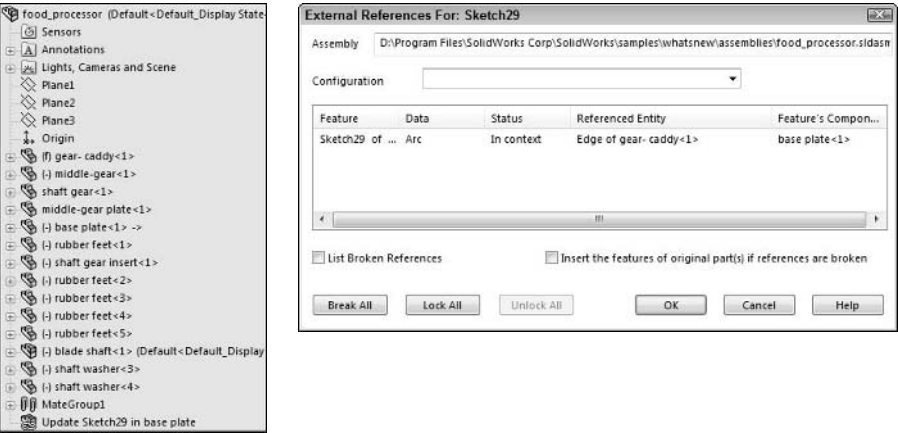
To resolve an external reference fully, you just have to open the other part, but to resolve an in-context reference, you have to have the assembly and both parts. In an in-context situation, the parent part has no record of the relationship. The child part has the → marker that shows the feature that contains the in-context relation, which is usually a sketch relation between parts in an assembly, but can also be a feature end condition like Up To Surface. The assembly also has a record of the relation, so that it knows when it is time to update which part, and which feature has to be updated in the assembly.

The marker that the assembly keeps to remember this relationship is called the Update Holder. By default, these markers have been hidden in SolidWorks for many releases, but they used to be visible at the bottom of the tree for all to see. It was easy to see if you were making many in-context relations in an assembly because you would have a long list of the yellow and green update holders at the bottom of the assembly tree. SolidWorks stopped showing this for a couple of reasons. First, they caused users to ask too many questions. Second, there really wasn't much that you could do with them except view the contents, which only told you which feature referenced another part in a particular assembly. You couldn't delete it, you couldn't edit it, and you couldn't move it, so SolidWorks hid them.

You can still see them if you want to. If you right-click on the top-level assembly name in the FeatureManager, you can select Show Update Holders. Figure 14.9 shows the update holder at the bottom of the tree and the List External References dialog box associated with that update holder.

FIGURE 14.9

In-context update holder and associated External References box



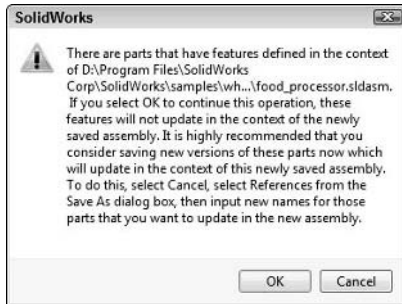
Notice that the base plate part has the in-context symbol \rightarrow . (Also, notice that this assembly does not follow any of the file naming conventions I've laid out in this book. This is an assembly taken from the SolidWorks example files.)

Using file management techniques with in-context features

If you were to try to rename the part or the assembly, that rename would have some effect on the in-context relationship. If you were to try to save the assembly in order to rename it, what would happen is shown in Figure 14.10.

FIGURE 14.10

The warning that appears when you change the name of an assembly with in-context relations



People often breeze through this kind of message even though it is very important to understand what it says and to follow its instructions. To summarize it for you, if you go through with the Save As, you will break all your in-context relations, and they will become out of context. The symbol for the out of context relation is $\rightarrow?$. Appropriate enough.

If what you really mean to do is simply rename the assembly and maintain all the in-context relations, then you need to follow the instructions, which sound like complete overkill, but are correct. In order to change the name of the assembly and maintain all in-context relations, you have to make a copy of the assembly and all the parts (at least all the in-context parts). This is what the References button enables you to do. This is a bit complicated. If this part of the software were smoothed out a little bit, in-context relations and file management in general would have less of a reputation for being so dangerous.

To complete the task properly, from the Save As dialog box, click the References button, assign new filenames for the assembly and at least the in-context parts, and possibly assign a new path for the new parts. SolidWorks then copies and renames the assembly and all the parts, re-creating the in-context relations in the new copies.

Part III: Maintaining SolidWorks

Another way to manage files effectively with in-context references is to copy the entire assembly with all its parts to a new location using Windows Explorer. You can't rename any of the files this way, but if you keep all the files together (assuming they were together to begin with), it will work.

SolidWorks Explorer is another way to copy, move, or rename assemblies without losing references. All your users should be familiar with this tool, even if you are also using a PDM product, just to enable local file management tasks outside of PDM.

Summary

Best practice rules are difficult to define, since every company has a different way of working, and different things they require SolidWorks to do. Best practice rules are generally conservative and usually sacrifice time-saving shortcuts. The goal of best practice rules is usually to make models that can withstand a wide range of changes without features failing, but can also be geared toward editability or rebuild speed.

The very best practice rules can be thought of as suggestions, but your 3D SolidWorks standard should spell out how rigid the rules need to be. The more experienced the user, the more likely the user can bend or even break best practice rules and get away with it.

Automating Tasks with the API

The SolidWorks API (application programming interface) is a simple yet powerful tool for automating almost any type of task in SolidWorks. Using macros can enhance your productivity, enabling you to do things not offered directly by SolidWorks software. Being able to create your own macros makes you worth more at your place of work, and you can use them to do almost anything that you can imagine. You can start by recording very basic macros, and develop your skills into more involved code writing with error checking and user input. You do not have to be much of a programmer to follow what I show here. I am not much of a programmer, though a basic understanding of Visual Basic is useful.

Visual Basic is a common programming language. VBA (Visual Basic for Applications) is included in SolidWorks in the same way that it is included in Word and Excel. If you have programming skills with these other common applications, those skills are largely transferable to working with SolidWorks macros.

Application Programming Interface (API) refers to the available SolidWorks functions that can be called programmatically (numbered in the hundreds), from VB, VBA, VB.NET, C++, C# (C sharp), or SolidWorks macro files that have the extension of *.swp or *.swb.

Note

The *.swb file type is a legacy macro. These files can be edited and saved into the newer *.swp format, or can be run as-is. Also note that *.swp is the extension used by the Windows Swap file. ■

IN THIS CHAPTER

Recording your actions as macros

Building a macro with a user form

Using Toolbox to build forms

Getting macro help

This chapter is a quick overview of how to create simple macros and how to hook them up to hot-keys and custom toolbar buttons, and it goes through an intermediate level macro to show some of the capabilities that are available to combine with Excel and SolidWorks at the same time.

This chapter is *not* a comprehensive “how-to” for API programming. That would be a book unto itself. For useful sources for help with macros, refer to the list of help sources end of this chapter.

Recording Macros

Recording your actions as you do various tasks in SolidWorks can be an effective way to begin writing useful macros, even if you have never done any real programming before. The macro recorder in SolidWorks does not record everything exactly for all types of tasks, but it does get you started, and in some cases your recorded actions can be useful as they are, right out of the recorder.

Everything has to start somewhere. Macros start from one of three places:

- Writing SolidWorks macro code from scratch
- Copying from another existing macro
- Recording actions and replaying them

Recorded macros do not always get you exactly where you want to go, but they do usually point you in the right direction. When recording a macro, SolidWorks lists the SolidWorks API commands used to create the action being recorded. The macro recorder does not always capture all the actions, and it does not always record the latest versions of the functions, but again, it is a running start.

Recording a rectangle sketching macro

As simple as this macro is, it serves as a good place to get started, especially if you are a novice macro writer. Recording a simple action and analyzing the resulting code is a great way to get familiar with how macros are structured, and the SolidWorks macro interface.

Before really getting started, you need to become acquainted with the Macro toolbar. This is shown in Figure 15.1.

FIGURE 15.1

The Macro toolbar offers all the tools you need for working with macros in SolidWorks.



The first macro this chapter tackles is a simple one that is used frequently: sketching a rectangle and centering it on the origin. When starting to make a new macro, you have to think about where to start and where to finish. What exactly do you want to automate? Do you want the macro to create a rectangle always on the Front plane, or just on a *selected* plane? Does it always center on the origin or on a selected point?

This macro needs to do the following:

- Create a rectangle on a selected plane
- Center the rectangle on the part origin
- Add dimensions and enable the user to enter values as the macro plays

This means that the selection of the sketch plane needs to be either something that is not recorded or something that is edited out afterward. Because this section discusses recording rather than editing, it will be recorded without the reference.

To prepare for recording the macro, turn off PhotoWorks if it is turned on. Macros will record some PhotoWorks settings, which are not needed unless you are doing something PhotoWorks specific. You can turn it off by choosing Tools ⇨ Add-Ins ⇨ PhotoWorks.

Also, when recording a macro, be careful not to make a lot of extra mouse clicks or keyboard strokes. These may record unnecessary data to the macro, which makes for additional editing.

Recording the macro

When you record a macro, the time you take to pause between clicks is not recorded, nor are your mouse movements. The only thing that is recorded is the position of your cursor when you click, and whatever command or option you click.

Start your recoding like this:

1. Open a new part and open a sketch on one of the planes.
2. Click the Record button on the Macro toolbar.
3. Sketch the rectangle around the origin.
4. Sketch a construction line from either corner to corner or from midpoint to opposite midpoint.
5. Select the construction line and the origin and assign a sketch relation of Midpoint.
6. Apply a dimension to a horizontal line and accept the default dimension.
7. Apply a dimension to a vertical line and accept the default dimension.
8. Click the Stop Macro button on the Macro toolbar, and save the macro with a name such as Rectangle.swp.

Part III: Maintaining SolidWorks

The Macro Editor appears and shows the recorded macro:

```
' *****
***
' C:\DOCUME~1\MATT~1\SON\LOCALS~1\Temp\swx3696\Macro1.swb - macro
_ recorded on 12/05/06 by matt
' *****
***

Dim swApp As Object
Dim Part As Object
Dim SelMgr As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Dim Feature As Object
Sub main()
Set swApp = Application.SldWorks
Set Part = swApp.ActiveDoc
Set SelMgr = Part.SelectionManager
Part.SketchRectangle -0.04490775510204, 0.02713176870748, 0, _
    0.06081258503401, -0.02806734693878, 0, 1
Part.ClearSelection2 True
Part.CreateLine2(-0.04490775510204, -4.677891156463E-04, 0, _
    0.06081258503401, -4.677891156463E-04, 0).ConstructionGeometry
    = True
Part.SetPickMode
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("Point1@Origin", _
    "EXTSKETCHPOINT", 0, 0, 0, False, 0, Nothing, 0)
boolstatus = Part.Extension.SelectByID2("Line5", "SKETCHSEGMENT",
    _ 0.007328696145125, -6.237188208617E-04, 0, True, 0, Nothing,
    0)
Part.SketchAddConstraints "sgATMIDDLE"
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("Line1", "SKETCHSEGMENT",
    _ 0.009355782312925, 0.02759955782313, 0, False, 0, Nothing, 0)
Dim Annotation As Object
Set Annotation = Part.AddDimension2(0.0057694, 0.039918, 0)
Part.ClearSelection2 True
Part.Parameter("D1@Sketch6").SystemValue = 0.0898155
boolstatus = Part.Extension.SelectByID2("Line4", "SKETCHSEGMENT",
    _ 0.04490775510204, 0.009667641723356, 0, False, 0, Nothing, 0)
Set Annotation = Part.AddDimension2(0.0661142, 0.0057694, 0)
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("D1@Sketch6@Part2.SLDPRT",
    _ "DIMENSION", 0, 0, 0, False, 0, Nothing, 0)
Part.Parameter("D2@Sketch6").SystemValue = 0.0551991
Part.ClearSelection2 True
End Sub
```

Taking advantage of journal files

As you work with the SolidWorks software, it is recording what you do in a journal file. My journal file is located at this path (using Vista): C:\Users\matt\AppData\Roaming\SolidWorks\swxJRNL.swj. These journal files can be used in several situations. First, they can be used to troubleshoot crashes since it is essentially recording a macro of your actions right up until the time of the crash. You can open these files in Microsoft Notepad or any other text editor, and they look like a macro.

Below is a journal file created by opening a new session of SolidWorks, sketching a rectangle, extruding it, and selecting a face to put fillets on.

```
' *****
' *****
' C:\Users\matt\AppData\Roaming\SolidWorks\SolidWorks 2010\
' swxJRNL.swj - journal recorded on 09/02/09 by matt
' *****
' *****

Dim swApp As Object
Dim Part As Object
Dim boolstatus As Boolean
Dim longstatus As Long, longwarnings As Long
Sub main()
Set swApp = Application.SldWorks
Dim myModelView As Object
Set Part = swApp.NewDocument("D:\library\templates\part inch.
    prtdot", 0, 0, 0)
swApp.ActivateDoc2 "Part1", False, longstatus
Set Part = swApp.ActiveDoc
Set myModelView = Part.ActiveView
myModelView.FrameState = swWindowState_e.swWindowMaximized
Part.ClearSelection2 True
Dim vSkLines As Variant
vSkLines = Part.SketchManager.CreateCornerRectangle(0, 0, 0,
    0.04047512653261, 0.02761917629789, 0)
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("Line3", "SKETCHSEGMENT",
    0.0163708477459, 0.01358492501176, -0.02009146485403, False, 0,
    Nothing, 0)
Dim myDisplayDim As Object
Set myDisplayDim = Part.AddDimension2(0.02885687991204,
    0.0400746902111, 0)
Part.ClearSelection2 True
Dim myDimension As Object
```

Part III: Maintaining SolidWorks

```
Set myDimension = Part.Parameter("D1@Sketch1")
myDimension.SystemValue = 0.0508
boolstatus = Part.Extension.SelectByID2("Line4", "SKETCHSEGMENT",
    0.03899736579774, -0.001412014653264, -0.02184247057603, False,
    0, Nothing, 0)
Set myDisplayDim = Part.AddDimension2(0.06167057644853,
    0.0136374278116, 0)
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("D1@Sketch1@Part1.SLDPRT",
    "DIMENSION", 0, 0, 0, False, 0, Nothing, 0)
Part.ClearSelection2 True
Set myDimension = Part.Parameter("D2@Sketch1")
myDimension.SystemValue = 0.0254
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("D2@Sketch1@Part1.SLDPRT",
    "DIMENSION", 0, 0, 0, False, 0, Nothing, 0)
Dim myFeature As Object
Set myFeature = Part.FeatureManager.FeatureExtrusion2(True, False,
    False, 0, 0, 0.023622, 0.00254, False, False, False, False,
    0.01745329251994, 0.01745329251994, False, False, False, False,
    True, True, True, 0, 0, False)
Part.SelectionManager.EnableContourSelection = False
boolstatus = Part.Extension.SelectByID2("", "FACE",
    0.02373751410033, 0.01101744300581, 0.023621999999999, False, 0,
    Nothing, 0)
Part.ClearSelection2 True
boolstatus = Part.Extension.SelectByID2("", "FACE",
    0.02373751410033, 0.01101744300581, 0.023621999999999, False, 1,
    Nothing, 0)
Dim radiiArray0 As Variant
Dim radiis0() As Double
ReDim radiis0(0 To 0) As Double
Dim setBackArray0 As Variant
Dim setBacks0 As Double
Dim pointArray0 As Variant
Dim points0 As Double
radiiArray0 = radiis0
setBackArray0 = setBacks0
pointArray0 = points0
Set myFeature = Part.FeatureManager.FeatureFillet(195, 0.00254, 0,
    0, (radiiArray0), (setBackArray0), (pointArray0))
```

You may not want to use this directly as a macro, but it is another source on top of recording macros that you can use to automate portions of macro creation.

Understanding what was recorded

I need to cover a few pieces of housekeeping first, especially for those of you who are not familiar with programming or need a bit of a refresher.

A single quote in front of a line of text is a way to “comment out” a line, which means that anything after the single quote is not processed but may be used for reference, explanation, or troubleshooting.

A space followed by an underscore at the end of a line means that the line is continued to the next line. You do not see this in the recorded macro, but it is used here because the lines are too long for the page and must wrap.

At the beginning of the macro, the line with the file path between two rows of asterisks is only added to make a record of who created the macro and when that person did it. The name is taken from the user’s Windows login name. It is common practice to leave this information in the macro, and even edit it as macros are updated by various people. This is one way of giving credit to the original author when macros are shared among users.

The six lines of Dim statements are establishing the various types of variable that are often used in macros. Just because a variable has a Dim statement does not necessarily mean it is used in the macro. Some of these may be trimmed out if you want. The first two, `swApp` and `Part`, must remain, in fact, these are used for almost all SolidWorks macros. In this case, leave them all.

The line starting with `Part.SketchRectangle` is placing the two corners used to describe the rectangle. These dimensions are in *meters*. SolidWorks macros all work in meters. To use other units, you will have to work with conversion factors as appropriate. In more sophisticated macros, the document units are determined, and any user input is filtered through that setting. The corner coordinate data is given in the form `X1, Y1, Z1, X2, Y2, Z2`, with the 1 and 2 representing the first and second corners. To clean up the macro a little, you could change some of the numbers to round them out to whole numbers in the unit system of your choice, but if you do that, you will need to make other changes, such as changing the endpoint coordinates of the construction line that is sketched later at the `Part.CreateLine2` command.

Going through the rest of the macro in this way helps you understand what each of the functions do. It helps even more if you keep the SolidWorks API Help file open and look up functions as you go through, such as the `Part.ClearSelection2 True` line. Searching the Index on just the `ClearSelection2` brings you to an entry that also indicates what the True option does. You access the API Help from the main SolidWorks help menu, under SolidWorks and Add-Ins API Help Topics. This help file has a lot of detailed information about specific methods.

Replaying the macro

The first step in replaying your new macro is to see what the recording picked up. The macro window should open automatically after you save it. It is best if you have two monitors to do this, one for SolidWorks and one for the macro window. If you have to make due with a single monitor, you may want to try to tile your application windows so you can move back and forth between SolidWorks and the macro window easily.

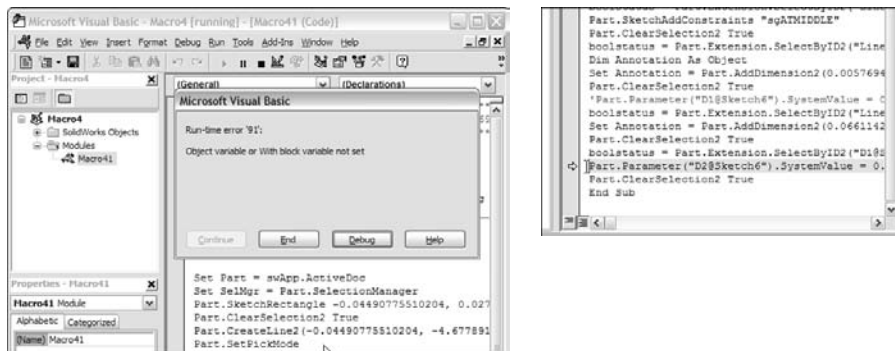
Part III: Maintaining SolidWorks

Often the recorded macro requires some sort of pre-selection in SolidWorks before it will play correctly. Exit and delete the sketch you started while recording the macro. As an experiment, do not pre-select a plane to see how the macro reacts. Return to the macro editor window, and click the Play button.

The rectangle is drawn, the construction line is drawn, the sketch relation is applied, and the first dimension is applied, but when you enter a value for the dimension, there is an error. The error is on the Part.Parameter line. In this chapter, there is not enough space to get into troubleshooting API functions, but in this case, it is safe enough to comment out the offending line. The error message enables you to debug the problem. After you click the Debug button, the editor highlights the problem line of code, and if you put a single quote in front of the line and press Play again, the same error occurs for the second dimension at an almost identical line. So comment out the second line as well. This is shown in Figure 15.2.

FIGURE 15.2

Commenting out trouble lines in the code



Clear the sketch and run the macro again. This time, with the two lines of code commented out, it works perfectly. Also, notice that if no plane is pre-selected, the code goes to the Front plane by default, but if a plane is pre-selected, the rectangle is built on it.

You may notice some irregularities, such as the sketch over defines if multiple rectangles are built in a single sketch or a new rectangle may pick up automatic relations to existing geometry.

However, with a simple recording operation and some minor tweaks, you have a simple yet functional and highly useful macro that draws a centered rectangle around the origin. As you'll see in the tutorials in this book, many start with an instruction to sketch a rectangle centered on the origin. I use this macro frequently.

Storing your macro

Out of necessity, I usually store macros in two locations. One location is in the SolidWorks installation directory in a folder called macros. It is important for this folder to exist and be spelled just

that way so that certain functions with the Tools⇨Customize dialog box will work properly, including assigning hotkeys and menu picks to the macros.

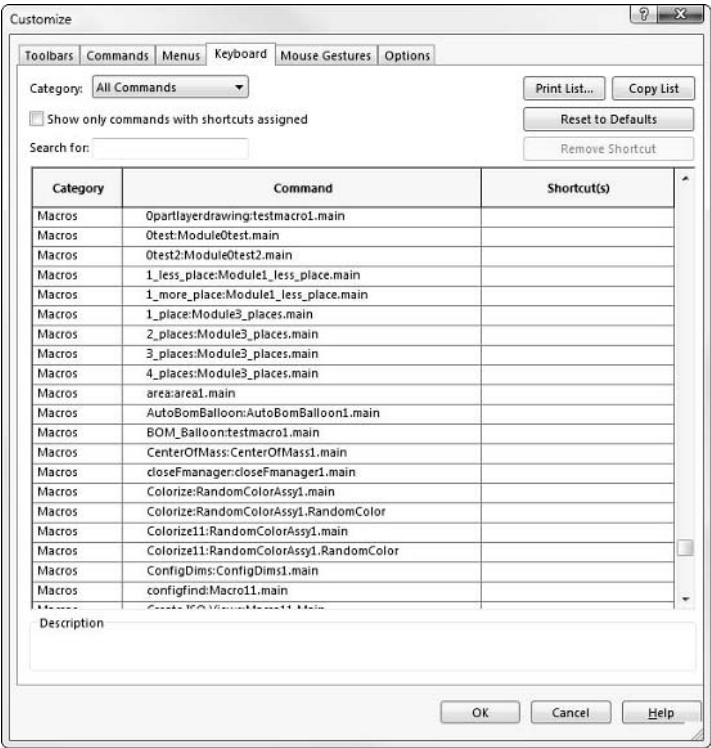
Of course, the other location is somewhere that is not in the SolidWorks installation directory so that when you uninstall the software, you do not lose all your macros. When you do this, remember to set the Tools⇨Options⇨File Locations⇨Macros setting. Telling SolidWorks where the macros are only affects where SolidWorks looks when you click the Play button on the Macro toolbar; it does not affect the need for the macros folder in the installation directory required by Tools⇨Customize mentioned previously.

Connecting to a hotkey

You connect a macro to a hotkey by choosing Tools⇨Customize⇨Keyboard interface. This interface has been greatly improved in SolidWorks 2007. This requires the macros folder mentioned previously. The software may have to be restarted if the macros folder has just been created, or if new macros have been added in the current session. Scrolling to the bottom of the list shows the macros in the folder. This is shown in Figure 15.3.

FIGURE 15.3

Assigning hotkeys to macros through the Customize interface



Part III: Maintaining SolidWorks

Notice that some of the macros listed already have shortcuts (hotkeys) assigned. Hotkey assignments are discussed in Chapter 2.

Connecting to a custom toolbar button

Connecting a macro to a custom toolbar button is far easier than it sounds. You can place custom toolbar buttons onto your toolbar by choosing Tools⇨Customize⇨Commands⇨Macro. When you drop it onto a toolbar, the Customize Macro Button dialog box appears, as shown in Figure 15.4.

FIGURE 15.4

Customizing a toolbar button for a macro



Enter a tooltip and a prompt. You can select a custom image for the button. The custom image must be 16 pixels square, in a bitmap format using 256 colors. Next to the Macro box, click the ... button to browse to the macro location. You can even assign a shortcut (hotkey) here, although I recommend doing either a custom button or a hotkey, but not both. In the end, you wind up using one technique or the other, but not both.

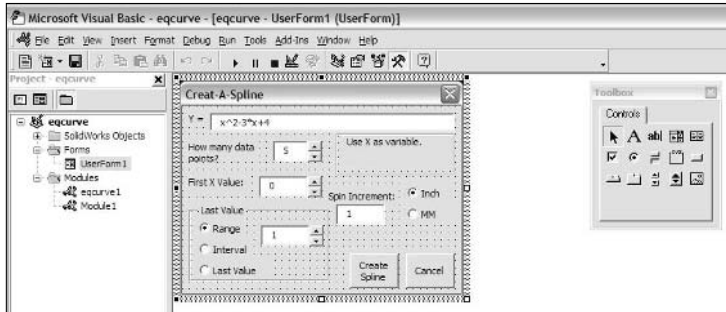
Creating a Macro with a User Form

The rectangle macro is a very simple macro that performs a set operation using standard SolidWorks input forms for the dimensional data. From that very simple macro, the macro covered in this section is an intermediate level macro, still not highly complex, but something that might take more than an hour to put together. The interface, or user form, is shown in Figure 15.5.

This macro creates a spline from an equation, using Excel to evaluate points in the equation. It enables the user to enter an equation in the form $Y = \langle \text{expression} \rangle$, where the expression can use quadratic or periodic functions. The user then specifies a first and a last value, and then tells the macro to create the spline.

FIGURE 15.5

The user form for the macro



Open this macro by clicking the Edit Macro tool on the Macro toolbar. You can find the macro in the material from the CD-ROM for Chapter 32 named Chapter 32 – User Form.swp.

Building the Form

Whenever an interface is created, specific actions are assigned to specific controls in the interface. The most obvious control that “does something” here is the Create Spline. In the macro editor, double-click the Create Spline button, and the code for the form appears.

Each control that accepts user input drives a particular variable, which is in turn used to calculate another value or to set a part of the interface. You should have enough controls to do what needs to be done, but not so many as to make the interface confusing. For example, the Spin Increment is an unnecessary control and could be removed to make the interface a little less confusing.

The form is built visually using the tools on the Toolbox toolbar. You can find the properties for each control in the Properties Window, which you can turn on via the View menu. The Properties Window and Toolbox toolbar are shown in Figure 15.6.

After the user fills out all the boxes and clicks the Create Spline button, the subroutine called `CommandButton1_Click` takes over. You can see this code by double-clicking the Create Spline button in Design Mode.

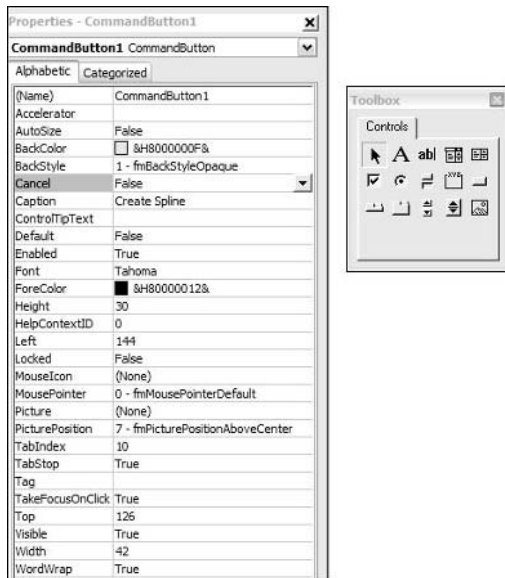
The macro starts by checking to see if there is an active SolidWorks document, as follows:

```
Set swApp = CreateObject("SldWorks.Application")
Set Model = swApp.ActiveDoc
If Model Is Nothing Then                                ' If no active doc found
    swApp.SendMsgToUser "No active doc found!"          ' Warning to the user
    Exit Sub                                             ' Exit this app
End If
```

Part III: Maintaining SolidWorks

FIGURE 15.6

The Properties Window and Toolbox toolbar



Then based on user input, the last X value of the spline is either calculated or used directly. The first X is entered explicitly by the user.

```
first = firstval.Value
last = rangeInt.Value
If rangeInt1.Value = True Then last = first + rangeInt.Value
If rangeInt2.Value = True Then last = first + (howMany - 1) *
    rangeInt.Value
```

Then the step interval is calculated based on the range and number of data points.

```
step = Abs((last - first) / (howMany - 1))
```

In macros, you'll find yourself checking for situations that you might normally assume correct, making the macro make a decision based on the results of the check. For example, this macro checks for an open sketch, and if it does not find one, it creates one.

```
Set activeSketch = Model.GetActiveSketch
If (activeSketch Is Nothing) Then ' If a Sketch is not active, then
    Model.InsertSketch            ' insert a new sketch for our spline
End If
```

The real functional part of the macro comes with the next few lines:

```
For row = 1 To howMany
'Rebuild user input equation
strlen = Len(equation.Value) 'length of equation string
neweq = "=" & equation.Value 'put an "=" in front of the eq
repl = "A" & row
neweq = Replace(neweq, "X", repl, 1) 'step through string and find "X"
neweq = Replace(neweq, "x", repl, 1) 'replace "X" with "a(row)"
ws.Cells(row, col).Value = xvalue 'write values to excel
ws.Cells(row, col + 1).Value = neweq
yvalue = ws.Cells(row, col + 1).Value 'read Y value from excel
    Model.SketchSpline (howMany - row), xvalue * unitconv, yvalue *
        unitconv, 0# ' Draw the sketch thru it
xvalue = xvalue + step ' read X value from excel
Next row
```

This writes an equation in each cell with the appropriate X value, then draws a spline point using the matching X and Y values. Then it steps to the next row and does it again until all the values have been calculated and all the points drawn. This has the bonus of being able to leave you with an Excel spreadsheet with the values in it.

Finding Macro Help

SolidWorks has some excellent resources with a lot of sample code for many applications. If you are starting from scratch with programming, it is probably best to start out by taking a class in Visual Basic or getting a book on the topic. This gets you up to speed for programming in general.

Beyond Visual Basic programming, you need to know some things specific to programming for SolidWorks, which is probably best handled by an API class from your local reseller. This class is not as popular as other classes, so you may need to wait a bit until the class is offered or you may be able to find the class available from another reseller. This class can be very useful when it is taught by someone who knows the material well. I advise checking into the instructor's credentials before committing resources.

Note

Sometimes you can find the book used for this class available in places like eBay. SolidWorks only sells these books through resellers and only for the purpose of the official class, so the books can be difficult to obtain. ■

SolidWorks has a Web site with a lot of great information that is dedicated to API help: www.solidworks.com/pages/services/APISupport.html. If you know what you are looking for, this is an excellent resource.

Also, do not forget that the Web is a great source of free information. A simple search will reveal many personal sites of private individuals who share or trade SolidWorks macros. In addition to searches through the major search engines, check the major forums including comp.cad.solid-works, eng-tips, and the SolidWorks Discussion Forums.

Summary

The SolidWorks API is extremely powerful, enabling you to create macros from very simple yet useful to complex and indispensable. Macro programming is not a casual pursuit, however. Some basic programming knowledge of some type is a definite prerequisite. This is not to say you need to be an expert; many macros are put together by “cut and paste” programming, pulling portions of useful macros from many different sources.

Supporting SolidWorks Users

Supporting SolidWorks users can be a challenge if you have never done it before. If you have functioned in a technical support role previously, you have an idea of the kinds of things you can expect. If you were a SolidWorks reseller support engineer in a previous job, it will be the easiest to transition. It takes a long time to learn the tools of the trade in support, and it takes a lot of effort to stay on top of the software from release to release.

This chapter helps you on two levels. No matter whom you are, you are going to need help from time to time when supporting your users, so the first part of this chapter addresses sources for help. The second part of the chapter covers how you provide support to your users.

Because the SolidWorks software grows more and more complex every year, it is impossible to be an expert on all aspects of it. No one person can be expected to know everything about the software, especially if your company uses add-ins like Simulation (Cosmos) or PDM, which are fields of study all on their own. So one of the necessary tasks of the SolidWorks Administrator, no matter how well informed she is in a specific area of the SolidWorks software, is to be familiar with resources that you can count on for professional level support.

Support is a difficult job. Generally, support providers are rather low on the pay scale, and support is only a small portion of your job as an Administrator. Face it support is the one thing that stands between productive users and a lot of money sitting around unused. This means you have to learn fast to provide effective and efficient support.

IN THIS CHAPTER

Providing support

Supporting users yourself

Finding answers

Providing Support through an External Organization

The most common way for SolidWorks customers to gain support through an external organization is to use the SolidWorks reseller. This is how SolidWorks Corp intends it, but it is not always the most cost-effective plan for companies, especially companies with a mature set of SolidWorks software users.

When you pay for SolidWorks subscription service, half of what you pay for is support from the reseller. The other half is service packs and new versions. If your company is a new SolidWorks customer, or if the SolidWorks administrator is new on the job, it might be a good idea to keep the subscription service available.

Some SolidWorks resellers may also offer support plans that do not include the SolidWorks service packs and upgrades. These are available on a case-by-case basis from resellers. If your reseller does not offer this arrangement but you are interested in it, they may be willing to negotiate a deal with you.

Reseller support technicians can vary in aptitude, with some having extensive knowledge and others trying to get up to speed while on the job. Many resellers put new hires on the support telephones because they consider it a burnout position, and pure overhead.

They tend to put talented technical people in sales situations or use them as consultants or trainers. Technical support is often viewed by resellers as an obligation that must be fulfilled rather than an opportunity to make a good impression on an existing customer. I have worked at several resellers: I have seen the rotation in and out of technical support roles, and have watched how that turnover affects customer experience and satisfaction.

SolidWorks Corp offers a training and certification program for reseller employees, which is a great thing. When I took it, the reseller support-technician exam was the most difficult SolidWorks exam I had ever taken. However, you don't have to be certified to start providing support; some resellers don't see a huge difference between having one certified support technician on staff and an entire staff of certified support technicians. Further, in larger reseller organizations, the turnover in support can be disruptive. As a technician, providing support can become tedious because customers are calling or e-mailing with problems. People with problems are often frustrated; and when technicians don't have the answer, it puts added stress on them.

Concisely, your experience with SolidWorks reseller tech support can be very good when resellers have highly experienced certified support technicians on the line, or very bad when you get a beginning user who is high on confidence and low in actual knowledge.

If you find the support you receive is no better than the support you could provide, consider talking with your reseller about your situation to see if you can get special access to advanced tech

support from a specific individual within their organization; or you may want to look around for a reseller who is better equipped to serve your needs.

Many companies using SolidWorks allow their subscriptions to lapse, remaining on a stable version of the SolidWorks software, and being self-supporting. They then wait until a significant version of the software appears with changes that are substantial enough to make a calculable impact on modeling or drawing speed. However, getting off a subscription and then back on usually incurs a penalty of some sort.

If you find you are doing work that is quite advanced and beyond the expertise of the reseller chain, you could consider retaining a consultant to help you. When resellers are faced with limitations in their knowledge of the software, they will begin referring you to additional services, for additional fees, such as training or consulting. While they may not have someone on their support staff who can answer your questions on advanced surfacing or large assemblies, they may indeed have someone in another branch of the organization that can help you. There is a point where the types of questions you ask can go beyond general tech support and branch into design consultation, and in all fairness, sometimes the line is more than a little blurry. You might get yourself into a situation where the tool you expect to work doesn't work, and you are forced into layers of work-arounds to get your technique to work. When this happens because of limitations in the software, certainly, tech support should be there to help you through. If, on the other hand, you are simply asking a very advanced how-to question, some resellers draw the line by telling you your question is beyond the scope of tech support, and a salesperson is likely to be your next point of contact.

When you have a reseller provide your support, you should keep track of the calls you have in to them, the answers they owe you, and the SPRs (software problem/performance report) that remain active. This is not the kind of information you jot on a Post-it by your desk; you need to have some sort of a simple database application that enables search.

Your reseller may offer something like this so you can track your issues. They will often use the term SPR for serious problems that they have been able to reproduce and log with SolidWorks, or SR for "service request," which is essentially an open call without an answer. The SR may turn into an SPR, or may simply be closed.

One thing to watch out for is that when SolidWorks or the reseller assigns an SPR, they may also close the call, which makes it appear that the issue is done and over with. An SPR is really just the beginning. It is when SolidWorks formally acknowledges that there is a problem, which needs further attention.

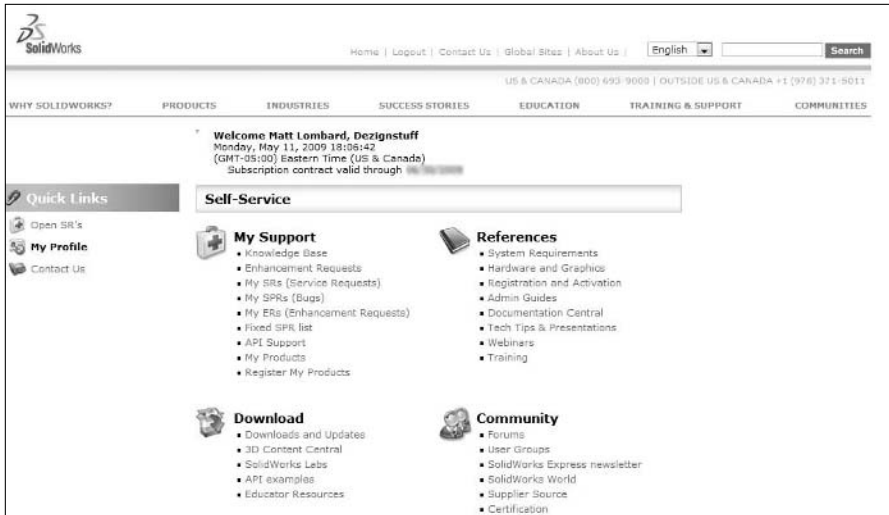
The resellers are able to access the SolidWorks support database, as can you to some extent through the Customer Portal. You may hear the reseller or SolidWorks support refer to Siebel, which is the system SolidWorks uses to track support issues. This is the system you can access through the Customer Portal using the link <https://customerportal.solidworks.com>.

Part III: Maintaining SolidWorks

SolidWorks itself can help you find answers to your questions through the Customer Portal. Areas of the Customer Portal are available to everyone, but for most of it, you need a valid subscription. With the exception of the Forums, this chapter shows areas only available through subscription. Figure 16.1 shows the front page of the Customer Portal from which you can access the other support resources discussed.

FIGURE 16.1

Getting information from the SolidWorks Customer Portal



The My SRs section shows the support calls that have made it to SolidWorks either directly or through your reseller. Figure 16.2 shows my support calls that have gone through directly to SolidWorks because of my status as a Solution Partner, because I write SolidWorks books. Again, the SRs are just service requests, and when the service is complete, the SR is closed. It doesn't mean that the issue is fixed; it can mean, for example, that an SPR number has been issued.

The SR number in the left-most column is a specific identifier that uniquely identifies your call for help.

The My SPRs section shows all the SPRs to which your name is assigned. I encountered a strange glitch in this list: if you have a very early SPR, say, one with fewer than six digits, the Customer Portal lists it by the left-most digit. This can make it very challenging to find some of the earliest SPRs in your list to see what has become of them. Figure 16.3 shows My SPR list with SPRs in various states, starting with an SPR issued in approximately the 2002 period.

FIGURE 16.2

Checking your service requests

My SRs (Service Requests)

Service Requests (SRs) are support questions or issues.
Software Performance Reports (SPRs) are confirmed software issues or enhancements.

My SPRs - Listing of your SPRs (Software Performance Reports).

[About Technical Support](#) - Learn about technical support options.

My SRs by My eMail Show by: **My eMail** [My Serial #'s](#)

[Query](#) 1 - 20 of 20

| SR Number | VAR SR Number | Summary | Status | Sub-Status | Account | Site | Created | Serial # | Beta |
|--------------|---------------|---|--------|------------|-------------|----------|------------------------|----------|------|
| 1-1180534186 | | Partner disable request. | Closed | Resolved | Designstuff | Steunton | 4/22/2009 10:44:55 AM | | |
| 1-1179152162 | | Solution Partner may possibly need a machine de-activated in the Activation Manager. | Closed | Resolved | Designstuff | Steunton | 4/21/2009 04:37:02 PM | | |
| 1-1101360320 | | The first problem with this part is the Thicken3 feature. As you can see it has failed, but if you delete it, it shows several features as children | Closed | Resolved | Designstuff | Steunton | 1/28/2009 10:45:00 AM | | |
| 1-1087443517 | | Surface model shows flipped trim behavior. Example file submitted to development SPR#470716. | Closed | Resolved | Designstuff | Steunton | 1/12/2009 09:22:32 PM | | |
| 1-1076979371 | | Follow-up on quality concerns on legacy split line features in SW2009 from comments made on Blog site. | Closed | Resolved | Designstuff | Steunton | 12/30/2008 09:28:30 AM | | |
| 1-999551673 | | FW: licensing | Closed | Resolved | Designstuff | Steunton | 10/7/2008 05:23:53 PM | | |
| 1-999231690 | | ** FW: licensing - "standalone license file failure" | Closed | Resolved | Designstuff | Steunton | 10/7/2008 02:45:06 PM | | |

FIGURE 16.3

Checking your list of SPRs

My SPRs (Software Performance Reports)

Service Requests (SRs) are support questions or issues.
Software Performance Reports (SPRs) are confirmed software issues or enhancements.

SPR List and Lookup - List of SPRs fixed within a Service Pack (SP).

My SPRs - Listing of your service requests (SRs).

[About Technical Support](#) - Learn about technical support options.

My SPRs by My eMail Show by: **My eMail** [My Serial #'s](#)

[Query](#) 1 - 20 of 20

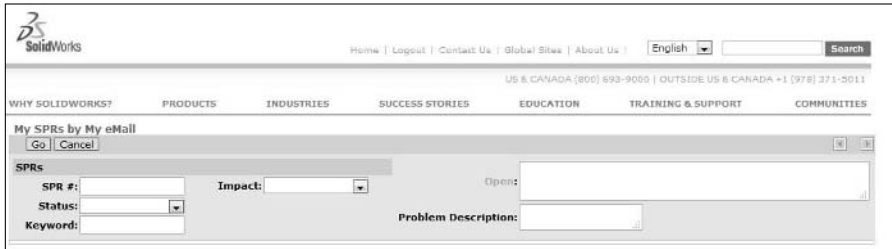
| SPR # | Problem Description | Status | Impact | Product | Technical Comments | Fixed in Version |
|--------|--|-------------------|--------|------------|--------------------|------------------|
| 112795 | Surfaces often have overlapping edges or gaps when they are extended, they should always be one continuous | Open | | SolidWorks | | |
| 203267 | Reference point defined at the intersection of a plane and an edge (or sketch line) is non-parametric | Implemented | High | SolidWorks | | 021006.001 |
| 217497 | Sketch automatic relations inferred by this operation would over define the sketch warning | Open | Low | SolidWorks | | |
| 240403 | Fillet preview does not correctly display loop selections | Closed (Inactive) | Low | SolidWorks | | |
| 245316 | Loft connectors do not appear in the lofted cut preview | Open | High | SolidWorks | | |
| 271618 | Incorrect error message warning of shell failure | Implemented | Medium | SolidWorks | | 2005 SP04 |
| 271675 | Unable to create a reference point at the intersection of an axis and a plane | Open | Low | SolidWorks | | |
| 273474 | Selecting a loop for a parting draft feature creates the draft only from the first edge selected. | Implemented | High | SolidWorks | | 2007 SP02 |
| 276744 | Instability when editing a sketch in this part and adding a relation | Implemented | High | SolidWorks | | 2006 SP01 |
| 279851 | Replace face does nothing but removing the split line in this part. | Closed (Inactive) | Low | SolidWorks | | |

Part III: Maintaining SolidWorks

To search through either the SRs or SPRs, click the Query button in the upper-left of the chart. The search itself in Siebel is rather primitive, but if you have specific information such as the SPR number, you can find what you are looking for. Figure 16.4 shows the search interface.

FIGURE 16.4

Searching for SPRs

The screenshot shows the SolidWorks Customer Portal search interface. At the top, there is a navigation bar with links: Home, Logout, Contact Us, Global Sites, About Us, and a language dropdown set to English. Below this is a search bar with a 'Search' button. A horizontal menu contains links: WHY SOLIDWORKS?, PRODUCTS, INDUSTRIES, SUCCESS STORIES, EDUCATION, TRAINING & SUPPORT, and COMMUNITIES. The main content area is titled 'My SPRs by My eMail' and includes a 'Go' button and a 'Cancel' button. Below this, there are search filters: 'SPRs' (a dropdown menu), 'SPR #' (a text input), 'Impact:' (a dropdown menu), 'Status:' (a dropdown menu), 'Keyword:' (a text input), and 'Problem Description:' (a text input). There is also an 'Open:' button and a large text area for results.

One of the quirks of the reporting system is that it will report that it is showing, for example, 1–20 of 20+ results. You never know exactly how many results there are until you get to the last one. This is certainly an annoying quirk, but if you approach the search with the idea that, if what you are looking for is not on the first page, you are probably not going to find it anyway, you may be less dissatisfied with it.

Another useful way to process SPRs is to use the Fixed SPR list, available from the Customer Portal front page. This page enables you to search through each version of software plus search add-ins and find which SPRs were fixed in that particular version. This is one of the best aids to help you figure out if you need to install a particular version or service pack. Figure 16.5 shows this interface.\

Beyond tracking your SRs and SPRs, you can also search the SolidWorks Knowledge Base for information related to a particular topic. The answers that exist inside the Knowledge Base are probably the information most worth having of any SolidWorks documentation. It is pulled from tech support answers, and you will sometimes find very candid and unspun answers that you may not see in any other official source. To get the best results, you have to be careful with your selection of keywords, and have a little patience with reading through solutions that may not apply, but the value of some of the answers here certainly outweighs the inconvenience of needing to sort through a fair number of solutions. The results for “sheet metal thickness” are shown in Figure 16.6

FIGURE 16.5

Checking for fixed SPRs in a service pack

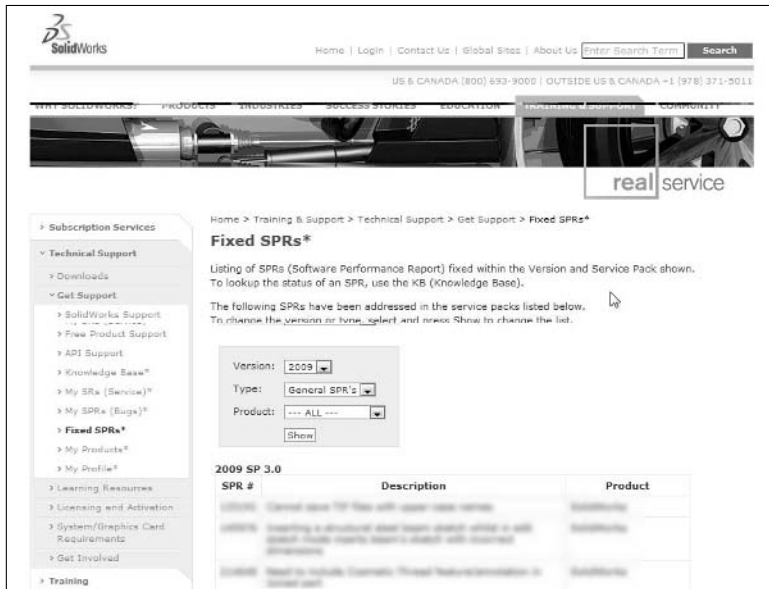
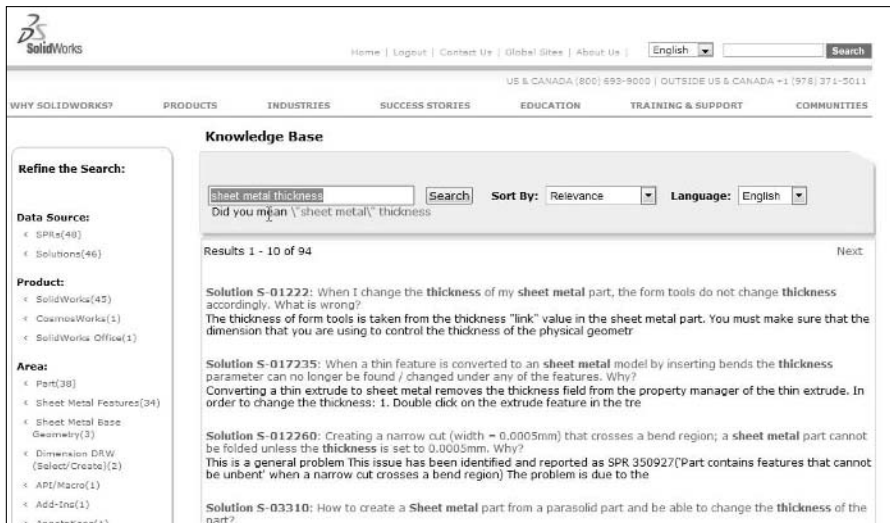


FIGURE 16.6

Searching the SolidWorks Knowledge Base for sheet metal information



If you go through all this searching, including searching the Help on a particular topic, and still don't find what you are looking for, you may want to submit an enhancement request. The enhancement request form is listed on the Customer Portal front page so you can access it easily.

Using the SolidWorks Forums

Probably the best source of technical support on the SolidWorks Customer Portal site is the SolidWorks Forums. I have researched much of the information for this book and other books I have written on the SolidWorks Forums. It is the single most active forum for SolidWorks users after the demise of the Usenet newsgroup comp.cad.solidworks.

The SolidWorks Forums have gone through many incarnations in the last several years, and they are scheduled to change again during 2009. The one thing you can count on is a high level of top-user traffic. The forums are a great place for everything from asking if other people have seen a particular bug in a certain version, to asking why you can't get something to work, to even asking straight engineering questions independent of the modeling. When I want to ask other users a particular question, the forums are a great place to do that.

It is possible that you could have a different kind of experience with the forums if you approach it incorrectly. I suggest the following:

- **Do a search before you ask a common question.** Longtime users will sometimes berate newer users if they don't do this. For example, before asking if the ATI Radeon is a good graphics card for SolidWorks, consider doing a search. A search should turn up results on frequently asked topics. A forum other than one administered by SolidWorks will have a FAQ section that outlines some of the more basic questions that people have. The search function in the current incarnation of the SolidWorks forum is limited.
- **Don't post a complaint unless you have done your homework.** If it turns out that you are incorrect, you may be handled discourteously. In particular, if you complain loudly about a bug that on examination turns out as a user error, don't expect sympathy.
- **If you have a question, add enough detail that someone who doesn't know what you're talking about could answer your question.** Upload a zipped SolidWorks part or an image file. These could be the difference between receiving an answer or another question.
- **Comparing SolidWorks unfavorably with a competing modeler will probably not entice anyone to answer your question in a useful way.** Often people transitioning from other software packages frame their question in terminology from the other software, and SolidWorks users may simply overlook the question. Try to be neutral, and remember that inside a SolidWorks forum, you are likely to run into people who are religiously loyal to SolidWorks. It goes with the territory.
- **It is okay to disagree with another person on the forum, but do not let it become personal or insulting.** People should be held accountable for passing incorrect information, and if you feel the need to point it out, do it in a professional way.

- **Try to give answers when you can, even if you are a new user.** Sometimes people who only ask questions without trying to contribute something back appear a little selfish. There are many types of questions that less experienced users can answer as helpfully as veterans.
- **Remember that the relationships you form in the SolidWorks forums can last for years, and can lead to real-life meetings at SolidWorks World or other user group events.** Meetings on the forums can also lead to work. Remember that there is a line between personal and professional interaction, and although social interaction on a forum may appear informal and personal, it is still professional interaction.

Using other social media

On-line forums are a form of social media, but some of the newer forms such as Twitter and Facebook are emerging as places to find resources. It is difficult to provide actual support via Twitter and Facebook, because the formats are not adapted for that sort of interaction as much as forums are, but you can certainly find and connect with people who manage available resources. For example, you might have some questions about a particular PDM tool that the local reseller is unable to answer to your satisfaction. You can use Twitter to connect with other users who discuss that software, and they may be able to guide you to a discussion group, blog, or Web site on the topic.

Facebook is full of SolidWorks groups that you can use to connect with experts in various fields. Again, Facebook itself is not likely to provide the actual answer to your question, but it is highly likely you can find someone who will know.

Tip

You need to be careful about mixing professional conversations with personal conversations. These lines are sometimes blurred in social media outlets. There is such a thing as getting too personal in a professional relationship, and some topics are better left alone in professional settings. Remember, online relationships still follow the common sense rules of real-life relationships. ■

Blogs are another social media source of information. Many SolidWorks related blogs are still used primarily for entertainment or as an outlet for news releases. The number of blogs grows monthly, but at the time of this writing, there are probably a couple dozen SolidWorks-related blogs that post anywhere from daily to monthly. The biggest advantage of most of the SolidWorks blogs is that they are written by SolidWorks users, not by marketers.

Going beyond technical support

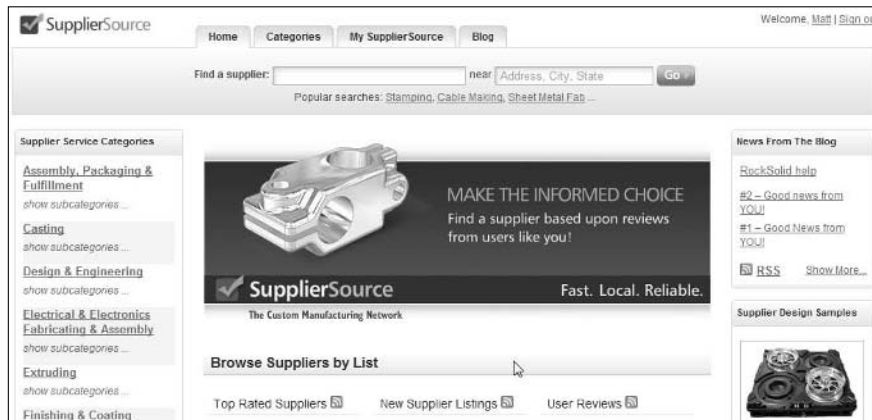
When general reseller support is not enough, you may need to call on the services of specialized consultants for specialized types of help, whether it is special technique development, suggestions on modeling, getting past a particular technical hurdle, or other issues. If you require help with Finite Element Analysis (FEA), PDM, plastics or mold modeling, or some other area of specialty for which your SolidWorks team does not have a skill set, it may be time to call on a specialist

Part III: Maintaining SolidWorks

consultant. One place you can find resources is the SupplierSource Web site (www.supplier-source.com). This link is also available from the SolidWorks home page. You can find consultants on a wide variety of topics on this page. The SupplierSource Web site is shown in part in Figure 16.7.

FIGURE 16.7

The SupplierSource Web site helps you find parts suppliers.



Providing Your Own Tech Support

Whether you are on subscription or not, you may still choose to provide your own tech support. Depending on the type of modeling work your SolidWorks users do, you may need to be a serious power user to be ready for this kind of task. It is best if you can try it for a while with the reseller as a backup. The worst thing to do would be to get in over your head with user questions, and then not be able to complete your other work while the users are losing productivity.

Even for a serious power user, the toughest trick of all may be keeping ahead of the rest of the users, even through software upgrades. This is partially why I think anyone functioning in an expert role needs to keep a hands-on grasp of the software. It is impossible to remain an expert through version changes without actually using the software.

One of the most difficult things about doing your own support is realizing that the software is huge in scope, and gets larger every release. You might be a modeling expert, but that does not make you an FEA expert, or a networking or hardware expert. Handling tech support internally at your company probably means assembling a team of go-to individuals. You might need help from IT to troubleshoot hardware problems and even someone from accounting to help you with PDM SQL database updates. You might need someone from the shop to help with technical problems with plastic parts or mold design. Keeping everyone going can be a team effort.

In the rest of this chapter, I talk about some of the smaller pieces that have to be brought together in order to successfully support a company full of diverse users.

Troubleshooting hardware and software

Troubleshooting can certainly be a tough job. Some people have a natural instinct for it, and others may struggle all day and never ascertain the cause of the problem. You may run into times when understanding the actual problem is not nearly as important as getting the system up and running again. In situations like these, you may find that a “replace first, ask questions later” mentality is required to get the job done. It may look inefficient at the time, but if you simply replace a faulty unit with one that works, then troubleshoot back in the lab, that might be the best way to approach the issue with the least downtime.

Troubleshooting software can be even more difficult because even if you can find the source of the problem, you cannot fix the software (unless you wrote it). In software, the only fixes you can look at are changes to the data (SolidWorks document) or a workaround technique to get around some faulty code in the software.

Eliminating PEBCAK errors

One of the first steps in troubleshooting hardware or software is actually determining that you aren’t dealing with operator error. Operator error is sometimes laughingly called a PEBCAK error. It sounds sufficiently technical, but it stands for “problem exists between chair and keyboard.” The PEBCAK error is one of the most common hardware or software failures, and eliminating it as a possible cause of whatever you are troubleshooting is a prudent first step.

Actually doing this can be more difficult than it sounds. First, you have to see what the user is trying to do, and see if that portion of it makes sense. When beginning to troubleshoot, I typically take a model from the user and roll it back just to see what was happening before the trouble began. I might also run some of the diagnostics in SolidWorks such as Tools ⇨ Check, or use Verification On Rebuild (press Ctrl+Q).

Next, I might ask the user to try to do the same thing that he was having trouble with before. Sometimes just breaking the chain of events resets something, or gets the user out of some particular command. Often when things like this happen, it’s not worth investigating unless it is happening frequently and causing lost work.

If I can see that the user is not doing anything that is obviously wrong, I might ask to sit down on their machine and try the same thing. I’ve seen situations where people had a book in the way that consistently clicked the right mouse button when the mouse was in a particular position, or maybe they were leaning on the spacebar when they tried to do something. A spaceball might be buried under a stack of papers that is causing the part to move consistently in one direction. It isn’t possible to write a thorough checklist for troubleshooting all the possible things that users could possibly do wrong. You just have to keep your eyes open and maybe get in and try to duplicate what the user is doing yourself to find the problem.

Determining software bug or data corruption

One of my pet annoyances is when people simply declare, offering no proof whatsoever, “It’s a BUG!” or “My data is corrupted.” All they know is that they tried to do something, and it wouldn’t happen. Therefore, there’s some sort of problem, and it’s about a 50-50 chance whether it is a bug or data corruption.

Although both kinds of errors do happen, you can’t just pick what type one is without saying why you think this. If the problem is going to get fixed, there has to be a reason for it. SolidWorks tech support people never fail to ask for a reproducible case of whatever the bug is today, but you can see why they do it. If you don’t give them a reproducible error, they could be grasping at straws and never give you a solution or answer. If you’re going to report a bug, you should be able to reproduce it or be willing to answer all applicable questions.

Making this choice happens to me frequently in my work with SolidWorks. The software crashes, so I do the same thing again, and it crashes again. I reboot and do it again, and it crashes again; then I move to another computer, move the file over, and it crashes again. Sounds like software bug, right. Well, then I try the same command or technique with a different part, and it doesn’t crash. Ahhh, it’s a corrupt file.

I have had success debugging corrupt files. You don’t always need superman powers to view to the guts of the file to do this; sometimes you can delete one particular feature and the data corruption is gone. Sometimes a simple Save, or even Save As with a different name, will do it. In one case, I suppressed a feature and then was able to edit the sketch for the feature without rebuilding the feature. That was enough to unsuppress, and when it was all rebuilt, the data corruption was gone. If you have the time to track things like this down, you can often fix your own data corruption. If the only other option is to remodel your part completely, you will usually find that you have the time to debug.

Most of the time, to find and fix data corruption errors you must be patient, willing to crash a lot without getting flustered, and be able to think through various scenarios methodically.

Another important and maybe contradictory skill is being able to admit when you are beaten. If an imported part comes into SolidWorks and has small holes on all types of faces, sure, that’s something you might fix unless the number of holes is something like 500. In that case, another solution might be better, such as hunting down the fellow who gave you such a mess of a part, so he can round-trip tests every export he does, or trying some other import technique such as importing to Rhino first and saving to SolidWorks.

Troubleshooting imported data is often a futile exercise because it comes down to finger pointing between people and software packages. My basic rule of thumb for import/export is that if I export a file for someone else to use, I should be able to import the file that I exported flawlessly. There are times with configurations and various other situations when exported files cannot be what you expect them to be. However, if you perform a round-trip test, importing your own work, you will be safe. Then you know the export starts with good data, and if someone else has trouble with it, it’s easier to pinpoint the problem.

As far as identifying bugs in the software and separating them from other types of problems, keep in mind that people who have a stake in the software, such as direct employees of the company that wrote the software or people who are paid to support it, are going to be biased toward the software in almost any situation. This is something you have to watch out for with reseller support technicians and even SolidWorks direct employees. Let's say you have a problem like a Trim Surface command that every time you rebuild, it flips the side of the surface that it deletes. Don't be surprised if an inexperienced support technician freezes when you say the word "surface." Most reseller tech support doesn't go past the Advanced Assemblies training book in terms of complexity. Reseller support techs are also prone to the "goal tender" syndrome, not letting anything, especially a bug report, get by them. "That's a limitation of the software." "Please submit an enhancement request." "Is your graphics driver up to date?" Don't laugh; you've heard it too. Part of the problem is that sometimes these support clichés are exactly the right answer. It's still frustrating to hear them because they just sound like another way of saying, "I'm sorry, I know you've paid for help, but I can't help you with this request."

Part of being able to confirm a bug is undoubtedly due to biased points of view like this. However, another part of it just comes down to an intimate familiarity with the functionality of the software. If you can show that it functions differently in a way that is not intentional in two versions or service packs, you can score yourself a bug or even a *regression* (a new bug introduced in a new version of the software or a service pack). When you are doing your own support, bugs and regressions are more difficult to deal with, because you just need to develop a workaround. If you let SolidWorks or the reseller know when you find bugs, that's great, because they don't get fixed unless SolidWorks has them documented. I guess you could say that all bugs that have not been in SolidWorks since the very beginning are technically regressions, but SolidWorks only uses that terminology if they can identify exactly when the bug was introduced. A regression gets first priority for a fix, presumably because it is easier to find, since you have already tracked it down for them.

Distinguishing between hardware failure and setting problem

This book has an immense Appendix C full of Tools ⇄ Options settings, most of which are obscure and rarely used. Users will sometimes find the settings and just start changing things to see what it does. As a SolidWorks Administrator, you have to know what these settings do. You have to be able to recognize the difference between bugs, hardware malfunctions, and bad settings. What would you do if you saw someone make a rectangle, and then drag one side of it only to have the side of the rectangle detach from the rest of the lines? What about if you saw someone fully define a sketch and then drag it and the dimensions changed to match the drag? Both of these are settings, not bugs. The first one is called Detach Segment On Drag and the second one is Override Dims On Drag, both from the Tools ⇄ Sketch Settings menu options.

Video problems are the most common type of errors in SolidWorks. Even if you are using approved hardware and drivers, you can still see video glitches. The main test for OpenGL hardware/driver errors is to choose Tools ⇄ Options ⇄ Performance (with no documents open) and select the Use Software OpenGL option. This essentially shortcuts your video card and driver, and replaces them with graphics driven by software run on the CPU. This slows down graphics performance; however, it should also eliminate the video card and driver as sources for error.

Troubleshooting with SolidWorks Rx

SolidWorks Rx is a diagnostic aid that enables users and SolidWorks technicians to have a look at your system and log files created during software failures to help you find real solutions to sometimes tricky problems. SolidWorks Rx logs are often requested by SolidWorks and reseller support technicians because they record answers to many of the standard questions that technicians must ask SolidWorks users looking for support.

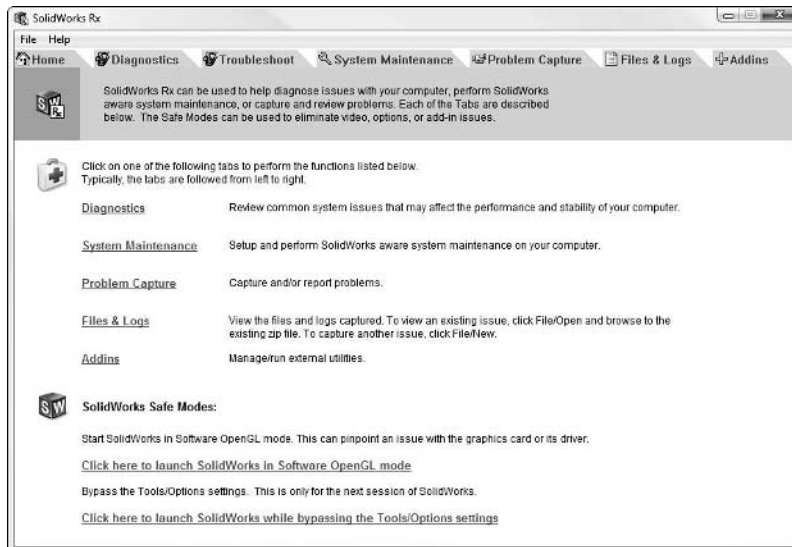
When most people think of the letters Rx, they think first of a pharmacy, then of a cure to a problem, or a prescribed set of steps that will result in a cure. SolidWorks Rx is not exactly that. Rx is more of a diagnostic tool than a prescribed cure. You may come out of Rx with more questions than answers. You are unlikely to receive a cure directly from SolidWorks Rx.

SolidWorks Rx has been mistrusted by users and administrators in the past because the data sent to SolidWorks was not very transparent. You needed a special reader to make sense of it, and some users felt that the seeming secrecy pointed to information that SolidWorks was taking was something they didn't want users to know about. Now the Files & Logs tab spells out exactly what is being sent, although some of the actual data is still rather cryptic.

This is not to complain about the functionality, because as a diagnostic or troubleshooting tool, it is extremely valuable. However, I do not want you to take a false impression from the name that it provides cures. You need to be aware of where your information is going, and yes, some of the information they collect and transfer can be sensitive. Figure 16.8 shows the SolidWorks Rx interface.

FIGURE 16.8

Initiating the SolidWorks Rx utility to help solve problems with users



The Rx Home tab has some very useful troubleshooting controls. The Software OpenGL mode is considered a video safe mode. Any crash that happens when in Software OpenGL mode is probably not a video card or driver crash.

By passing the Tools ⇄ Options settings cuts out another common possibility for errors, which is called by several names. You might hear it called Profile corruption, user corruption, or Registry corruption. It all boils down to the same thing. This option is meant to bypass your Windows user registry entries with default settings. If the problem goes away in this mode, you will probably have to recreate your Windows user, and delete the current login. Depending on how your Windows permissions are apportioned, you may need to recruit IT to help you with this kind of thing.

Using Problem Capture

The standard way to use SolidWorks Rx is the Problem Capture function. This starts a SolidWorks session running video capture, during which the instructions ask you to reproduce your problem. This is often easier said than done, and is sometimes the cause of users' frustration with SolidWorks tech support in general and impatience with the Rx application in particular.

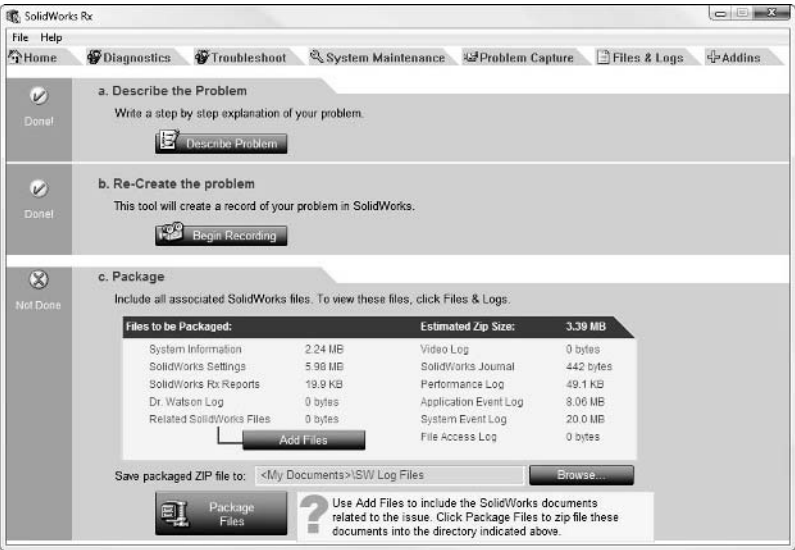
If you have a problem, you would like to report to tech support, follow these steps:

1. **Describe the problem in words.** Especially if your problem is repeatable, describe the steps and preconditions that lead up to the problem. The problem could be a graphics glitch, software crash, data corruption or a wide range of other possibilities.
2. **Record the problem on video.** SolidWorks support technicians have a difficult solving a problem if they can't see it or reproduce it. Don't be surprised if technicians back away from offering you further assistance if your problem does not seem feasible or it cannot be reproduced. Support people spend a lot of time hunting down problems, and they are difficult enough to find when you have good solid clues. However, with no clues or evidence, they may be unwilling to spend much time on your problem other than to give you suggestions for things you can try to do to capture the problem.
3. **Send in the packaged files.** SolidWorks Rx accumulates information on your system, and other items shown in Figure 16.9. If you send in this packaged data support technicians will stop asking you what version of video driver you have (because they will already know).

You may find that these items may be presented to you out of order. Specifically, in preparing to write this section, the Describe Problem button was not available until after the Begin Recording button had been pushed, and the Package area was grayed out until a problem description had been added.

FIGURE 16.9

Preparing a SolidWorks Rx package to send to support



Diagnosing with Rx Diagnostics

The stability of SolidWorks software depends heavily on your system configuration. SolidWorks Rx Diagnostics tab accumulates and automatically evaluates your system specs for you immediately. A diagnosis of my computer is shown in Figure 16.10. If I were having problems running SolidWorks, this view of my system says that my video driver is not certified and may be part of the problem, or that write access is limited to my Temporary directory.

The *I* symbol is providing information, not necessarily a problem or even a warning, just information. The *X* symbol is a definite problem. The check mark (✓) means the item is okay.

This information is useful, but would be even more useful if they provided a path to the folders they were talking about, since the software must presumably know where those locations are. It would also be useful if the name of the computer was provided in these results, and the results could be saved as a report.

Troubleshooting your system

The links on the Troubleshoot tab are just pre-defined searches in the SolidWorks Knowledgebase. While this information may be of great general interest, it may not be useful for finding an immediate answer to a specific problem. Certainly, searches of the Knowledgebase are recommended on key words that are more specific. Figure 16.11 shows the available links on the Troubleshoot tab of the SolidWorks Rx.

FIGURE 16.10

Reading through the suggestions for fixes on the Diagnostics tab

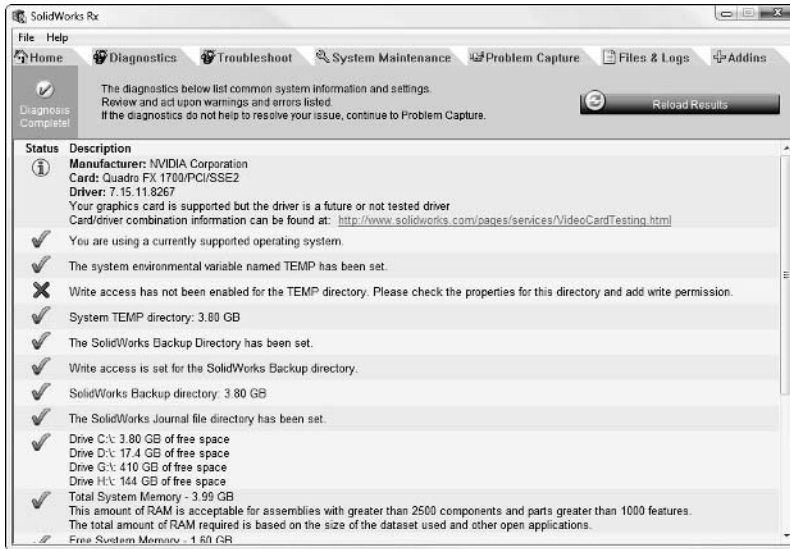
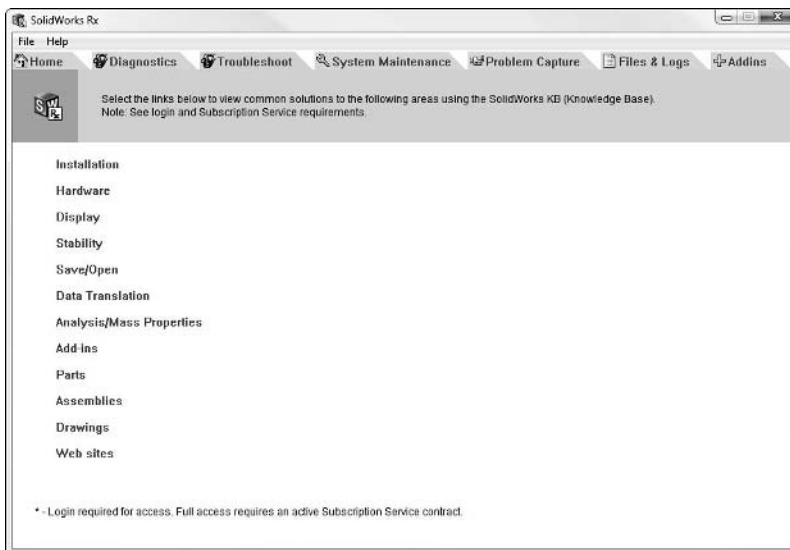


FIGURE 16.11

Links from the Troubleshoot tab help you search the Knowledgebase for clues to your solution



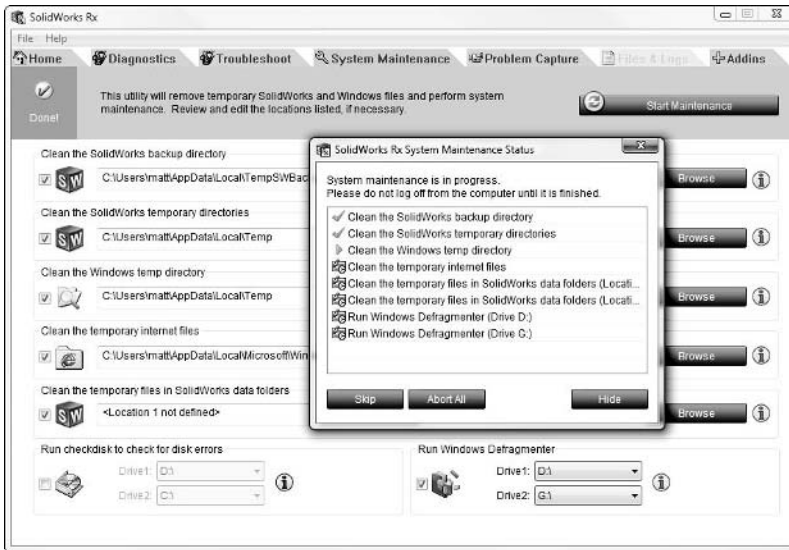
Part III: Maintaining SolidWorks

Performing regular system maintenance

The System Maintenance tab is probably the most important tab in the Rx. Many of the SolidWorks system maintenance best practice recommendations are automated here as a part of a maintenance routine that you can run at the press of a button. You or your users may not even be aware that some of these checks are necessary. Deleting files from some of the locations named on the System Maintenance tab will certainly go a long way toward keeping your SolidWorks running fast and clean. Figure 16.12 shows a routine maintenance run in progress.

FIGURE 16.12

Running a routine maintenance check using SolidWorks Rx



Understanding Files & Logs

The Files & Logs tab summarizes some of the information SolidWorks Rx has accumulated and you have entered. The information includes data from your SolidWorks installation, usage (journal files) and your Windows settings (registry). All of this data, along with any video you may have recorded is being sent to a packaged zip file specified on the Problem Capture tab, and saved to a directory you specify there. SolidWorks is trying to be more transparent in this process hoping that more users will trust the process and use it, making the job of SolidWorks and reseller support technicians easier. Figure 16.13 shows the Files & Logs tab of the SolidWorks Rx.

FIGURE 16.13

Examine a summary of the data you are sending to SolidWorks on the Files & Logs tab



Transferring data to SolidWorks

To transfer this data to SolidWorks, you need to use the file you create on the Problem Capture tab, and either FTP or email (if the data is small enough) it to your reseller or SolidWorks directly. This is another obstacle for the acceptance of the Rx tool. It does require some effort. Some users' eyes simply glaze over at the mention of FTP, even though it is nearly as easy as Windows Explorer to use.

This type of file transfer was preferred over the more secretive Performance Feedback, which is no longer sent by email, but simply sent directly from your computer to the SolidWorks server.

Looking past the obvious for solutions

Potential hardware conflicts can include the mouse. Depending on mouse driver settings, you may see that when you zoom into a certain point on a model, the scroll wheel suddenly changes from zoom to panning the scroll bars. This is usually due to a particular Microsoft mouse driver and can usually be fixed by uninstalling the driver. Unless you are using advanced mouse driver functionality, you can probably get away with using the built-in Windows default mouse drivers. I have never installed a special mouse driver and do not have the types of problems with mice in SolidWorks that other users have reported.

Having done a fair amount of tech support, I found that one of the most common conflicts in SolidWorks comes from add-ins, and in particular what used to be known as the spaceball drivers. The devices formerly known as *spaceballs* (currently sold by Logitech under the 3Dconnexion brand) are prone to conflicts with other software and prone to bugs in the software and drivers for the devices themselves. A common part of troubleshooting is to turn off the add-in software and disconnect the device. This can even be true with other more integrated SolidWorks add-ins such as Simulation and PDM applications.

Finding Answers to User Questions

Sometimes users come up with tough questions, or the simple question they pose has implications that are more complex. For example, maybe a user has asked how to handle FEA results in PDM. Do you manage all the little files created by the FEA program, or do you just create results PDF and manage that?

How would you go about researching a question like this? If you ask a SolidWorks reseller a question like this, the answer will inevitably be preceded by dollar signs. Answers that are too predictably one-sided may not be trustworthy. I would look for answers from people who have tried it, regardless of if they failed or succeeded. Their experience is what really matters here.

You might predict that my favorite source for information of this sort is going to be the SolidWorks Forums. To get something possibly less biased toward SolidWorks-centric solutions, you might try *eng-tips* (www.eng-tips.com) or ask a question on one of the higher traffic blogs or even a PDM-specific blog.

General Google searches can sometimes turn up surprising results and help you learn about new sources of information. SolidWorks is widespread enough on the Web that you will get results. If you were having difficulty getting answers from other sources, I would not overlook a Google search.

Quite often, even areas such as Yahoo! Groups and social media sites like Linked-In can help you find technical answers, so don't overlook places like that either.

Sometimes, however, there is nothing that works quite like good old-fashioned, rolling up your shirt-sleeves and just doing the work yourself. When you are asking for help on a topic, other people often do not provide the kind of detail you wish you had. When you do the research yourself, this is something you can control easily. Sometimes you just need other people to get started, then once you're rolling, you can answer your own questions.

When you do find answers, whether others lead you to them or you figure it out yourself, you might consider documenting the answer in some way. You can bookmark a link to a Web site or online forum, or you can just write up a Word or PowerPoint summary of what you found. Even better would be a five-minute video showing how you achieved the results you were looking for. Keep all your documentation in an easy to access place where you and your SolidWorks users can refer to it for reference as needed from time to time.

Summary

SolidWorks support from your local reseller is part of the annual maintenance you pay for. There is no way to separate the payments for the support from the payments for the service packs and new versions. Sometimes companies with highly experienced users can provide their own support internally. Doing this presents several challenges that reach well beyond simple how-to modeling questions, into operating system questions and hardware troubleshooting. Providing support for your own employees will give you a new appreciation for what the reseller employees do.

If you do choose to provide support yourself, there are many places to find answers, such as traditional on-line forums, and some of the newer social media services such as Twitter and Facebook. Be aware of on-line etiquette and commonsense relationship guidelines even on these services. They may not provide the direct help that you are looking for, but they are very likely to be able to point you to a place where you can find what you need.

Collaboration

The word collaboration literally means, “work together” (co-labor). This is something that is difficult to do in a process (design) that is in most respects an individual activity, and on a computer that is called a “personal” computer. The part of CAD or engineering work that is the most collaborative occurs when you work with others on large projects. The word has become overused in the product-development world, especially in marketing circles.

Collaboration in the real world isn’t like what is shown in the glossy business magazines. It is more often two people working across the room or across the planet from one another, sharing data and making individual contributions to a larger project.

When it comes to collaboration in SolidWorks, the details are about sharing files, who gets read/write access, and how you communicate back and forth.

The obvious collaborative solution is Product Data Management (PDM). This chapter only touches on PDM as a reference point. Chapters 21 through 25 specifically cover SolidWorks Workgroup PDM. Collaboration in this chapter refers to the details of manual file management and working together with other SolidWorks users when you do not have a PDM program in place, and in particular how the settings on the Collaboration page of Tools ➤ Options affect your ability to share files with other SolidWorks users.

IN THIS CHAPTER

Sharing SolidWorks data

Making the setting work

Sharing Data

Sharing SolidWorks data without a PDM system is something users do on a daily basis, but is a risky way to treat your data. Inevitably, the best answer to almost any file management, data sharing, or collaboration question is “Get a PDM system.” PDM software enforces best practice rules and guarantees user discipline when doing manual file management.

However, PDM is only as good as its implementation; it cannot prevent people from making serious errors in judgment or from malicious behavior, so it will not completely erase problems in your document management system. If you take a poor manual document management system and implement a PDM system around it, the result will be a poor automated document management system.

Sharing models and other data among a group of SolidWorks users using manual file management techniques is not always simple, but many SolidWorks users still do things this way. Doing it correctly requires special settings on all computers, along with a good bit of training for, and discipline from, all the users involved. It’s unlikely all SolidWorks users who use manual file management techniques are aware of the potential problems they can encounter, so many SolidWorks data are probably lost or altered daily. Each company develops its own processes for dealing with data, likely based on learned lessons from making errors. Interestingly, manual file management is not typically taught in reseller classes early on. It is assumed users will figure it out on their own, even though it is complex and, in some cases, counterintuitive. In fact, for the times Help touches on the topic of file management, the information is buried indirectly under other topic headings, and you will not find it directly, only under something else.

Therefore, while the topic of manual file management is certainly one of the most important to address, and is potentially a source of costly errors, most users and companies are left on their own to figure it out unless they buy a PDM system and pay for training or implementation consultation.

One of the limitations of computers is that, generally speaking, only one computer can have a single document open at a time. Each document has a single person who is responsible for changes at a given time.

In SolidWorks, and more generally in Windows, when a user with write access opens a file, Windows locks that file from other users writing to it. Other users can still open the document, but only with read-only access. They can make copies of it, but the copies become disassociated from the original. A user with read-only access could try to perform a Save As, and either save the file with a different name or overwrite the original file. The operating system will prevent overwriting the file if the file is still open, but if the file has been closed by the original user with write access, the Save As by the user with read-only access can overwrite the file.

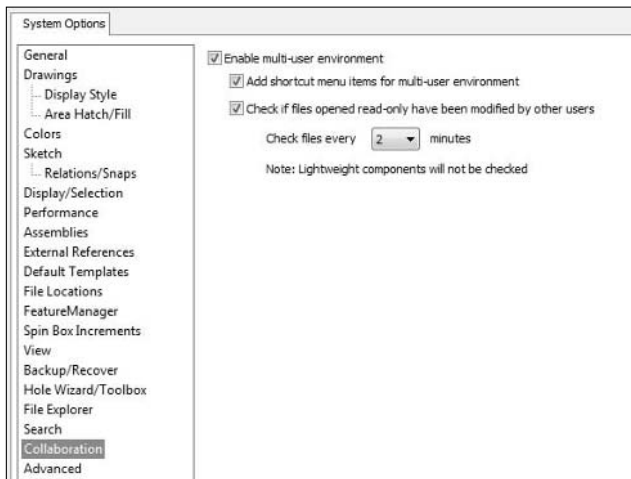
Part of what helps you share the data in SolidWorks is the use of special settings. These settings are the subject of the next section.

Using the settings

The nuts and bolts of the Collaboration settings in SolidWorks include the Collaboration page and some settings on the External References page. You can access these settings by choosing Tools ⇨ Options. First, I will show the settings so you can get a feel for what they are and where they are. Then I will cover how you need to use them to set up a sharing environment. Figure 17.1 shows the Collaboration page.

FIGURE 17.1

The Collaboration page found under System Options in the Tools menu



The settings shown in Figure 17.1 must be enabled on all the computers involved in the file sharing. They boil down to some very simple concepts.

Enable multi-user environment

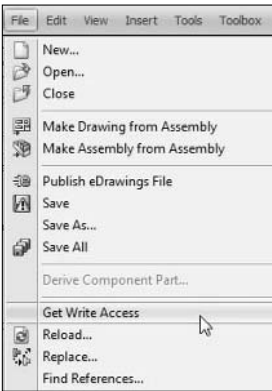
The Enable multi-user environment option simply enables access to the rest of the options on this page, as well as adds some tracking capabilities so that SolidWorks can identify (via tooltips) what user has write access to a particular document.

Add shortcut menu items for multi-user environment

The Add shortcut menu items for multi-user environment option enables either Get Write Access or Make Read-Only menu options in the File menu or the right mouse button (RMB) menu, respectively. Figure 17.2 shows the Get Write Access menu option.

FIGURE 17.2

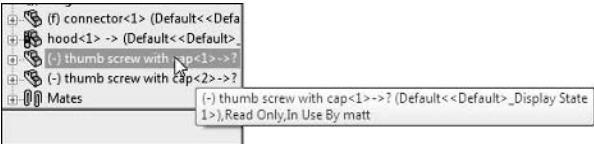
Get Write Access in the File menu



Another part of the functionality is that when you hover the cursor over a part that you have open as read-only, tooltips, shown in Figure 17.3, show you who has write access. The Get Write Access option does not automatically grant you write access if someone else already has it. You must have a conversation with the other user and get them to agree to relinquish control of the part by using Make Read-Only. Once the other person has made the part read-only, you can gain write access.

FIGURE 17.3

Tooltips on components in FeatureManager show who has write access to documents.



You can also change between read and write access in the Reload command without needing to use the Collaboration settings. The read-only option in Reload is somewhat less convenient than the RMB option, but it shows some additional information, and it can be accessed without prior planning to have the Collaboration options set. Figure 17.4 shows the Reload dialog box. You can find Reload in the File menu, or on the Standard toolbar, although it is not there by default. You can use Tools Customize to put it there, or anywhere else, if you want.



Reload is closely related to another function called Check read-only files that helps you manage read/write access and collaboration. This command uses the Reload interface to report on read-only documents.

FIGURE 17.4

The Reload dialog box is in the File menu.

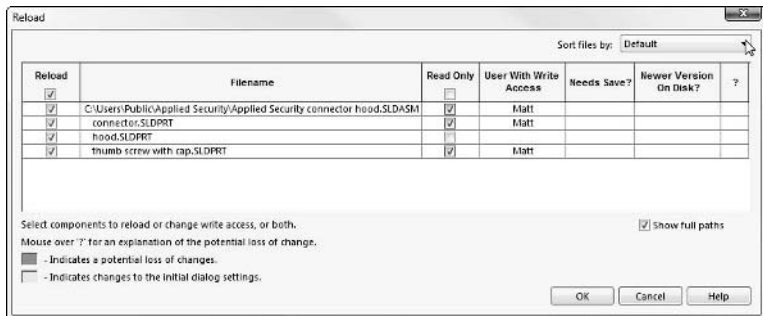
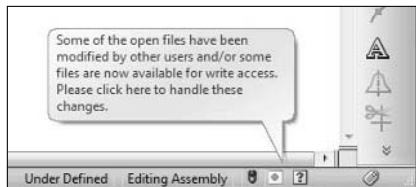


Figure 17.5 shows a balloon that pops up in the lower right-hand corner of the SolidWorks graphics window, announcing when a read-only part has been modified by another user, or when write access becomes available on a part that you have open as read-only. The icon does not appear on the Task Bar, though this may be fixed in later versions of the software. This screen shot was taken in a pre-release version of the software.

FIGURE 17.5

A balloon announces changes to read-only documents.



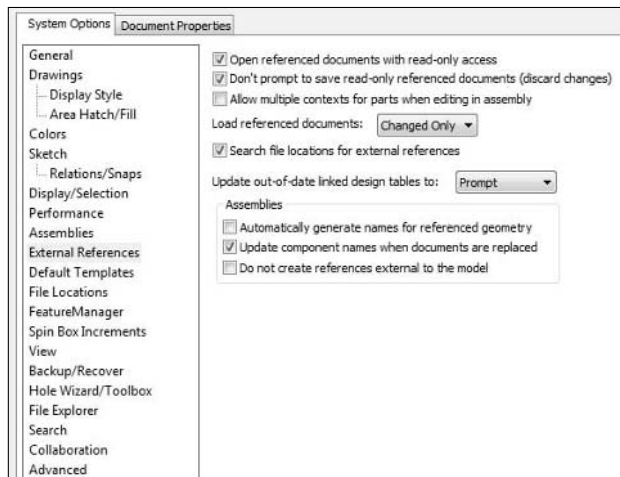
You can access additional settings on the External References page by choosing Tools → Options. These settings, as shown in Figure 17.6, combined with the Collaboration settings, enable you to manage multiple users sharing files back and forth between read-only and write.

The most relevant settings on this page for file-sharing environments are the first two: Open referenced documents with read-only access, and Don't prompt to save read-only referenced documents. To work with sharing files with others frequently, both of these options should be selected.

Now that you have seen the settings and functionality SolidWorks offers for collaboration, it is time to put it together into a narrative that demonstrates how collaboration works in practice and why you may need or not need the settings.

FIGURE 17.6

The settings found on the External References page on the System Options tab



Making It Work

Consider this a real-life scenario to put all the theoretical settings into context for you: One person has write access to all the documents, and no one else can get anything done. Your job is to turn this situation around and come up with a new way of working where everyone who needs write access can get it. In this situation, you can make the entire scenario work with or without the Collaboration settings, thanks mainly to the Reload command. While the Collaboration settings make some of the notices and tools more convenient, it is probably more the product of clever marketing than clever technical solutions.

In this scenario, you need to work with these settings before the first person in your CAD department makes it in to work. The first person to open the SolidWorks assembly gets write access to all of the files, and anyone who comes after has to make do with read-only access. Your early riser is going to be the one who fights change the most because he has everything to lose in a new way of doing things. In his old way of working, he opens the assembly at 6 a.m. and gets write access to the assembly and all the subassemblies and parts. He gets first dibs on the drawings, too.

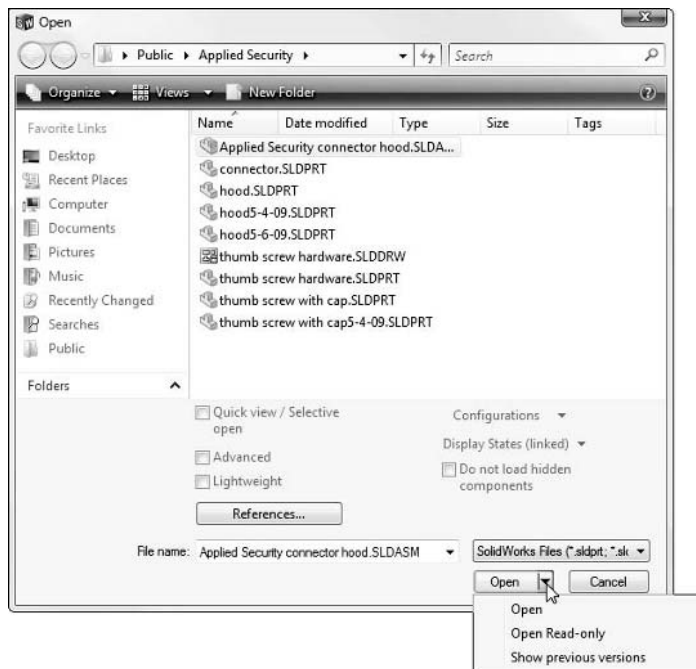
In the new way, every SolidWorks user has the Open referenced documents with read-only access option selected. So when your early riser arrives in the morning and opens up that assembly, he may get write access to the top-level assembly, but he has read-only to everything else. If he needs write access, he can use the File ➤ Reload tool, with the References option (refer to Figure 17.4). He can then check and uncheck the parts as necessary to get the appropriate access.

Of course, this is easiest early in the day when you are still the only one wanting access to the files. If someone else has a file he wants, the interface shows who that is. You must negotiate with that person to get them to relinquish control of that file.

If the early riser is feeling generous, he might open the entire assembly read-only from the beginning. To do this, he can click the drop-down arrow on the Open button in the Open dialog box, as shown in Figure 17.7. This opens the top level and everything else as read-only right from the beginning. To gain write access, he has to use the Reload command again.

FIGURE 17.7

Using Open Read-only



Once he has the entire assembly open read-only, if he goes to close it down, SolidWorks might prompt him to save each read-only part as a different filename. If the assembly is more than a handful of parts, this will get very tedious. To avoid this (dismissing a dialog box for every read-only part in the assembly), make sure your early-rising user is using the Don't prompt to save read-only referenced documents. The first time you avoid a hundred clicks, you will be glad you used that setting. On the other hand, if you have made changes to a read-only part, this setting also simply closes the parts without warning and glosses over the changes you made, throwing them away. If you use this setting and make changes, you must make sure you perform a Save As yourself; there will be no reminders, or second chances, after the data is gone.

Part III: Maintaining SolidWorks

So now, you have seen how to open an assembly with full write access, full read-only access, and write at the top level and read-only for everything else. What if the early riser in your scenario knows right from the start which parts he needs access to? Does he always have to go through the Reload routine? No, Reload is not the only way. There are two more ways to open files with only selective write access. The first one is a relatively new function called Selective Open (refer to Figure 17.7).

Note

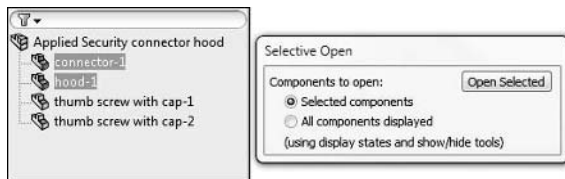
If you have a part selected in the Open dialog box, the Selective Open option does not appear. You must first have an assembly selected before it becomes available. ■

Using Selective Open

The Selective Open option opens the interface shown in Figure 17.8, which enables you to see the entire assembly until you start selecting parts that you want to open with the assembly. Parts that you do not select are hidden, and not opened, as if they were suppressed. When you click the Open Selected button, SolidWorks creates a display state from the visible and hidden parts, and the hidden parts are not opened at all, avoiding the write/read-only issue altogether.

FIGURE 17.8

The Selective Open dialog box



Selective Open may create a lot of extraneous display states that must be deleted now and then, but the advantage of being able to selectively gain access to just the parts you need is worth the bit of required housekeeping.

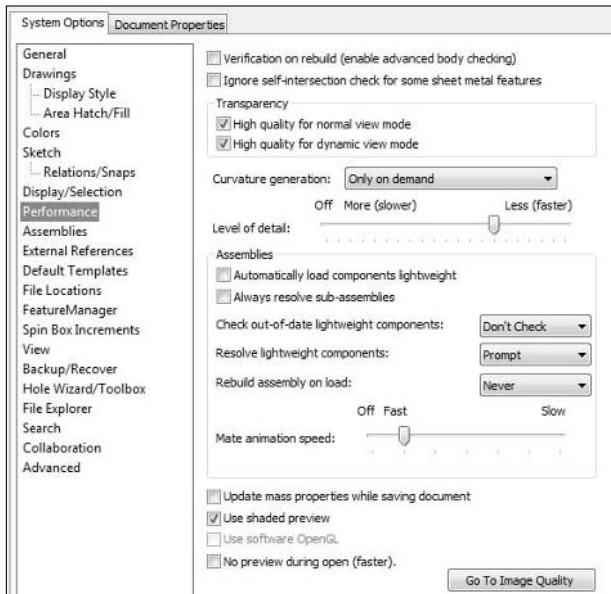
Using Lightweight as a collaboration option

Another method you may not have thought of as a collaboration technique is Lightweight. In an assembly, Lightweight information for the parts is stored so that parts in Lightweight state do not have to access the part file itself. While you cannot make changes to Lightweight parts, they certainly help you see other parts in the context of the rest of the assembly. You can open an assembly in Lightweight mode using two different techniques.

The first technique is to have the Tools⇨Options⇨Performance settings set to open all assemblies Lightweight by default. Figure 17.9 shows these options in the Assemblies area. Changing from Lightweight to fully resolved is automatic, and somewhat easier than changing from read-only to write. Remember also that it is possible to go from resolved back to Lightweight.

FIGURE 17.9

Lightweight settings in Tools⇨Options⇨Performance

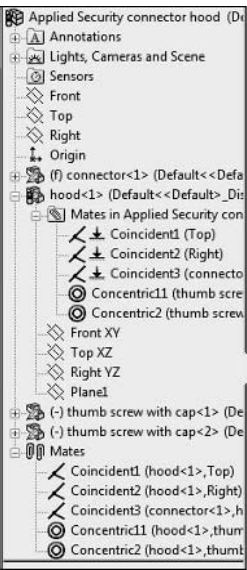


A second method that you can use on the fly rather than having a dedicated system setting is to use the Lightweight option in the Open dialog box (refer to Figure 17.7). Remember that the Lightweight option is only available when an assembly is selected. Lightweight parts show mates and the standard planes but do not show features in the FeatureManager, as you can see from Figure 17.10.

Many people have prejudices against working with Lightweight parts, but the functionality has come a long way in the last several years, and it may warrant a second evaluation. You can do a wide range of things in Lightweight mode, and if you are careful to keep tables and drawings updated with changes, you can certainly find a workflow that works for you and takes advantage of what Lightweight functionality has to offer, whether for performance sake or for collaboration/file access management.

FIGURE 17.10

The FeatureManager of a Lightweight assembly



Summary

While PDM is certainly the preferred way to manage access to files, many users prove on a daily basis that you do not need it to do professional-level work in SolidWorks. PDM simply helps enforce the manual file management rules of thumb that you establish.

If you choose to manage files manually, giving everyone access to the files that they need when they need them could become a tricky endeavor. Settings in various places in the SolidWorks Tools ➔ Options setting pages can be helpful, as can various techniques and tools to help manage the level of access each user requires.

Managing Change

One thing that is inevitable in life is change, and that is doubly true for computers and software. You cannot expect to keep up with all the change that happens. If you need to be an expert in SolidWorks for your job, it is such a huge topic that it is impossible to know about all the various corners of the software, much less all the changes in a new version or service pack. Beyond that, you may sometimes be asked to remember the version and service pack when a certain change takes place. Add in the compatibility of versions of the SolidWorks software with versions of hardware such as video cards, which change like the wind, and it is simply overwhelming.

A substantial part of your job as a SolidWorks CAD Administrator is to manage the process of change. Several tools can help you do this. Obviously, you need to read and maintain a record of the SolidWorks documentation of release notes and What's New documents. This record, along with using some search skills, will help you keep a handle on what has changed, and when it happened. If you have documentation for SolidWorks versions you have not yet installed, it will help you decide when the time is right to update to the next version.

Beyond the technical changes, another aspect of change applies to humans. Some people are addicted to change and exploring new things, while others are terrified of change once they have finally grasped the present. Both extremes can be problematic for SolidWorks CAD Administrators. As the Administrator, changing the software too early can be as disastrous as changing it too late. Your decision must be driven by rational thinking, not by feelings or emotions. In business, time and money are the main factors. If a change does not save you time or money, you should avoid it, because it is likely that upper management will not support it. If a change will benefit the company, you should proceed regardless of user resistance.

IN THIS CHAPTER

**Evaluating your situation
before considering change**

**Choosing your battles when
making changes**

Encouraging disinclined users

Evaluating Your Situation Honestly

Well-considered change involves knowing where you are starting before the change begins. You can't make great decisions about where you want to go if you don't know where you are presently. To decide the direction of CAD usage at your company, you have to consider a few things:

- **What are the CAD needs of your company?** Don't make the mistake of only asking the "experts." Too often, they have opinions based on self-interest, and when situations change, "expert" opinions often stay the same. You have to ask the people who use the CAD data. Do people need additional or different types of data? Is any of the data you are providing going unused?
- **How well are the current CAD needs being met?** Do the documents or data you produce fit the requirements? Could someone's job be more efficient if she were given a different type of data?
- **Are you using the best tools for the job?** You won't get a straight answer from anyone. Everyone has an ulterior motive when trying to steer you to a particular software package. The only real way of knowing which software is best for a particular job is to use it. That can be tough with CAD software because the learning curve takes so much time.
- **Will training improve your efficiency?** Evaluating individual users can be tough, but you have to do it. Training is a great thing, but it doesn't always solve the problem. The problem might be that the software doesn't really fit the need, or you might have a personnel problem to deal with. In addition, you may not want to pay for a lot of training right before you consider an upgrade to your software. Timing can be everything.
- **Are updates or upgrades available that will improve your situation?** Often in the drive to always have the newest options available, technology users tend to be blind to facts. The "cool" factor is emotional and should not drive decisions that cost companies money.

Too often evaluations of situations are exaggerated. It happens in business and the rest of life so frequently that we assume exaggerating is the right thing to do, but it isn't. Situations commonly have both good and bad sides to them. An honest evaluation realistically takes into account both. Being blind to either side makes you vulnerable to making a poor decision. Exaggerating reports to make change look either more or less attractive or to make the results of change look more positive will always come back to haunt you. If every year your initial report is gloomy and your final report is superlative, someone is going to see through that. Trying to manipulate management through exaggeration will backfire on you. People in management are smarter than that.

Acting on things you learn

When you do an honest evaluation of your situation, you learn a lot. Chances are that just getting people to agree on how the current system works for product development and documentation will be a challenging learning process for everyone involved. Finding problems that have existed for some time or new problems due to a new software or process always give you opportunities to improve. If other people scrutinize the CAD operations and find deficiencies, it is always a little uncomfortable to own up to the problems. The first thing in your mind in this situation should be

“Okay, so that’s a problem, what do I need to do?” You should be prepared for the answer that for the moment, it is most appropriate to do nothing at all.

Thinking like a manager

Managers think in terms of time and money, schedules and budgets, and sometimes just one or the other. Most problems can be reduced to just a money problem when it comes right down to it. Rarely is a problem purely technical. For example, if you need to develop a new technique to use your CAD data to figure out a manufacturing problem, the issue seems technical until you consider hiring an expert to solve the problem. Even though this costs money, it can also save money and time in the end. If you run into a situation where, even when hiring an expert, the answer is not guaranteed, or where the answer is that there is no good answer, maybe you have a problem that cannot be solved in the way in which you hope.

One thing a manager does not do is get caught up in emotional arguments about which CAD system is better than the others, or be affected by the “cool” factor of always living on the edge of new software to the point that it can be difficult to meet deadlines. If managers require users to stay on the front line of technology, it is because it is part of the business to be out front. The expertise that employees gain is of some value to the company in general.

Part of thinking like a manager involves calculated risk. In some situations, the value of trying something that is not guaranteed to work may pay off, and may be balanced by trying two different ideas simultaneously where the chances are that one of the ideas will work. Of course, both could also fail, or both may work.

When results are not guaranteed, and you have to take a risk to move forward, sometimes the answer may be a limited evaluation where you don’t risk the entire business. Sometimes going forward with limited resources on a project that is important, but risky, doesn’t make sense to the people working on the project. However, it is the best way for the larger picture, including keeping the entire business running and bringing in income, while couching against a possible failed project.

The same sort of concept can be applied to managing change, where you start a pilot project on new CAD software with just a couple of people. If the project succeeds, you roll the program out to a wider audience, building on what you have learned in the pilot project. If it fails, you back up and try a different approach with possibly a different small team, again building on what you have learned.

Failed projects are usually not a complete waste of time and money because you can learn a lot from failure. Failure is nothing to aspire to, but you don’t need to be afraid of it, and you need to be able to recognize it when it happens. Continuing in a failed implementation of software can be extremely costly for a company. Recognizing the failure so you can regroup and change directions is a very important skill. Denying that an implementation has failed doesn’t change the fact that it has failed.

Choosing Your Battles

When you are dealing with making changes to engineering or documentation processes at your company, you can run into people entrenched in a particular way of doing things that may be

Part III: Maintaining SolidWorks

resistant or even hostile toward making changes. You can approach this several ways, and some are more effective than others. Most of the battles can be won before they are ever fought by planning for the conflicts early, because you already know they are coming.

For example, one of the biggest changes a SolidWorks CAD Administrator can make at a company is to implement a PDM (Product Data Management) system. This completely changes the way people interact with CAD files, and, in fact, just about any kind of company data whatsoever. Twenty-five years ago, people had to learn how to sort things on a computer instead of in a physical file cabinet, and now they have to learn to do it some other way. You should formulate a response to the argument “I’m faster with the old method” that you will be sure to hear from someone. If you have management behind you, this is an argument you will only have to make once.

One of the things you have to do to get to the point where you can implement a system as costly and disruptive as a PDM system is to get management buy-in. You can’t accomplish much without their support. It helps with so many things down the road if management supports this change up front and publicly; this way, when you speak as the CAD Administrator in charge about the mandated PDM implementation, it is as if management is talking, and people automatically comply.

On the other side of the issue, let’s say that part of your PDM implementation includes a new part numbering system, and to be complete and correct, you want to renumber all the existing parts. You will see tremendous resistance to an idea like this, simply because of the amount of work it introduces and the possible number of errors that could occur through these changes.

Is this a battle you really want to fight? While what you are looking for is academic purity of a well-considered new numbering system, the opposition is right. It involves renumbering all the company parts, redoing all the company Bills of Materials (BOMs), Enterprise Resource Planning (ERP), and who knows what else. This may be a compromise that you should make. When making change, there is sometimes such a thing as pressing it too hard.

It’s true many people are simply resistant to any change because they liked the 1950s. Similarly, many people are too willing to change, and commit resources that could be used for the main mission of the company rather than for having the latest and greatest. Still, it seems to be the default assumption that older people are the ones resistant to change. However, this assumption isn’t always true. I have seen situations in which older people were excited about the change and younger ones were hesitant. Be careful about assuming stereotypes like this apply to individuals. Decisions about resisting or encouraging change are individual decisions, and every individual is different.

Change for the sake of change, while cool and exciting, does not necessarily create a profitable business. If you want change, you need to be able to provide evidence of a particular benefit, probably related to time, money, quality, or saleable innovation. Changing the company’s internal communication to Twitter or Yammer just because it is new and the uber-cool Wired blog said it was the new “it” app is not good enough. You have to demonstrate how it is an improvement over the e-mail or instant messaging your company currently uses.

At the same time, you have to find a balance and cannot resist necessary changes. The most obvious example in this respect might be a company that creates castings for the railroad industry that is using a 2D application for design. Complex castings are three-dimensional objects. When you

look at the way they are manufactured, your shop is using techniques from the 1800s to create new parts from your drawings. If you give them 3D data, they won't know what to do with it. To make the changes required to utilize 3D design data for manufacturing, they will have to lay off or retrain employees and buy new equipment and hire someone to run it.

This is a lot of disruptive change, but the end result is that you can hand off a 3D model without a drawing to your new machine shop, and they can make a casting pattern directly from that. This cuts 75 percent of the time and labor out of the process of creating a new casting from new engineering data, making it a change worth having. Upper management will support it if you lay out the scope of the entire change for them and can demonstrate low risk. This type of change requires planning for changes in personnel, training, equipment, process, vendors, and so on. You will make some enemies in the shop, but you will also make some friends.

Encouraging Reluctant Users

When you advocate for changes that affect people outside of your department, you can run into internal company politics and resistance. Even making change within your department, and within the scope of your CAD Administrator responsibilities, can bring about resistance.

The first part of being able to bring along people who resist the changes is to do your homework before they confront you. You must have a convincing answer for the people who think that they are more efficient the old way or just don't understand the new method, or for when someone points out some deficiency in the new process. You can't be blind to faults of newer technology, but you do need to have good answers. If you are taken off-guard, people will think you haven't thought through all the consequences of the new process, and they may be right.

Often times as a CAD Administrator, you are a junior employee, while the person reluctant to change is a senior employee. Authority or credibility can be an issue. This is one of the biggest reasons for getting buy-in from upper management. Most people at a company respect the decisions of upper management without question, even if they don't respect you or your decisions without question.

Remember that regardless of your seniority, most decisions affecting CAD users will need to be based on consensus between representatives from multiple affected departments. Big changes need to be overseen by a group of people who oversee the change and evaluate the results. The group itself can be led by a CAD Administrator, a representative from IT, or others, depending on the scope of the change.

The purpose of implementing technology in a design-related process is usually to automate the process and enforce standards. In this way, workers know what to expect, and the standards will be applied consistently. In product development and manufacturing, the drama should be in the ideas put into the product and its manufacture, not in the documentation process that surrounds it.

There is usually a positive aspect to change for those who are fearful of it. The silver lining is one tool in your toolbox to use to win over people resisting change. For example, you might encounter someone's concerns about people ignoring electronic forms routed by e-mail rather than paper

Part III: Maintaining SolidWorks

copy forms routed to physical in-boxes by pointing out the automatic spell-checking feature, the copy-paste capabilities, the reduced desk clutter, the fewer lost forms, the visibility of the workflow flow chart for everyone to see, the automatic statistics that you can get to evaluate the system, and so on.

If a process or piece of software or hardware is really worth replacing, you already have a convincing argument because you were able to sell management on the idea. When that isn't enough reason, however, maybe the replacement really isn't necessary. If you find yourself using answers like "It's just better," or "Because it's new," or "Yeah, but it is SOOO cool" without much additional substance, you might want to reconsider the change.

Regardless of which end of the change-no-change spectrum you find yourself, you need to consider the concerns of the people at the other end of the spectrum. If you are a proponent of the change whenever something new is available contingent, you need to respect the concerns of the more conservative people who want to see what the supposed benefits of the change are, along with some testing to back it up and maybe reliability testing on local company data. If you are of the "upgrade when SP5 has been out for a month" crowd, you might consider taking a look at the What's New document for the new release to see if there is valuable functionality added to the new version, or look at the release notes for each service pack and see when critical bugs are fixed.

Sometimes you need to keep some of the legacy system available for older work that doesn't need to be brought forward into the new system. If you have a real die-hard proponent of the old system, you might be able to make everyone happy by making that person responsible for maintaining and using the old system when necessary. This gives you an ally who wants to stay on the older system, and someone who is probably proficient at running it. It also gives that employee some peace of mind in connecting to the old way of doing things, even if there is clearly no future in it. You can use this as a bargaining chip to get that individual to at least take training in the new system, and enable her to ease into it slowly.

Summary

Change is not inevitable, and not always necessary or beneficial. Change should happen only when you are able to establish a quantifiable benefit, like bug fixes or new time-saving features or completely new software. Change is often disruptive, especially when it is necessary. It's easy to get addicted to the "cool factor" in change and the latest and greatest features, but in business you need to keep your eye on the dollar signs and rational benefits rather than on emotional benefits.

Some people will always resist change, and the best you can do is to coddle them a little to bring them along with you. This is a benefit, because when you know you are going to get some resistance, you will be more careful to justify changes more thoroughly. Solid decision making and doing your homework thoroughly help you ensure less resistance and better results.

Rolling out New Service Packs

For each release of the software, SolidWorks generally creates five major service packs, and minor “point” releases (such as SP 2.1) as necessary. The initial release of a version such as SolidWorks 2010 is SP 0.0. The service packs are available to all active subscription customers. If a service pack has a major flaw that SolidWorks thinks should be fixed before the next service pack, they issue a point release.

Service packs are usually released approximately every six weeks. These service packs are usually used to fix bugs as listed in the Release Notes that you could find in the Help menu, or in the same area in the SolidWorks Customer Portal from which you download the service packs. From time to time new functionality can show up in service packs, but this is usually reserved for new major releases unless the added functionality is minor.

Many users establish a best practice where they do not install a new version prior to SP 3.0, and some wait all the way to SP 5.0, which should be the last service pack for the version. Depending on your business, this may or may not be something you want to follow. Also, be aware that the five service packs are not guaranteed as a rule, but the last several releases have followed that pattern.

I tend always to install the latest service pack available for two reasons. First, because I write books and give various types of public presentations using the software, I need to be up on the latest and greatest. Second, I work by myself and know that I can usually recover from a bind without losing too much information. The larger your group of users, and the less proficient they are, the later you might consider waiting in the service pack cycle.

Remember that to apply a service pack, you need to have administrative access to the computer. It is also a good idea to exit out of other running

IN THIS CHAPTER

Choosing when to update SolidWorks

Options for installing a service pack

Uninstalling a service pack

software before applying the service pack. It is alright to run most antivirus software running during installation, but check the SolidWorks Web site (at www.solidworks.com/sw/support/AntiVirus_SW.html) to see if the antivirus software you use has any special considerations.

Deciding When or If to Update

Updates to the SolidWorks software come in the form of service packs, which SolidWorks publishes roughly every six weeks for customers on the maintenance subscription plan. In a typical release, SolidWorks issues five service packs, and possibly some unplanned fixes (point releases such as SP5.1). These service packs contain mostly bug fixes. Subscription is optional and includes technical support, access to new versions and service packs that become available during the subscription period, and access to special areas of the SolidWorks Web site such as the Customer Portal.

With SolidWorks, it is very easy to get on the subscription treadmill and just update your software automatically, without questioning why you are doing it or if it is a good idea. When you start out with SolidWorks, it is a good idea to be on subscription to help you get oriented with the software and for the inevitable technical support that you need in getting on your feet. Evaluating your situation every now and then is also a good plan. You may reach a plateau with your need for technical support and new functionality.

If you are in a business with an established way of using the SolidWorks software, and your need for new tools or bug fixes is not great, you might consider avoiding new service packs and initial releases of the SolidWorks software. From time to time new bugs are introduced with service packs, and these bugs are discovered by users.

Best Practice

If you do not need what is offered in a new service pack, don't install the service pack. ■

Too often, the “need” to update is driven by the “coolness” factor of always wanting to have the latest gadgets available. Unfortunately, the coolness factor does not translate into a tangible or measurable value. The cool factor is more about emotional needs than rational business decision making. More often than not, the first people to install brand-new software serve as a warning to the rest of us when they hit a major deficiency in the software.

Prior to the release of a new SolidWorks version such as 2010, SolidWorks does internal testing and opens up a period when users volunteer to find bugs under the beta testing program. Not all of the bugs found in customer beta testing are fixed before the First Customer Ship (FCS, or SP0.0), so these bugs must be fixed in later service packs. SolidWorks has an internal method for rating bugs by the level of customer impact as Low, Medium, High, and Critical, and this rating, along with the number of customers who report a particular problem, helps determine which bugs get fixed first and which possibly do not get fixed at all.

Note

Be aware that starting with SolidWorks 2010, you can no longer maintain parallel service packs of the same release. In the past for troubleshooting purposes, some people would have, for example, SolidWorks 2009 SP 2.0 and SolidWorks 2009 SP 2.1 installed on the same computer at the same time. ■

Calculating return on investment

The return on investment (ROI) is a frequent measure used by management to help determine if a particular expense will pay for itself. It simply involves subtracting all the costs of the implementation from all the projected cost savings of the action in question. Assigning a monetary value to a new service pack is almost impossible to predict accurately. The best you can do is to estimate how much time you will save by using the new software compared to using the current version of the software. For there to be a time savings, there has to be some significant difference between the old and new software, such as new functionality added to the service pack or a bug fix. Stability (absence of crashes or system hang ups) also factors into the ROI calculation.

When you make this calculation, you have to figure into the cost of implementation: the time you take to do the installation; any time that you take to download, set up, and maintain the Administrative Image if you are using one; and checking or troubleshooting.

Making an intuitive decision

The ROI number is the result of a lot of estimation and can be inaccurate. You may be inclined to follow intuition rather than guessed or biased numbers. Intuitively, installing the first release of a new version (SP 0.0) where known bugs exist is a significant risk, based on time potentially wasted by bugs. If you need to be on the latest version, you must be self-sufficient with bug workarounds and not sensitive to time lost due to detecting and tracking down bugs. If you have to do it, you have to do it, but chances are that you don't really have to.

You could wait for SP 5.0, at which point if there are still any bugs, they are not going to be addressed until an entirely new version. Waiting until SP 5.0 is considered a very conservative approach, and you might miss using the improvements made in earlier versions. Waiting until, for example, SP 3.0 enables you to split the difference while still being somewhat conservative. After SP 3.0, most of the more serious bugs from early in development have been fixed. General Rule: The earlier the SP to be loaded or applied, the more time is allocated for testing.

One way to gauge how a particular release is going in terms of usability and stability is to keep an eye on the SolidWorks forums and many of the SolidWorks blogs available. It might take a couple of releases to establish the baseline of complaint noise, but keeping an eye on the community's discussion of the new version is always a useful barometer of the general state of things.

Another way to evaluate if you need a service pack is to read the Release Notes for that particular service pack. You can find the Release Notes from the Customer Portal home page, then under References, select Admin Guides. The first item in the list is SolidWorks Release Notes. Clicking on the link shows you some standard information such as system requirements, important technical alerts, and Release Notes by service pack.

Part III: Maintaining SolidWorks

The only way to be completely sure that a particular service pack will give you good results on your company's data is to have a test installation update, and a separate copy of your SolidWorks data to do the testing on. You should not take the chance of corrupting a file, if it is your only copy.

Testing for service packs should include having two computers side by side, and opening files that are known well in the old version, and comparing them side by side with the same file opened up in the new version.

Note

Because of the new limitation that you cannot install multiple service packs of a single released version on the same computer at the same time, this type of testing will need to involve two computers instead of a single computer, as in the past. ■

Check parts for errors or warnings in the FeatureManager, for volume, surface area, number of bodies (solid and surface), for mate errors in assemblies, and in drawings to make sure that the position of annotations are the same between versions. Ideally, you should report all bugs through your reseller. Bugs cannot get fixed unless SolidWorks knows about them. You cannot assume that someone else has found and reported the bug you have found.

Considering Early Visibility testing

You don't have any way of knowing what the quality of a new service pack or the usefulness of its new features is unless you use it. Other people can't answer that for you, certainly not marketing materials, or What's New documentation. For that reason, I recommend you, as the SolidWorks Administrator, participate in the Early Visibility (EV) program. Any customer on subscription can sign up for this. It amounts to an early look at service packs, like beta testing for the annual major release. EV versions usually come a week or two before final versions. From time to time, the EV version is different from the final version because a customer may have found a bug in EV that SolidWorks was able to fix before the release. When this happens, sometimes you have to uninstall the EV in order to reinstall the final version of a service pack, and even though it is an inconvenience, that's when the EV program is doing its job.

Unlike the incompatibility between SolidWorks major releases, all the service packs within a major release are compatible with one another. Therefore, if you have one workstation at SP 2.1 and another at SP 3.0, there should be no difficulty in sharing files back and forth between the two.

Playing with EV certainly kills your ROI calculation, if you're honest about adding the time to install, research, and test the new version, but if it prevents you from installing a release with nothing new in it for you, it may well be worth your time.

Installing a Service Pack

You have three options when installing SolidWorks service packs: Administrative Image, SolidWorks Installation Manager (SWIM), and manual installation. SolidWorks is designed for you to use the SWIM, which is best suited to individual installs. These techniques are similar to the

techniques to install the software initially. If you use the Administrative Image to install the software, you need to stick with Administrative Image to install service packs. You can use the SWIM and manual methods interchangeably.

Cross-Ref

Chapter 6 deals with all aspects of initial installation in detail. Be sure to read Chapter 6 thoroughly if you have questions about material in this chapter. ■

How you apply a service pack depends on how the original software was installed. If you installed SolidWorks using Administrative Image, you also have to install the service pack using Administrative Image. If you installed SolidWorks from the DVD or download, you cannot change your mind and apply service packs using Administrative Images without uninstalling and reinstalling the entire software. If you want to change methods of installation, you have to go back and uninstall the software, and reinstall it using the new method.

Applying a service pack to software installed from DVD or download

This book has been primarily written from the SolidWorks 2010 beta release, and SolidWorks does not supply service packs for beta software. Almost all the information in this book on applying service packs is derived from the behavior of the SolidWorks 2009 and previous releases, except for some material added at the end of this chapter covering the process of applying service packs to SolidWorks 2010 I received from SolidWorks late in the book writing process.

When you install SolidWorks from DVD, copied DVD data, or downloaded install data, and then want to install a service pack, SolidWorks will want to see the original install media in the original location in order to proceed with the service pack installation. This means that if you originally installed SolidWorks 2009 from a 2009 SP 2.1 DVD, and you are installing SP 4.0, SolidWorks will want to see the 2009 SP 2.1 DVD. The SP 0.0 disk will not work for this purpose. It is OK if the disk is a different disk, as long as the service pack level is the same as the original.

If you have downloaded data and allowed SolidWorks to place the data into a temporary location, you need to manage this carefully. The installation dataset is huge; depending on what you are installing, it is in the range of several gigabytes of data. SolidWorks places this data by default in (using Vista) C:\users\<user name>\Documents\SolidWorks Downloads\. This is on the same drive as the operating system. Frankly, putting the data in a location like that by default is a bad idea. You should put it on a large drive where dedicating a few gigabytes to storing the install files will not cause a problem.

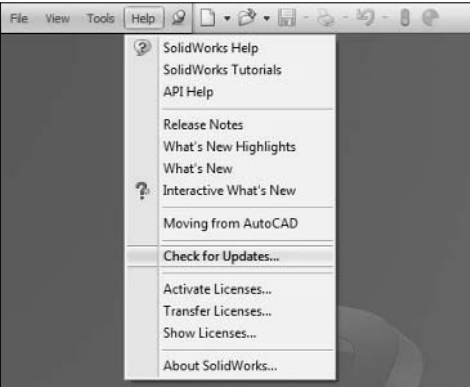
SolidWorks needs the original install files to verify and possibly repair the existing ones.

When you apply a service pack, you do it through the SWIM. You can initialize service pack installations in a couple of ways. First is through the Check for Updates menu selection in the SolidWorks Help menu, shown in Figure 19.1.

Part III: Maintaining SolidWorks

FIGURE 19.1

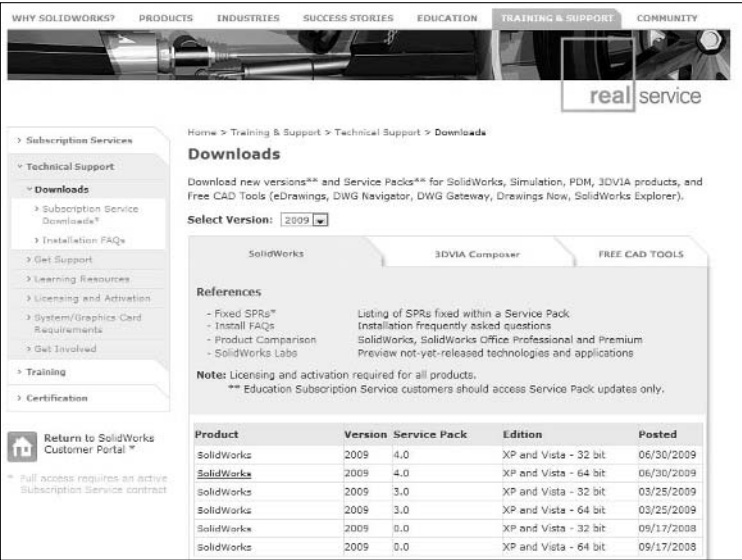
Initiating a service pack update by choosing Help menu → Check for Updates



The other way is to initiate the update by downloading the SWIM application from the SolidWorks Web site, shown in Figure 19.2. This is available through the Customer Portal's Download section, by following the link called Downloads and Updates.

FIGURE 19.2

Initiating an update through the SolidWorks Web site



After clicking through the End User License Agreement (EULA) page, a link gives you the option to download the service pack. However, it does not allow you to download the service pack — only the `sldimdist.exe` file, which is the SolidWorks Installation Manager, or SWIM, application.

Caution

Your browser may display a warning and tell you that the `sldimdist.exe` file could harm your computer because executables can contain viruses. SolidWorks is a trusted source for this type of file, so you can feel confident accepting the file from the SolidWorks Web site. ■

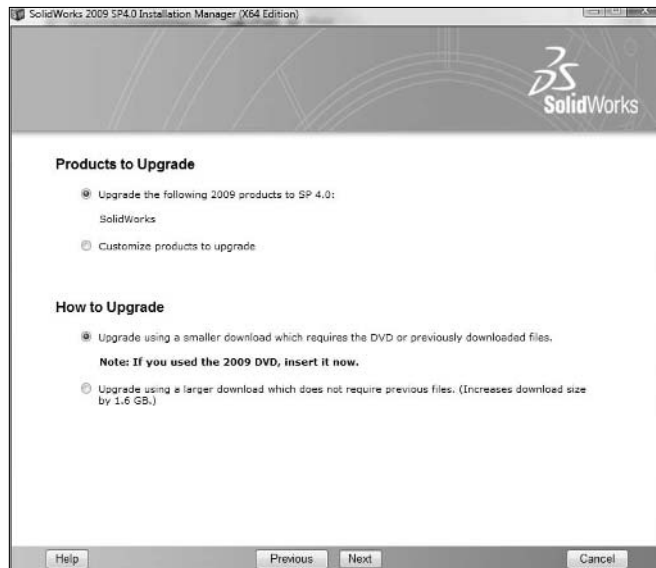
When you run the `sldimdist.exe` file, it extracts zipped files to the location specified and then runs the application. At this point, you will see the familiar opening screen of the SWIM, where you specify your serial number unless it appears automatically. Keep in mind I am describing the 2009 service pack process.

Next it asks you which type of installation, and the choices should be straightforward. Individual install is generally where the user is managing his or her own workstation, or an administrator is managing just a few machines. An administrative install is generally done by the SolidWorks Administrator. The Download Files option is used when one person is administering several machines, but an Administrative Image has not been set up. This is also used when you want to keep an archive of service pack data in case you need to use it again in the future.

Figure 19.3 shows the products that will be updated and enables you to insert the DVD for a smaller download. SolidWorks uses the installation media to verify the existing files and repair any if needed.

FIGURE 19.3

Selecting what and how to update



Part III: Maintaining SolidWorks

Notice that the terminology between the Web site and the SWIM is not consistent as far as “upgrade” and “update” go. I am using the term upgrade to mean moving between major versions and update to mean applying a service pack.

The step, shown in Figure 19.4, enables you to use previously downloaded files to update the service pack. Therefore, if you have several machines and the users update their own machines, you can download to a specific location and allow the users to apply the service packs to their own machines. (If you are doing several machines yourself, I recommend using Administrative Image.)

FIGURE 19.4

Identifying the source of the installation files



In the past, the SWIM has not been very reliable in terms of allowing access to the correct download data, so many users and administrators are in the habit of downloading files manually and bypassing the SWIM. In the most recent releases, the SWIM has shown marked improvement, so I have been using it in the way it was meant to be used. The automation really does help significantly, especially compared to manually downloading and installing.

Figure 19.5 shows the options for download and install. A nice side benefit of using the SWIM is that it estimates the download time and size for you. This estimate is also much better than it used to be. In previous releases, it seemed that the SWIM always severely overestimated the download, so it would often tell you the download would be 3GB when it was actually much less.

The download monitor does its work reliably. The indicator shows the combined download and extraction processes, as shown in Figure 19.6.

FIGURE 19.5

Preparing to download the installation files for the service pack

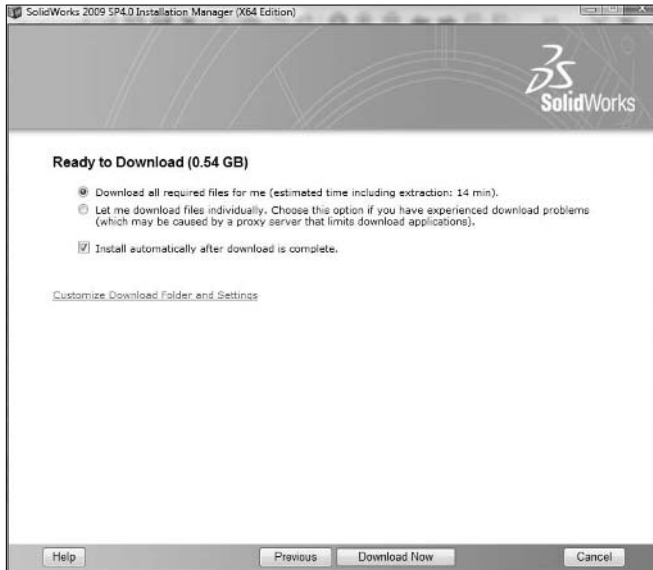
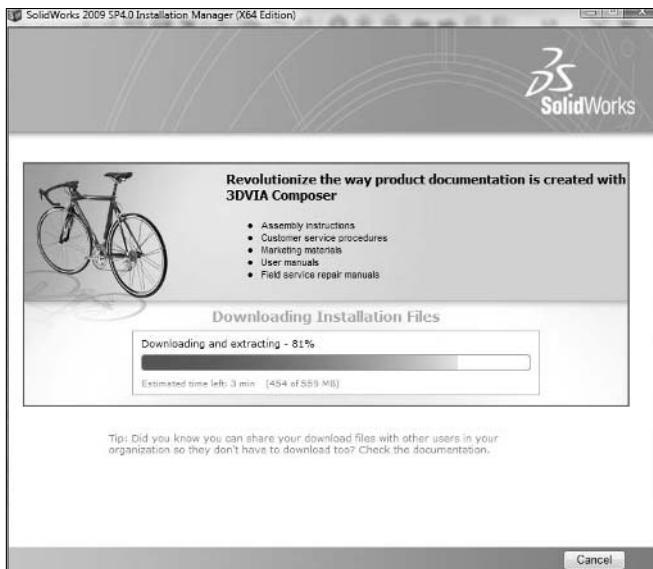


FIGURE 19.6

The download monitor shows the remaining time and file size of the download.



Part III: Maintaining SolidWorks

When the download and extraction have finished, the SWIM installs the downloaded files. Figure 19.7 shows this process in progress.

FIGURE 19.7

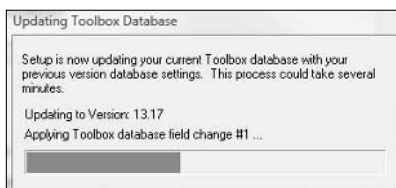
The installation monitor shows progress of the install.



While it is installing, you may see a little dialog box, shown in Figure 19.8, pop up that shows that it is updating your Toolbox installation as well. Upgrading Toolbox between versions and fresh installs of Toolbox require special attention, but updating service packs in Toolbox usually takes care of itself uneventfully.

FIGURE 19.8

Updating the Toolbox database



Cross-Ref

For a complete reference on installing Toolbox, refer to Chapter 10. ■

When the installation is complete, the page shown in Figure 19.9 appears. This page gives you the option to turn on automatic checking for updates (which requires an Internet connection), show the What's New documentation, and possibly even launch an installation survey.

FIGURE 19.9

Choose options after the installation is complete.



The SolidWorks Customer Experience Program is a program in which SolidWorks collects data on how individual users use the software. This includes what types of features or files they use, how frequently they crash, and so on. SolidWorks claims this data is collected anonymously without identifying the user, but they do not make the data user readable so you can see the information your software sends back to the Concord mother ship. This function requires an Internet connection, and it does not ask for permission when it sends the information packets. Follow the link in the page to learn more about the program. Larger installations turn this option off. Besides the implied security risk, uncontrolled network traffic is never a welcome addition.

Applying a service pack to an Administrative Image

This chapter only deals with the Administrative Image when applying service packs, not for the original installation process. Please keep in mind that I am referring to service packs for

Part III: Maintaining SolidWorks

SolidWorks 2009 in this chapter, because at the time of this writing, SolidWorks 2010 is currently in beta and does not have any service packs written against it. Keep in mind also that the install workflow has changed in 2010, and so the installation of service packs is likely to be different.

Cross-Ref

Chapter 6 covers SolidWorks installation using the Administrative Image. Refer to Chapter 6 for general information on the Administrative Image. ■

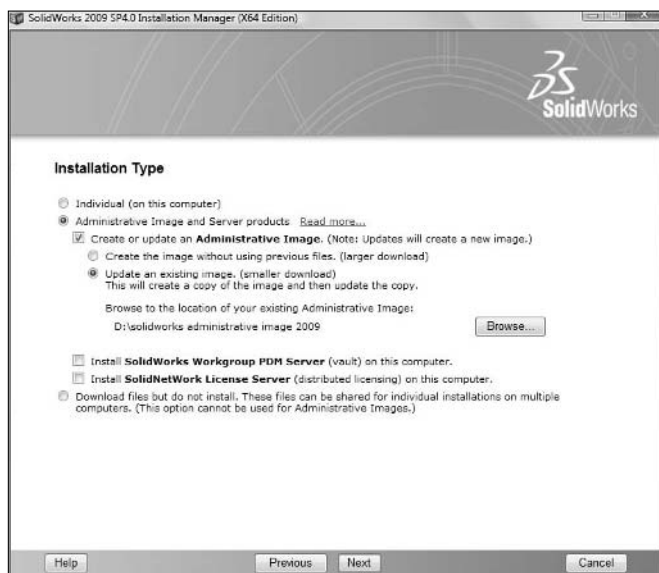
The process of applying a service pack to an Administrative Image is similar to applying it to a DVD installation. You have a few additional options for the Administrative Image. In addition, remember that applying the service pack to the Administrative Image is just the beginning; you (or the user) still must apply the Administrative Image to the user's installation.

When the updated Administrative Image updates the users' computer, it does not just apply a service pack update; it applies the entire release. You can use the updated image both to update existing Administrative Image installs and to create fresh Administrative Image installs at the new service pack level.

You start the update the same way as a normal update, but in the second screen, the screen after you enter the serial number, you specify that you are updating the Administrative Image. You should do this on the same computer where the Administrative Image exists. Figure 19.10 shows this screen of the SWIM.

FIGURE 19.10

Selecting an Administrative Image service pack update

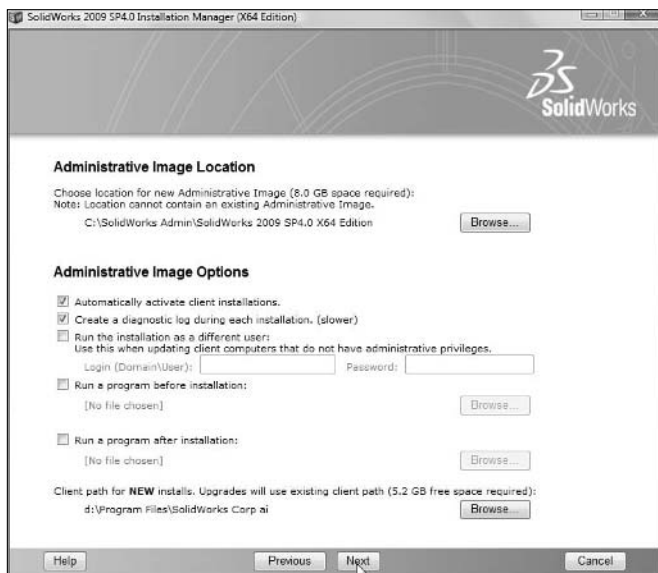


In this case, you specify that you want to update an existing image, but that isn't what happens at all. SWIM copies the original Administrative Image and applies the changes to the copy. In fact, it requires you to create a new name for the new folder later on. If you apply many service packs to Administrative Images, you need to keep an eye on the amount of space they take up. A single Administrative Image of SolidWorks 2009 Premium can take up nearly 6GB.

The next screen enables you to select software or add-ins to update along with SolidWorks. In SolidWorks 2010, this step is absorbed into another step. Refer to Chapter 6 for complete details on the workflow for installing SolidWorks 2010. After selecting the software to update, you have to set the new Administrative Image location and set the options. Figure 19.11 shows this step in the process.

FIGURE 19.11

Administrative Image location and options



If you need to run the installation on the user's computer as some domain user other than the normal user, this is the place to set that up. Remember you need administrator access to install a service pack.

Be aware that by default SolidWorks is setting a directory on the C:\ drive for the new Administrative Image, rather than simply placing it next to the original Administrative Image that you are updating. You may want to consider clearing out old Administrative Images that you do not anticipate using. Keeping these around is expensive disk-space-wise and may not have any value.

Part III: Maintaining SolidWorks

Client Install Options are set in the next screen, shown in Figure 19.12. Remember that the product of this update will be a general purpose Administrative Image — it will be able to install fresh installs or update a service pack from any service pack. That is why the options here include options that you would only use in a complete install.

FIGURE 19.12

Controlling the Client Install Options



Do not forget to use a settings file created by the Copy Settings Wizard to copy SolidWorks System Options from one computer to another. You can access the Copy Settings Wizard through the Start menu, in the SolidWorks flyout menus.

To deploy both service pack updates and major version installations, send the users the StartSWInstall.hta file found in the newly created Administrative Image top-level folder. This updates the users to the service pack that was used to create the Administrative Image as well as installs a fresh, new copy of the software for anyone who does not have the software installed yet.

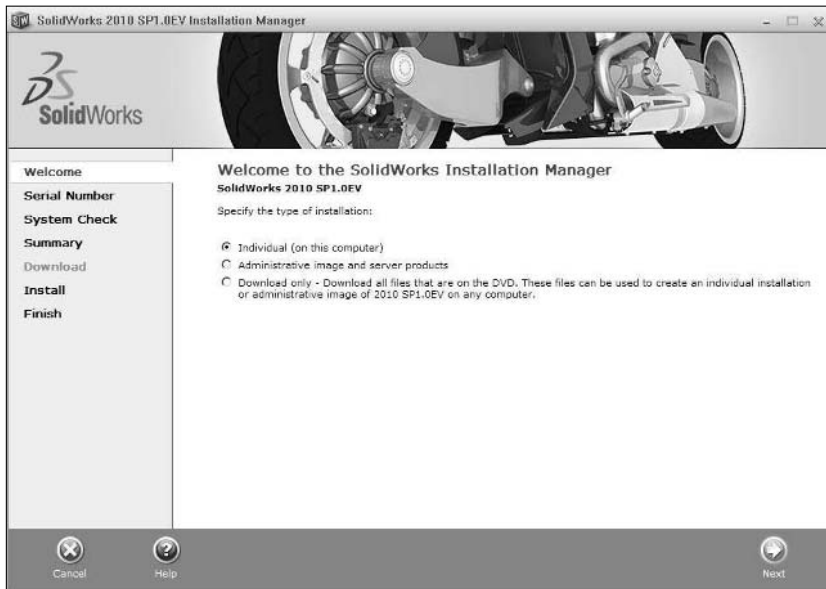
Applying service packs in SolidWorks 2010

Service packs in SolidWorks 2010 should be easier to apply when compared to previous versions. It is a two-step process, where you select the type of installation you need to update, and review the Summary information. You cannot use a service pack to change the installation location for an existing install, but you can change the download location and the products installed.

The steps involved in the 2010 service pack installation are shown in Figures 19.13 and 19.14. This information was provided by SolidWorks Corporation early in their service pack development cycle and late in my book writing cycle. You may see adjustments between these figures and the actual released software.

FIGURE 19.13

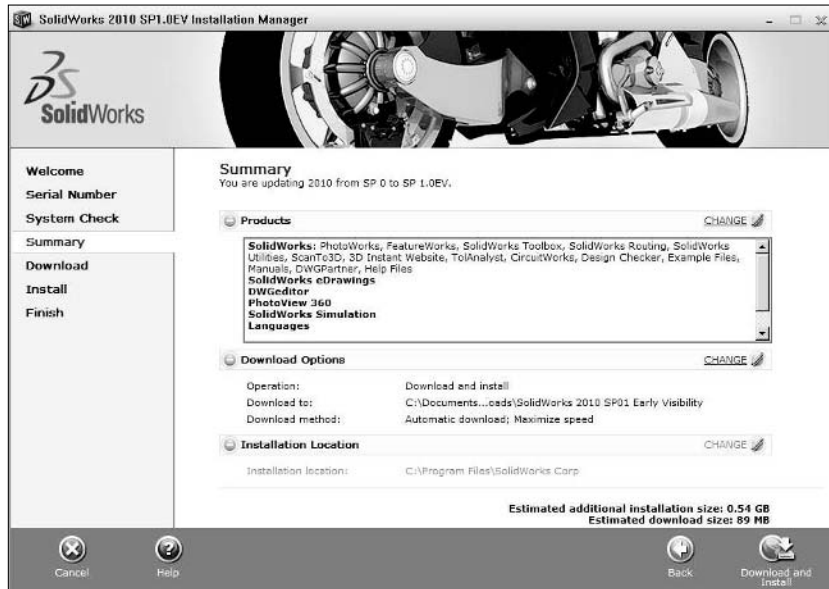
Selecting the type of install to apply the service pack to



These steps have been designed to be as similar to the original installation steps as possible, and to allow a very fast and simple installation for people who use default settings for installing service packs.

FIGURE 19.14

Reviewing the installation Summary information



Rolling Back a Service Pack

Service packs cannot be rolled back in SolidWorks 2009 and 2010. This tool was removed, presumably because the rollback was causing problems within the Windows registry that SolidWorks felt were not time effective to fix. The alternative to rolling back a service pack that was installed but is now not wanted is to uninstall. Unfortunately, uninstallation means uninstalling everything, including the base software install and any applied service packs.

The best way to achieve this is to uninstall the software fully, then reinstall service packs up to the one before the service pack you wanted to roll back. If you installed from an Administrative Image and have kept your old Administrative Image folders, this is very easy: you just send a link from the appropriate service pack folder.

If you installed from DVD or downloaded data, you uninstall the software, then reinstall from DVD, and apply patches on top of the new install. Once you have uninstalled the software, you do not have to reinstall from the same media. For example, after the uninstall, you could change from Administrative Image to DVD or download installation or the other way around. You could also install from a SP 2.1 DVD instead of an SP 0.0 disk.

Summary

In order to have access to SolidWorks service packs, you have to have an active maintenance subscription. Subscription provides you with many things, but you need to evaluate for yourself whether it is providing enough value to justify the cost.

In later versions of SolidWorks, starting with SolidWorks 2009, the SolidWorks Installation Manager works far better than it has in previous releases. It figures out what installation components you need, then downloads, and installs everything automatically. Manual options exist for downloading and installing the various components, but there is no tool to tell you what you need to install on your system to make SolidWorks function optimally. The main advantage of the SWIM is the automatic tool that recognizes what you need to install.

Whether you install from the DVD or an Administrative Image, the SolidWorks Installation Manager simplifies the download and installation of any kind of SolidWorks software.

Rolling out New Versions

Rolling out a new version of SolidWorks to a set of users can be a challenging prospect, depending on factors such as the new functionality in the release and how your company uses the software. Some releases of SolidWorks can be very disruptive to a company's workflow, and some may be relatively painless for users to adopt. For example, the jump from SolidWorks 2007 to 2008 was, and continues to be, difficult for companies to cope with. The interface changed drastically in several areas, and some important features were added at the same time. The jump from SolidWorks 2008 and 2009 was less drastic, however, but finished several interface changes that had begun in 2008 but that were not completed until 2009. The jump from 2009 to 2010 promises to be relatively easy again, with small changes to a wide range of the software, but nothing too drastic in the interface.

When interface changes are forced on users, the change is more likely to be disruptive. When discussing CAD technology, you can find many people who are simply proponents of *change* without much regard for what the content of the change is. They just like to see things that are new and different. When you are charged with running a CAD department for a company that is in business to make money in order to keep employees' paychecks coming, sometimes your priorities must be somewhat different. It is not a question of old people resisting change or cool people having all the newest toys, it is about productivity and return on investment. If you make your engineering tool selection based on anything else, you are doing yourself and your company a disservice.

IN THIS CHAPTER

Choosing when to upgrade

Reinstalling a fresh copy of SolidWorks

Deciding When to Upgrade

One of the first things I want to put to rest here is the difference between the words *upgrade* and *update*. Service packs are used to update your computer, but an upgrade only happens when you change versions. SolidWorks uses the terms interchangeably, especially in the Installation Manager interface. In this chapter, I cover upgrading the software between major versions.

Cross-ref

Software updates from one service pack to another are covered in Chapter 19. ■

Reasons to upgrade between versions are easier to identify than reasons to update between service packs. SolidWorks does put new functionality in a service pack release from time to time, but more frequently, it saves the major changes for the new major releases. You can find several sources of information that will help you decide if a particular release is worth upgrading to or if you might decide to skip it.

Weighing software maintenance costs

Built in to any discussion about upgrading to a new release is a discussion about money. SolidWorks typically puts out a new major release every year, which not so coincidentally corresponds with their billing cycle for subscription or maintenance costs. SolidWorks charges a set yearly fee for each type of license. You get technical support from the reseller, access to all SolidWorks Web information, and all the service packs and new releases that are released during the period of your subscription.

Because SolidWorks has tied together the tech support and the new versions/service packs, if you let your subscription lapse, you give it all up. SolidWorks does not make it easy to separate the reseller tech support from the SolidWorks Corp. supplied upgrades and updates. In addition, not surprisingly, SolidWorks does not make it easy to get off and then back onto subscription. They assess a per-license penalty for getting back onto subscription after allowing it to lapse more than a certain amount of time. Your reseller should be able to provide details on these issues.

If you take a serious look at your situation and determine that the money you pay for subscription is not providing you with value, it may be time to make a change. Because SolidWorks does not provide alternatives, I have some suggestions to help you achieve your goals, taking a couple of different possible scenarios into account.

1. Your users are mature SolidWorks users, and you don't typically benefit from new features in the software.
2. Your users are mature and self-supporting, but they like the new functionality with each new release or your company is looking for closure on a couple of major bugs.
3. Your users require some level of support that you cannot provide, but you do not really need the new functionality of each new release.
4. Your users require support that you cannot provide, but you need the new functionality provided with new releases.

To clarify the subscription scenarios further, I have suggested

- If you find yourself in the first situation, then you probably could just drop the subscription and your users might not notice the difference. For those times when you require support and you cannot provide it yourself or you are not available, consider getting your users keyed into various forums, such as the SolidWorks Forums (<http://forum.solidworks.com>) or Eng-Tips (www.eng-tips.com/threadminder.cfm?pid=559). Individuals on both sites are eager to assist people with modeling or administration problems, although between them, the SolidWorks forums are better attended and have a higher concentration of more reliable expertise.
- In the second situation, again, your users might need occasional support, but the new versions are the main benefit to them. SolidWorks does have a policy of allowing customers to buy an upgrade at a price somewhat reduced from full support. They do not publicize this option much, and it is likely priced such that it is marginally better than the full support, but again, you need to check with your reseller for details on this program. In the end, it may make the most sense to pay for subscription.
- The third situation presents some options. Your main need in this situation is for training and low-level tech support. In this situation, it is probably best to use an internal resource for your tech support, depending on how many licenses your company has. If the number of licenses is well in excess of 50 licenses or so, it may make financial sense to hire a high-level SolidWorks guru who can handle all the support necessary internally.

The main drawback in handling technical support internally is that you do not have access to the SolidWorks Customer Portal, from which you can access bug information and the knowledge base. It would be useful if SolidWorks would provide courtesy access for large installations or if you were able to negotiate having a single license on subscription for testing purposes. SolidWorks and resellers are reluctant to support a single license like this, but with some negotiation skills (and a firm hand), you should be able to negotiate something for your benefit. A large installation that is not on subscription represents a significant missed opportunity for resellers, so be prepared for the hard sell.

- In the fourth situation, it seems clear that you need the subscription service. When you have a large installation on subscription, you need to have a single individual responsible for the reseller contact and managing the support as a whole. This is not to say that you should be the only person contacting the reseller support, but when your users contact them, you should be aware of this, and also what kinds of issues the SolidWorks users at your company are having. Most resellers now provide some type of online customer support database that enables administrators to get an overview of the support their company is receiving.

SolidWorks subscription is expensive, but it has not increased in price since the early days of the software in 1995. The base level of SolidWorks, now called SolidWorks Standard, has not increased in price either. Price increases have only come in the form of additional functionality for the higher levels of software, including Office (no longer available), Professional, and Premium. Overall, you cannot argue with the value of SolidWorks pricing, but in these days of decreasing CAD pricing, it is not unreasonable to expect SolidWorks to remain in step with other providers, such as Alibre, SpaceClaim, Rhino, Moment of Inspiration, Shark!, and a number of other players

who are providing capable geometry creation and documentation tools for very reasonable prices. I'm not endorsing any alternative software, I'm just advocating the point of view that you should buy software that fits your needs, and there is no need to be blind to other suppliers.

SolidWorks is a highly generalized 3D CAD tool, which can be great if that is what you need. If you really don't need all the generalization (sheet metal, weldments, surfacing, molds, and so on), then a smaller tool may fit your needs (and your budget) better.

Evaluating a new version

When SolidWorks has a new version of their software available, take a look at it as soon as your workload permits. If functionality in the new version would help your users, you owe it to them to check it out and implement it when it is ready.

The earliest opportunity for most users to evaluate a new SolidWorks version comes during beta testing. This is available to all subscription customers, and is run as a contest, although you can just use the software for your own testing without joining the competition. A non-disclosure agreement (NDA) is required to participate, which just means that you can use the software but are not allowed to talk about it with others outside of the terms allowed for in the NDA itself. You need to refer to the SolidWorks beta Web site for details on this program.

If you are interested in the contest, SolidWorks support people assign points based on how many bugs that you report they are able to verify. Points are also assigned for a variety of other things such as turning in reports from automated tools such as the Conversion Wizard. Prizes are given to the top scorers for each of usually three beta periods for base SolidWorks, as well as other products such as Simulation (Cosmos) or Enterprise PDM (Conisio).

When testing a new version of the software, you should be looking at several different aspects to gauge how much work it would take to implement the new version. Here are a few things you should be looking at:

- Do your old files convert flawlessly to the new version?
- How much difference is in the interface (important for training)?
- What new functionality that your company needs is included in this version?
- Are there new hardware requirements, in particular, video?
- Are there important bug fixes that you have been waiting for?
- Will all your add-ins be compatible with this new version of SolidWorks?
- How much implementation time will you need for upgrades such as libraries, Product Data Management (PDM), Enterprise Resource Planning (ERP), Finite Element Analysis (FEA), Computer Aided Manufacturing (CAM), Coordinate Measuring Machine (CMM), and/or Toolbox?
- Does the new version have new distribution capabilities, such as improved Administrative Image or Active Directory support?

You can answer most, but not all, of these issues in beta testing. You will not find answer about how to install service packs, for example, until a month or two into the regular release. You can use the Task Scheduler to convert files to the newer version automatically (make sure to do this on copied data). You need to do most of the other checking manually.

After you have checked through the new software and determined to go forward, make a list of all the new functionality, possibly using the What's New documentation. Be aware, though, that the What's New documentation is generally not completely thorough or very detailed. It is still the best assistance SolidWorks provides for planning the scope of a training class for the new version.

Upgrade training is one area where your reseller might be willing to help you for no cost, probably assuming you are on maintenance. Many resellers conduct four-hour What's New sessions around the time of a new release. While this is not typically what you would consider a thorough hands-on training session, it is certainly worthwhile to take advantage of this opportunity if it is available to you and your SolidWorks users. After a reseller session like this, some issues will require some clarification and some changes. Specifically, when the Custom Property Tab Builder interface is introduced, the reseller's presentation simply shows its basic functions, so you will have to demonstrate to your users in an in-house session the way you plan to implement it, and what they are required to do as end users.

Depending on the size of your operation, you might consider enlisting the help of a "steering committee" to either help or act as a sounding board for implementation ideas. If you, as Administrator, are rather junior in the CAD department, it isn't a bad idea to try to build a consensus. If you are a senior employee, you may already have the influence to push ideas through. Also, remember some earlier comments in Chapter 18 about enlisting the help of senior management to add backing to bigger ideas.

Upgrading as an option

It is possible that after an evaluation, you decide not to implement a new version. It might not have enough changes that matter to your way of working. You need to make good decisions for your company, not for the good of your CAD vendor.

For example, several SolidWorks using companies, including those paying maintenance, chose to skip the SolidWorks 2008 release because it was deemed too disruptive, and further, changes due in the 2009 release covering some of the same territory would again disrupt users workflow. Most of these users, that I am aware of, have caught back up with the 2009 release because the interface turmoil has settled down a little. SolidWorks relented on some of the more radical changes and finished what 2008 started in other cases.

Not upgrading while you are still paying maintenance is a very painful decision, and you need to make a good business case for paying the subscription money without using the upgrades. SolidWorks retires support for old versions, although in 2009, they announced they would extend support for old versions of software beyond the previous retirement schedule. This gives users who need support but not new versions some incentive to keep paying maintenance.

Part III: Maintaining SolidWorks

There is certainly no one answer to the debate over whether you should drop maintenance or continue paying for it. Each company has a different situation, and each needs to make its own decisions based on circumstances.

Taking into account third-party issues

Sometimes companies can be put in a bad position by third-party software. I have seen several companies are stuck on a version of SolidWorks because they had a large investment in customized implementations of very expensive PDM systems. Every company has to make its own decisions, but to me, this puts the cart before the horse. If the money is already spent on the custom PDM implementation, and upgrading the implementation to work with the new version of SolidWorks is too costly or too time intensive, you are presented with some difficult decisions. Do you simplify the PDM implementation to the point where it can be used with the new version of SolidWorks, or do you skip the SolidWorks new versions? Add to the financial balance that you may also be paying subscription fees for both PDM and SolidWorks.

It might also just serve as a lesson for companies to find a PDM system with as standard an implementation as they can live with, primarily due to the question of if it will remain compatible with the SolidWorks software new releases.

Implementing a new version

After the new version training has been planned, you need to plan what type of rollout to perform. The type of rollout depends on several things, including what type, or the length, of projects your people do, if there are several projects or just one going on at any given point in time, if you have customers that require certain projects be delivered in a particular version, and other factors. Methods you can use to roll out an implementation of a new version of the software are as follows.

Going Cold Turkey

In the Cold Turkey method, everyone changes all at once, and immediately. This has some advantages and some disadvantages. The main advantage is that you never run into a stalled implementation, because it all happens at once. You flip the switch, and everyone works in the new version from this point forward. Because an implementation requires discipline and hard choices, sometimes something abrupt is the only way to go, but not all situations or temperaments can handle this type of change. This type of change may be more feasible for service packs than for entire new versions, mainly due to version compatibility issues and potential interface training issues.

Phasing in by group

Phased implementations may or may not be more successful, depending on how your company is organized. If you have different groups of users that use the software on independent projects, and do not share information, you may be able to use one group as a pilot, and then phase the new software into production use with subsequent groups. This approach is probably more in line with how a large organization has to work. If you have several groups with 20 or more users in each group, installing everyone at once is probably not even possible, much less a good idea. By breaking

the installation work into smaller chunks, you also break the troubleshooting work into smaller chunks. In addition, a pilot group prepares you for what you might need to plan for with the rest of the groups.

Phasing in by project

If you have several projects going on simultaneously, and users do not widely overlap on projects, phasing in new software by project may be a good idea. Sometimes it is just impossible to take any more time on non-productive work. Making a change to the software used in a project in its early stages is certainly a lot easier than making a change in its later stages. This type of rollout depends heavily on the length of the active projects when you start the rollout, or the timing of the project, such as whether it is early or late phase.

One thing to consider in all these types of rollouts is that you might want to have a hand-picked group be the first ones to go through using the new software and new procedures in production. Having your strongest users, or maybe your most flexible users, succeed can be an encouragement to the others, so there is less fear about moving to the new version, and everybody can see how the project was handled.

Failed implementations

Failed implementations of software are something you need to keep a wary eye for, especially if you have users who are very resistant to the change. Failed implementations happen when you cannot reach your goals. For example, if instead of completing a change over to a new version, you have some change over and others not changed over, and there doesn't seem to be any resolution to the reason for the lack of progress, it could be considered a failed implementation.

If you get part way down the implementation path and realize that some difficulty, such as the new version no longer shares data correctly with a crucial third-party application such as Computer Numerical Control (CNC) machining software, it can put the brakes on your implementation. If you bring this error to SolidWorks's attention, they can classify it in several ways. The best way for you as a SolidWorks customer is for them to call it a regression, which means that it is something that used to work, but due to a bug in SolidWorks, no longer works properly.

Another possible classification is that SolidWorks says it is the responsibility of the third-party software provider. In this situation, you have to negotiate with the third-party provider, which may or may not be productive.

Be aware that both companies may wind up pointing fingers at one another or at you, saying that the fix is someone else's responsibility.

Remember that delaying a rollout for too long after the implementation planning and training can lead to a rollout that never really happens. In a situation where there is just no good time to make a change, but it has been determined that the change will be useful and beneficial, the best thing to do is to jump into the change. If all times are equally bad, there is no time like the present, to use an unfortunate mix of metaphors.

Training

You need to install the software before the training takes place. The software needs to be in place before or immediately after training, depending on if you have to use the user's machines to do the training on, or if it will be a hands-on training session. I have seen several instances where training money and time were wasted by companies because of poor scheduling,

After training, users need to jump right in to using the software or they will forget what they learned in the training. If they go back to using whatever software they were using before, they will very quickly forget the new training.

Installing

Installation of a new upgraded version is just like the main installation of the software described in Chapter 6, but you should have all the variables worked out. In previous versions of the software, you used to get a choice to upgrade the existing install or to make a parallel install. This is no longer an option; the only way you can now do it is to make a parallel install, but remember that starting with SolidWorks 2010, SolidWorks does not support multiple installations of the same SolidWorks version at different service packs.

Parallel installs of two versions at once, say, SolidWorks 2009 and 2010, do function. Problems from time to time do crop up, but they are usually in the form of registry problems, where one version may change a setting somewhere in the other version. Rarer are crashes where performing a command with both versions open causes one version to crash. These are the exceptions rather than the norm; in general, I always have multiple versions installed, and at times have two versions running at once to research differences.

When installing, the things you need to be careful of are your Toolbox location settings and general library settings. The Toolbox settings have to be carefully monitored during installation.

Working with upgraded installations where Toolbox already exists

A default installation of Toolbox on a parallel install installs a completely fresh database and library, essentially setting your new installation back to zero. This is not something most users find useful. If you install the new version of SolidWorks and only upgrade the existing Toolbox, users of the previous version will not be able to access the database (they get a wrong or unexpected version error), or any Toolbox parts saved with the new version (they get a future version error). This is probably not a situation most users would find useful either.

If you are installing Toolbox and want to still use both versions of Toolbox, here is what you need to do:

1. **Copy your existing toolbox folder to a new location, possibly named with the new version year.**
2. **Perform the installation of the new version of SolidWorks, and point to the location of the copied Toolbox.** SolidWorks upgrades the database and attaches the copied library with all its existing configurations to the new version.

It is important that you understand the consequences of just following the default installation, especially for Toolbox functionality.

Other libraries

If you have other libraries that you want to bring forward from one release to another, you need to understand that saving library items in the new version may prevent previous versions from being able to read the files. This certainly happens for library parts and templates. It is possible that some file types, such as blocks or linestyles, will not be subject to the non-compatibility between versions.

Getting a Fresh Install

You may run into a situation where your installation of SolidWorks has become “corrupted.” The word corrupted is often misused simply to represent any error you don’t understand or can’t solve. Corruption describes a specific kind of error when you get the wrong files, wrong versions of files, or files themselves that have been corrupted by file transfer errors, tampering, or other means. When you get an installation subject to this kind of error, fixing it is sometimes very difficult. It is often a better solution to simply reinstall the software. Reinstalling works, it is predictable, and you know approximately how much time it is going to take. Troubleshooting an unknown corruption could take literally weeks to find, and in the end, the best solution might be to do a fresh install anyway.

When I am facing a problem with SolidWorks that I don’t understand, such as repeated crashing, and I know it doesn’t happen on other computers, or other strange, unexplainable errors that seem restricted to the local machine rather than a systematic problem with the software, I often simply wipe the computer as clean as I can, uninstall SolidWorks, wipe the SolidWorks registry clean, restart the computer, and reinstall the software as a fresh copy. I cover this process in more detail to help you understand the decision-making process.

Caution

Tinkering in the Windows registry can lead to problems that can require reinstalling software or Windows itself, or even to hardware failure. Before editing anything in the registry, it is wise to employ the assistance of a professional. ■

Taking care of activations

When you are installing and uninstalling the software, you may run into issues with license activations. When you uninstall SolidWorks, if that license is activated, SolidWorks tells you that it has detected an activated license, and asks you if you want to proceed. If your intention is to reinstall the software on the same computer, you can proceed without deactivating. The license is still activated on the SolidWorks server even if you don’t have the software installed on your computer.

If your intention is to uninstall the software on one computer and reinstall it on another, you need to go through with the deactivation. It is best to perform deactivation while the software is still installed, because the software needed to do the activation/deactivation is already on the computer.

Part III: Maintaining SolidWorks

For your purposes here, I assume that you are going to reinstall the software on the same computer, and want to leave the activation as it is.

Caution

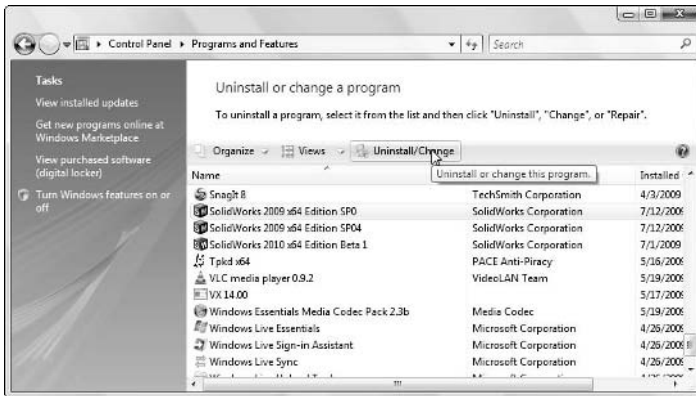
SolidWorks limits the number of computers from which you can activate the software, even if they are not activated simultaneously. For example, each serial number is allowed two simultaneous activations, one from work and the other from a home installation. However, you are only allowed to activate a license from four different machines. You might run into this if you have a license over a period of time and often buy new computers. The activations are identified in part by the name of the computer on which the software is activated. If you run up against this limit, contact your SolidWorks reseller, and they should be able to help you by resetting the accumulated machine names back to only your currently activated machine(s). ■

Uninstalling SolidWorks

Uninstalling SolidWorks is the same as uninstalling any other software. In the Control Panel, go to Add/Remove Programs (in XP) or Programs and Features (in Vista), select the program to uninstall, and click the Uninstall link at the top. Figure 20.1 shows this part of the uninstall interface.

FIGURE 20.1

Beginning to uninstall SolidWorks



From the Uninstall or change a program interface, the SolidWorks Installation Manager starts up and enables you to uninstall selected items from a list, shown in Figure 20.2.

Use this list to uninstall whatever software you want to reinstall.

Note

If you want to get a clean install, and intend on wiping the SolidWorks registry, you need to delete all the software installation files. If you wipe the registry without uninstalling everything, you still need to do a complete reinstall of all the software anyway, because Windows will have forgotten that it ever was installed. ■

FIGURE 20.2

Select SolidWorks software to uninstall through the SWIM.



When SolidWorks is done uninstalling, it leaves a few things behind. Some folders are left on your hard drive, and most of the SolidWorks registry is left intact. The reason for this is so that when or if you reinstall the software, all your settings are waiting for you. In some cases, those settings may be the ones causing trouble, and you may not want them there at all. To remove them, you have to do it manually from the Windows registry.

If you were running a 30-day courtesy license, a simple uninstall and reinstall does not reset the 30-day clock.

Wiping the SolidWorks registry

Settings like the System Options are specific for each user, and, therefore, are stored in an area of the registry called `Current_User`; but some settings are set for the entire machine, and they are in an area called `Local_Machine`. The most important of the information that goes into this area is the SolidWorks serial number. You may have noticed that you can only install a single SolidWorks serial number on each computer. For this reason, you cannot mix educational and commercial licenses on a single computer.

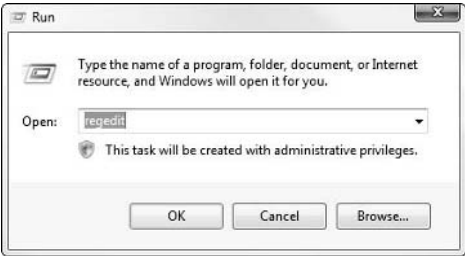
If you are handing off the computer to someone else, selling it, or donating it, you may want to remove information like your serial number, because it could be used to access information from the SolidWorks Web site or even download and install the software using your license.

Part III: Maintaining SolidWorks

To access the Windows Registry Editor, choose Start ⇨ Run, and type **regedit**, as shown in Figure 20.3.

FIGURE 20.3

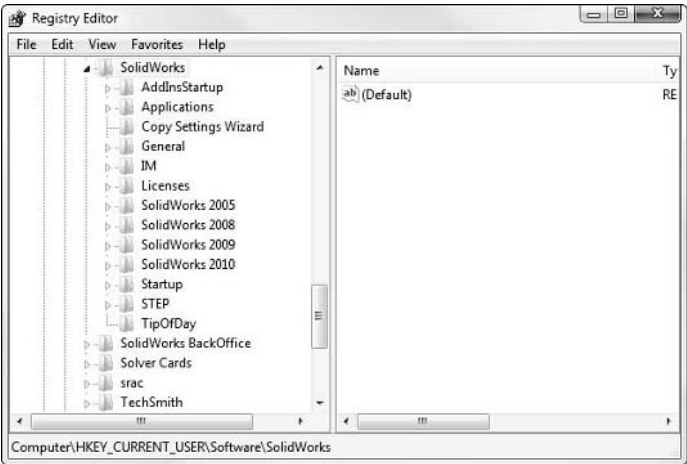
Accessing the Registry Editor



The Registry Editor looks and works like Windows Explorer. The SolidWorks folder of the Current_User section is shown in Figure 20.4, and the complete path is shown in the task bar at the bottom.

FIGURE 20.4

The SolidWorks folder in the Windows Registry Editor



Notice that some additional SolidWorks items are listed below the SolidWorks main entry. SolidWorks BackOffice is generally things like the Task Scheduler. To remove SolidWorks

completely, this needs to go as well. In addition, srac is the name of the company that started Cosmos that became SolidWorks Simulation, so it is related to the SolidWorks installation, as well. You may want to look around the registry a little to make sure you delete everything you can. If you are in doubt about a folder that you find, don't get too carried away. Getting back things that you deleted is much more difficult than simply deleting more items.

Caution

You should back up your registry and your computer before editing the registry. Mistakes in the registry are not easy to undo. ■

If you are trying to clear out a SolidWorks installation, delete (or simply rename with a suffix such as OLD) the entire HKEY_CURRENT_USER\Software\SolidWorks key. When that is complete, do the same for the HKEY_LOCAL_MACHINE\SOFTWARE\SolidWorks and the SolidWorks Corporation and SRAC keys. These deleted the licenses and installed add-ins for SolidWorks that affect the entire machine.

Once the uninstall has been performed and these registry areas have been deleted or renamed, you should be ready for a fresh, new installation.

Using Ccleaner

Whenever I am trying to clean up maintenance issues on my own computers, I use the free utility Ccleaner. Ccleaner takes care of most of the common maintenance things SolidWorks users, as well as other heavy computer users, should take care of periodically. Ccleaner is available from www.ccleaner.com/. It can be used in many ways, but I typically use it to clear out temporary files and remove clutter from the Windows registry.

I mention Ccleaner here because after uninstalling and manually editing the registry, running Ccleaner is never a bad idea to put things right.

Summary

Installing a new version of the SolidWorks software is not as simple as just putting a DVD in a drive and installing the software. It requires thought, planning, and some evaluation before you implement a new version (plan for how it will be used), and then roll it out to users (train them and give them projects to do). When making decisions about how to spend your company's money, you should have good, solid reasons for whatever you do, whether it is to upgrade or not to upgrade.

When you decide to upgrade, you also need to plan how the implementation takes place. You can use various methods, or even mix methods, to get something that works with your needs and your schedule.

Part IV

SolidWorks Workgroup PDM

Workgroup PDM simplifies all of the manual file management tasks that commonly trip up users and administrators. Part IV helps you understand what Workgroup PDM does, how to install it, and most importantly, how to configure it to do what you need it to do.

IN THIS PART

Chapter 21

Introducing SolidWorks
Workgroup PDM

Chapter 22

Installing and Configuring the
Vault

Chapter 23

Establishing Permissions and
Managing Toolbox

Chapter 24

Establishing Client Best Practice

Introducing SolidWorks Workgroup PDM

SolidWorks Workgroup PDM is the product formerly known as PDMWorks by DesignSource. SolidWorks purchased the software several years ago as an add-in, easy-to-use file management system. The administration overhead on Workgroup PDM is low and it is reasonably flexible; but the main intent for this software was that it be easy to use and easy to administer, and available with the Professional and Premium levels of the SolidWorks software.

SolidWorks Corporation and its parent Dassault Systemmes offer additional enterprise level PDM software named SolidWorks Enterprise PDM and SmarTeam, respectively. These are available for additional cost and are not bundled in to any of the SolidWorks software bundles. This book does not cover either of these products.

PDM stands for Product Data Management. For most purposes, this simply means file management with revision management capabilities. The reason it is so important is that with SolidWorks, and in fact with any CAD system that uses relationships between distinctly separate files, maintaining the relationships manually is difficult but critical. PDM software applies most of the discipline you would use in manual file management, while removing the requirement for you to remember to do the right thing. The software ensures that you do the right thing; it remembers for you.

IN THIS CHAPTER

**Making Product Data
Management work for you**

Executing Workgroup PDM

**Getting ready for vault and
client maintenance**

You Need Product Data Management

Throughout this book, I spend time talking about how difficult some task is when done manually, and how most of that difficulty disappears when you add a PDM software to the solution, where it is taken care of automatically. I find myself saying this frequently to people who ask many different SolidWorks Administration questions.

Does PDM complicate things? Other than adding the obvious task of installing, configuring, and managing the PDM application, it doesn't complicate things. If you are already managing files manually, making the switch to using a PDM requires some initial organization, a little thought, and some time to execute the implementation, but overall you will find your work simplified.

One of the best things about SolidWorks Workgroup PDM is its simplicity. Workgroup is not based on a database, and therefore doesn't require an administrator with database admin skills. Most SolidWorks users with basic IT skill levels have the skills to administer Workgroup PDM. You do need a firm grasp on manual file management techniques, but the product was developed to be installed, implemented, and administered inside a small workgroup of SolidWorks users without outside intervention.

Some people believe that the lack of a database presents some undesirable issues for Workgroup PDM, such as speed or size limitations. Those criticisms may have some merit, but if you are concerned about that, the upgrade path to SolidWorks Enterprise PDM for you is clear and easy. Actually, by utilizing Workgroup PDM, you will be setting the stage for moving to any other PDM product. Workgroup PDM is a proven application of technology, and when used by its intended target audience for its intended purpose, it is a fantastic tool.

When you hear criticisms without detailed explanations, you should approach the issue with a little skepticism. Oftentimes people misunderstand what is happening, which clouds their judgment. Criticisms that you may hear about Workgroup PDM are the lack of a real database, the inability to handle very large vaults, and a perceived lack of support for configurations. There is a grain of truth to each of these criticisms, but also a flip side of the issue. The configurations argument, for example, claims that Workgroup PDM does not support configurations. It's true that Workgroup PDM does not treat configurations as separate documents, but it allows for configurations and searches configuration specific custom properties. As long as you are not trying to assign revisions to SolidWorks configurations, Workgroup PDM configuration support should be sufficient for you.

In some situations, however, Workgroup PDM is not the recommended tool. If you work with remote sites and require database replication for access from multiple locations, you should move on to Enterprise PDM. Workgroup PDM is designed for a local group with fewer than 15 users. Vault size can be an issue for Workgroup PDM depending on your archival practices (it should be fewer than 200,000 documents), but there are ways to manage the size of the vault.

PDM is not the solution to everything

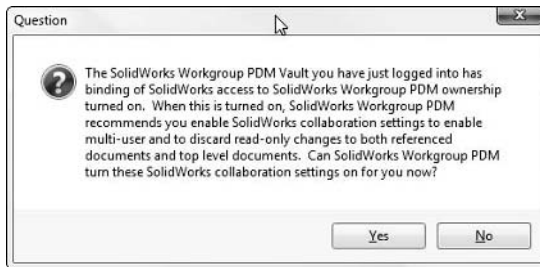
You can run into file management problems even if you are running PDM, but these are typically limited to file management errors on your local computer, outside of the PDM application.

Chapter 21: Introducing SolidWorks Workgroup PDM

In addition to file and revision management, PDM also takes care of most of the collaboration difficulties discussed in Chapter 17, and it adds very valuable search and reporting capabilities that SolidWorks alone does not have. In fact, one of the settings for the vault enables Workgroup PDM to work with the SolidWorks Collaboration settings. When you start a Workgroup PDM vault with this setting active, if your SolidWorks Collaboration settings are not also active, Workgroup PDM prompts you to change the settings. The prompt is shown in Figure 21.1.

FIGURE 21.1

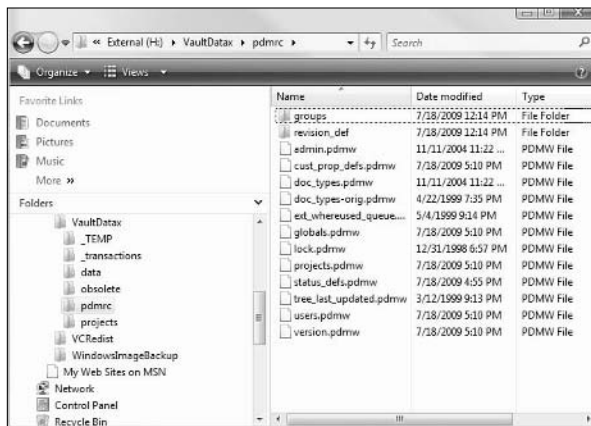
Workgroup PDM prompts for SolidWorks Collaboration settings



For such a low cost (approximately \$500 per user), it is worth it for everyone to use a simple tool like SolidWorks Workgroup PDM. It is less trouble than Toolbox to install, configure, and maintain, and it helps you avoid some of the most difficult entanglements described in this book: file management, revision management, file sharing, lost data, and collaboration in multi-user environments. It offers a structured way to work, along with a reasonable level of flexibility. Figure 21.2 shows some of the flat file structure of a fresh Workgroup PDM installation. The *.pdmw files are simply text files, and grow to hold all the information for the vault.

FIGURE 21.2

An example of the flat file structure of the Workgroup PDM vault



Discovering how PDM works

Most PDM applications work on the same concept. A “vault” (essentially a large protected folder) is used to store documents that are being managed. Usually the documents are renamed in some way so that the PDM application knows which file is actually the document you are trying to manage, and the relationship between the encrypted name and the actual data on the hard drive is managed through a database or flat text file in the case of Workgroup PDM.

When one user is using a file, all other users are locked out from making changes. Changed documents are put in the vault under a different name, and through the database related to the document and revision level. The database keeps track of which user has control of which files. You can also make relationships between files like Excel or image files that can’t be related in Windows Explorer.

One of the key features in most PDM systems is “metadata,” which is essentially text data used to describe or identify the file in some way. SolidWorks files contain their own metadata in the form of custom properties. PDM systems can read these properties and include them in the database that connects all of the metadata with the actual file data in the vault.

As I explain how Workgroup PDM works, I assume a group of users is all accessing the same files. This is because the more people that are accessing your data, the greater the need for Workgroup PDM. Single users can and do use Workgroup PDM primarily to manage revisions and for its search capabilities, but the advantages are greater for larger groups of users.

The vault

Workgroup PDM works by moving managed files to a central “vault.” The vault is just a folder on a network location where users do not have direct write access. You never browse the vault directly using Windows Explorer; you always access it through a Workgroup PDM client, which is a viewer application that acts as a go-between between you and the encrypted information in the vault.

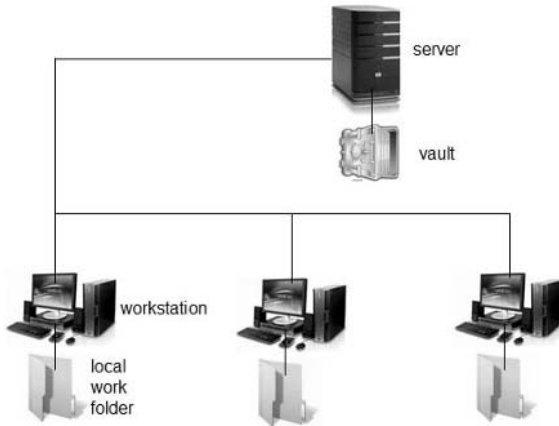
The diagram in Figure 21.3 shows the relationship between the workstations on which users run SolidWorks and the Workgroup PDM client, and the server where the vault resides.

When Workgroup PDM stores the file in the vault, it renames it temporarily. When you ask for the file back, it renames the file back to the name you assigned it and delivers it to your local computer.

The “server” that the vault runs on does not need to be an actual server machine or use a server operating system. It does need to have a Windows operating system, and it needs to have IIS (Internet Information Service) running for some of the more advanced functionality. The machine the Workgroup PDM vault is on should not be functioning as a workstation. The vault needs to be running all the time, or at least when users will access it. If it is rebooted during the working day, critical data may be lost and the vault will be inaccessible while the machine reboots. Once installed, the vault application is designed to start up immediately after the machine reboots, but depending on settings, it may take the vault application some time to do this.

FIGURE 21.3

The Workgroup PDM server and SolidWorks workstations



Note

One of the huge advantages of using a Workgroup PDM system is that it stores the files on the network for everyone to access, but when you work on the file, it gives you a local copy. While you are working on the file, it remembers that you are working on it and does not allow other people to make changes to that file. This means that you can share data with other users but still work with the speed and security of using your files locally. ■

The vault keeps track of which revision is which. Further, it remembers that revision D of the drawing uses revision H of the assembly, which uses revision B of this part and revision D of that part. If you go back and get a previous revision of the drawing, for example, revision B, the vault gets the appropriate assembly and part revisions associated with that drawing revision. If you were to manage all this information manually, it is very unlikely you could keep it all straight. There is simply no way for you or your manual system to keep track of that level of data reliably and as quickly as a PDM product does it.

Workgroup PDM stores *metadata*, or text-based, non-geometrical data associated with each SolidWorks document. This information helps Workgroup PDM keep track of the relationships between SolidWorks documents. It also makes use of the custom properties associated with SolidWorks documents. For this reason, it is extremely important to have identified and input all required file properties for all files. Once users have seen the importance of metadata input, they'll understand and even embrace the entire concept.

Organizing data into folders with PDM

Workgroup PDM organizes data into folders it calls projects. You can organize these project folders any way you want. Some companies organize their data by type of data, the department that

Part IV: SolidWorks Workgroup PDM

created it, the project, or any of a number of ways. This is flexible, and I discuss it further later in this chapter.

Note

The name “project” was chosen as an unfortunate naming coincidence for the folders. This make some users think they have to organize their data by work project. For this reason, in this book, I will use the word “folder” rather than “project” to try to keep the organization discussion as generic as possible. ■

Workgroup PDM can also use a lifecycle status, so that as a document moves through the product development process, its status can change. For example, a document may start out at the lifecycle status of Concept, move on to Development, then to Engineering, to Manufacturing, to Production, and finally end at Obsolete. You might also choose to insert a Pending lifecycle status in between the phases when it is being approved or reviewed.

PDM uses a concept of “ownership.” If you take ownership of a document, then only you can make changes to the file and put it back into the vault. Other users can look at the file and get copies of it, but cannot uprev the file in the vault. This is how PDM handles the concept of write and read-only access. Each time you put a new version of the document into the vault, the revision level changes. The administrator can control the revision scheme, using letters, numbers, or a prefix or suffix.

Another useful piece of information about each document placed in the vault is the Where Used list. If you open a random part in SolidWorks, the software cannot tell what assemblies or drawings that part may be used in. SolidWorks Explorer can search the files in a directory and report if an assembly or drawing uses a file with the same name as the one you are asking about. Workgroup PDM keeps track of which part goes where through its data file for that particular part. If the part is used in multiple assemblies or if the part has a drawing, and those documents have all been checked into the vault, the Workgroup PDM knows about it. In addition, by reading the references inside the file, Workgroup PDM knows which files reference the document you are looking at. This way, you can traverse up and down the Bill of Materials (BOM) to both parent and child documents.

In order for the whole arrangement to work, you must have three software components installed:

- **Vault server application (the program that makes the vault work).** Goes on server computer
- **Vault admin client (to set vault settings).** Can go on server or client computer
- **Client application (to access the vault data).** Goes on client computer

Implementing Workgroup PDM

Electronic documentation standards and practices vary widely from company to company and depend on many factors, such as the size of company, self-discipline of users, sophistication of documentation procedures, presence of dedicated IT or documentation personnel, and demands of internal documentation “customers,” among others.

The factors determining the success or failure of a Workgroup PDM implementation include a well-defined process and users who understand (and follow) the process. The process should be defined *before* implementing the Workgroup PDM tool, because a Workgroup PDM tool on top of a bad process simply automates problems.

While I cannot make decisions for you on how to implement a process, I do present you with the information you need to make the decisions yourself. My recommendations lean toward the conservative side. The more you, as SolidWorks Administrator, and your users know about the product and process, the more you can relax the rules. How much to relax the rules is the difference between conservative best practice and a possibly riskier but more efficient or flexible system.

Implementing general PDM

Before implementing a PDM system, the first step toward making improvements is understanding what you are doing now. You may be surprised to see the wide range of answers you get when asking people directly involved with the process to define it. PDM processes are often best represented in a flowchart form that shows the different paths a given document takes as it is created, reviewed, approved, released, modified, and obsoleted.

Once you understand your existing process and have documented it, you can make conceptual corrections, adjustments, or changes. The next step is to understand the strengths and weaknesses of the PDM tool, and what is required to fit the tool into your process, or how your process needs to be tweaked to accommodate the tool, if necessary.

You may be dreaming of an ideal situation in which you never have to compromise your goals to fit the tools that are available in real life, but this is not likely to happen. There is no such thing as perfect software or software that does exactly what your company's special process requires. Insisting on a one-to-one equivalence between your old system and a new implementation of any kind of PDM is impractical and restricting. Be prepared to accept at least some compromise in order to come up with an automated document management process that really works well. It is not ideal to allow your tools to dictate your workflow, but on the other hand, fighting against something that will not change is a losing battle, and you need to recognize it early on.

Mapping the road ahead

One of the most important aspects of process implementation is to plan the road ahead. You need to understand where possible problems might be encountered before they arrive. This is the biggest advantage of going forward with the help of a professional implementer. This person knows the territory completely, and having been through similar situations with other companies, should be well acquainted with solutions.

File naming non-negotiable

In Workgroup PDM, there are a couple rules that are non-negotiable:

- You must not put the revision level in the filename.
- You must not create duplicate filenames.

Part IV: SolidWorks Workgroup PDM

Breaking these rules systematically always causes failure with a Workgroup PDM implementation. You may put the revisions levels in the filenames with your AutoCAD files or even SolidWorks files before using Workgroup PDM, but you cannot do this with Workgroup PDM. To put it another way, while these techniques are considered merely “bad practice” with manual file management, with Workgroup PDM, they are showstoppers.

The recommended best practice is to have a sequential part number for the filename. Descriptive names cause problems because they can easily become duplicated. Descriptions are best used as a file property. Intelligent or semi-intelligent part numbers are often attractive in various ways, but there are almost always problems with these as well due to the possibility of duplication or the system not accounting for all possible parameters in the intelligent naming system.

Cross-Ref

Naming conventions are covered in detail in Chapter 3. ■

Using installation best practice

I provide detailed installation instructions for Workgroup PDM later in this book, but in this chapter, I talk about the installation conceptually, in a high-level overview to help you understand the scope of an implementation project.

When setting up Workgroup PDM, you need to install the server portion first. It should go on a Windows server operating system, XP, Vista, or Windows 7 machine, as appropriate for the version you are using. SolidWorks Administrators installing Workgroup PDM typically ask how much hard drive space to allocate. A good minimum starting point is 50GB for a low traffic vault. It is easy enough to move the vault to a new drive or to move it to a new server altogether. The available space needs to be *contiguous* space, meaning it is all on a single partition; it can't be distributed across multiple drives. With the low cost of immense hard drives, this shouldn't be an issue. How quickly you fill it up depends mainly on the types of files your company makes, the frequency with which they are revised, and your revisioning policies.

The installation will ask where to put files for the program executables and for the vault data. They can be in different locations, even on different computers, but the executables should be on a machine with a fast processor and an amount of RAM commensurate with the size of the vault and other applications it is running. The vault data should be somewhere with a fast network connection and, optimally, a fast hard drive. Ideally, they should be on the same computer. Setting up on multiple computers means that the vault is twice as likely to go down, because both machines must be running for the vault to function.

I recommend installing the Vault Admin tool on the server, if only for convenience. Vault Admin is part of the Client install, so after the server installs, it asks you if you want to go ahead with a client installation. You can install the vault admin wherever you like, or in multiple locations to ensure you or other administrators can access vault settings wherever you are. Vault admin does not install on the server by default, but not having it there is a real inconvenience. Certainly, you will want it on your own computer, and possibly on other user computers as well.

Other recommendations for server setup would be to use Windows permissions to make the vault data folder inaccessible to users. No one needs to be browsing through the vault unless he is doing advanced troubleshooting. This is something some users who are used to wielding control may have some difficulty with, but the only thing that can come from general users poking around inside the vault is trouble. These files are not meant to be user-readable, although you can certainly reverse-engineer them to figure out how the files are structured.

The Workgroup PDM vault should never be installed on an end user's workstation unless there are very few other end users, the end user is also the Administrator and the primary user, and the machine isn't rebooted when other people need access to the vault. This is not recommended at all for more than two to three users. It would be better to install the vault on a retired or inexpensive computer that meets the other criteria for the operating system, hard drive space, processor, connectivity speed, and RAM.

Attempting unsupported cross-platform installations

Workgroup PDM vault will not run on Mac, Linux, or Novell operating systems. However, it is possible to have the executables installed on a Windows machine, and the vault storage area to be set up on a Linux or Novell server. Please note "possible" is not the same as "recommended." SolidWorks does not test or support the software on these cross-platform installations. It is likely that neither SolidWorks Corp. nor your reseller will be able to assist you if an installation of this sort runs into trouble. I have done a few of these installations on a Novell server, and they are most prone to problems with user access to the vault data, in most part due to assigning permissions to the account running the service that accesses the files on a Novell server. I have seen IT people insist on this type of installation for one reason or another, but you need to know that while it does work, it is not highly reliable.

I show you how to redirect the server to a different location for the vault files later in this chapter, in the section on moving the vault.

Planning for the Workgroup PDM Viewer

The Workgroup PDM Viewer is a browser-based view-only client interface used to access data inside the Workgroup PDM vault. It can be used by people inside or outside your company, and requires some IT infrastructure setup. Fortunately, all you really need is to have IIS (Internet Information Service) installed and running on the vault computer, and the installation takes care of the setup.

The Workgroup PDM Viewer was formerly called the Web Portal, and was a part of a larger suite of software called the Advanced Server, which also included Triggers and API for custom programming. The triggers and API have been added to the basic Workgroup PDM package, and now the only additional cost component for Workgroup PDM is this Workgroup PDM Viewer.

To activate the Viewer, use the Change License button on the Vault Settings page of the Vault Admin tool to enter your license string.

Preparing for Vault and Client Maintenance

Maintaining the vault and client software is not difficult, but it does require some preparation, some follow up, as well as some process following. Usually this involves starting or stopping services, rebooting computers, warning users that the vault is about to go down, and other simple housekeeping of this sort.

Workgroup PDM maintenance is not difficult, but it is a routine to which you need to adjust. Because of the relationship between pieces of software on different computers, it matters if one is upgraded and the other is not. It also matters in which order they are upgraded. This section of this chapter prepares you for your simple maintenance routine.

Utilizing your backup plan

Obviously, you should always back up important data like your company design data, especially before running maintenance procedures where in the worst possible scenarios, something could go wrong. The Workgroup PDM vault can be backed up, but it should be done when changes are not being made to the vault, and the backup should be done in the vault's entirety, not in parts. For restoring, the only recommended restore technique is to restore the entire vault. Incremental restores may be possible, but will not work unless you have a thorough understanding of the vault structure, which I don't cover here. Save yourself time and just do a complete restore. I don't advocate hacking the vault. Correct methods are provided, and there is no need to take the risk involved in relying on unproven hacks.

To do a backup or restore, you will want to shut down the Workgroup PDM service on the vault server through the Computer Management or Services window. Then backup or restore the vault and restart the service.

Performing vault maintenance

The vault and the clients should always be at the same service pack level. If the client is updated to a later service pack or version than the vault, the client will not be able to access the vault. If the vault is updated and the older client tries to access it, the vault pushes the update down to the client, updates it, and then allows it to access the vault.

The vault pushing the update down to the client only works for service pack level updates; it does not work across versions.

Service pack updates

To update the vault server with a service pack, download the appropriate file from the subscription service area of the SolidWorks Web site. Workgroup PDM downloads are broken up into Server and Client files.

The Workgroup PDM service pack files are not incremental like SolidWorks service packs. Each download is the entire program, so it doesn't matter what service pack you upgrade from, as long as the "upgrade to" is correct. Additionally, SolidWorks does not label the download files according to which service pack they belong. You should remember to rename the files when downloading. The file will be called `pdmwserver.exe`. I would suggest you rename it `pdmwserver_2010_sp1.exe` to keep in your archive of downloads.

When you run the update file on the server, the vault service temporarily shuts down, so users should be warned that the vault is going down for a few minutes. Once the service pack is applied, the vault must start again. Depending on your hardware, the vault takes approximately one minute per gigabyte of data to restart. If you use the Validate Vault setting from the vault admin tool, starting up the vault can take even longer.

Major version upgrades

To upgrade the vault server from one version to the next, make sure to back up the data first. Next, run either the downloaded file or the file from the installation disks just as if you were installing a service pack.

When you install a new version of the vault, it is not required to convert all of your SolidWorks data to the new version. In fact, I recommend against using the automated tools that reach inside the vault and convert your models and drawings. The reason for this recommendation is that often, the conversion of a file from one version to the next causes obvious changes in the file. In parts, sketches relations can fail, splines can change shape, and other unexpected errors can show up spontaneously. In assemblies, mates can fail, and parts may move in unexpected ways. In drawings, you may see annotations lose attachment points or move to a new location. If this kind of thing happens to your file without changing the revision, and in fact by overwriting the revision, that is sloppy file management.

My recommendation for converting vaulted SolidWorks documents to a new version is to do it as needed, and whenever it is done, it should always involve a revision change. If you have a very large drawing or assembly and you are concerned about speed while making conversions to a large number of files, check out all of the files you want to convert, pull the files into a local workspace, do the conversion there, and then continue working with the files, and check them in normally when done. There is no need whatsoever to convert files inside the vault. This is also true for files that are managed manually. There is no need to convert files that you are not working on.

Another reason for the recommendation is the version non-compatibility of SolidWorks. Once a file is converted going forward, you cannot open it in the older version or return it to its previous state. This is a ratchet that only drives you to later versions, and serves mainly the CAD vendor, not the customer.

Moving the vault

Sometimes companies outgrow their original vault space. It is possible (and easy) to move the vault to a location with more storage space. To do this, follow these steps:

Part IV: SolidWorks Workgroup PDM

1. Stop the service of the original vault. (The service is called PDMWorks Workgroup Server. Click the Services tab of the Windows Task Manager, press Ctrl+Alt+Delete to access.)
2. Install a fresh vault in the new location.
3. Stop the service of the new vault.
4. Move the vault data folder from the old to the new location.
5. Restart the vault service.

If the move involves moving the vault to a new computer, when you log in from any client, you need to redirect the client to look at the new server name.

It is also possible to make a copy of a vault data folder and then point the existing installation to the new vault data, in essence running it from a backup vault:

1. Stop the vault service.
2. Choose Start → Run and then type in regedit to Registry edit HKEY_LOCAL_MACHINE\SOFTWARE\Wow6432Node\SolidWorks\SolidWorks 2010\ PDMWorks Workgroup\Server to reflect the new location.
3. If this key does not exist, do a search within the registry for “rootPath.” (This path is for a 64-bit operating system.)
4. Restart the vault service.

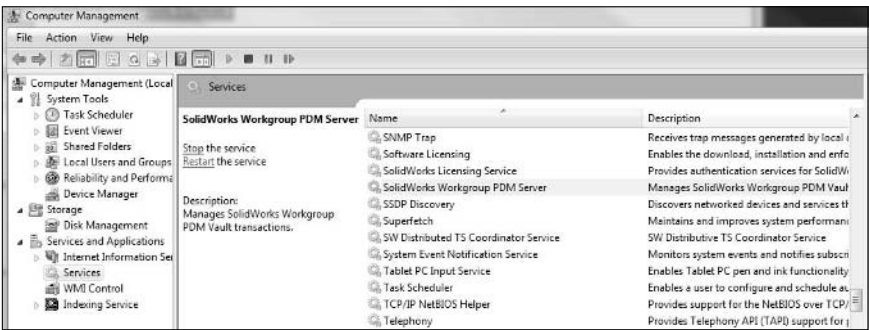
Note

You can find the vault service by right-clicking Computer in Windows Explorer, selecting Manage, and then selecting the Services item in the list. The SolidWorks Workgroup PDM Server is listed alphabetically in the list, as shown in Figure 21.4. ■

Only one set of vault executables can run on a single machine at once; if you want to run multiple vaults simultaneously, you have to have them running on separate machines.

FIGURE 21.4

Turning on and off the SolidWorks Workgroup PDM server service



Summary

SolidWorks Workgroup PDM is low-cost, easy-to-use, automated file management software included with SolidWorks Professional and Premium that solves many problems manual file management and multi-user environments can create.

Installing the software is not difficult, but getting it right the first time requires some planning and understanding of your current documentation processes. Doing some homework on your current process can help you replicate it to some extent in Workgroup PDM. However, you shouldn't plan on a perfect match between your manual and automated documentation processes; the software is flexible, but not infinitely tweakable.

Installing and Configuring the Vault

Installing the vault is straightforward. You have to make a few decisions, but you can change anything you do later if you need to.

Vault configuration on the other hand is a bit more complex and requires more planning. Some of the options are difficult to change once they have been set, especially if your vault has any real production data in it, so you need to get the settings set the way you want them from the outset.

One thing you might consider is doing a test installation before you finalize things with your production vault. If you create a vault and it turns out not to be what you want, it is easy enough to wipe it clean and start fresh. Again, none of this is very difficult, but much of it requires decision making that affects the way you use the software into the future. How you will use the software later can be difficult to project when you haven't started using it yet. This is where a test installation comes into play. In this chapter, I walk you through an installation and configuration of a test vault.

Cross-Ref

Chapter 24 covers the Workgroup Product Data Management (PDM) user interface along with client best practice usage, which is where you will see the final effect of many of the options you set in the admin configuration stage. ■

IN THIS CHAPTER

Installing SolidWorks
Workgroup PDM tools

Setting up the vault

Installing the Workgroup PDM Vault and VaultAdmin Tool

The SolidWorks Workgroup PDM tool consists of a vault (big storage location), vault executables (program files) that both go on the vault server, and a set of client executables that go on the end user's computer. In some rare situations, the user and vault server computers can be the same. A test installation of the vault is one situation where they can be the same. You would not want to use this scenario in a real production installation unless you were installing Workgroup PDM and only one or two people were using it.

I am using the SolidWorks 2010 beta 1 64-bit Vista version for this test installation. Installing onto Windows 7 should be somewhat similar, as should be Windows XP, and the 32-bit and fully released versions after 2010 Service Pack (SP) 0.0.

This installation starts from downloaded data rather than a DVD. In the Customer Portal area of the SolidWorks Web site, which you have visited several times in the course of this book, go to the Downloads area. In the Select Version drop-down list, select 2010. Select the SolidWorks tab, and SolidWorks 2010 XP and Vista – 64 bit, as shown in Figure 22.1.

FIGURE 22.1

The Downloads page on the SolidWorks Customer Portal

Home > Training & Support > Technical Support > Downloads

Downloads

Download new versions** and Service Packs** for SolidWorks, Simulation, PDM, 3D VIA products, and Free CAD Tools (eDrawings, DWG Navigator, DWG Gateway, Drawings Now, SolidWorks Explorer).

Select Version: **2010**

SolidWorks | 3D VIA Composer | FREE CAD TOOLS

References

- Fixed SPRs*
- Install FAQs
- Product Comparison
- SolidWorks Labs

List of SPRs fixed within a Service Pack
Installation frequently asked questions
SolidWorks, SolidWorks Office Professional and Premium
Preview not-yet-released technologies and applications

Note: Licensing and activation required for all products.
** Education Subscription Service customers should access Service Pack updates only.

| Product | Version | Service Pack | Edition | Posted |
|------------|---------|--------------|-----------------------|------------|
| SolidWorks | 2010 | Beta 1 | XP and Vista - 32 bit | 06/22/2009 |
| SolidWorks | 2010 | Beta 1 | XP and Vista - 64 bit | 06/22/2009 |

| Product | Version | Service Pack | Edition | Posted |
|---------------------------|---------|--------------|-------------------------|------------|
| SolidWorks Enterprise PDM | 2010 | Beta 1 | XP/Vista/2003 - x32/x64 | 07/02/2009 |

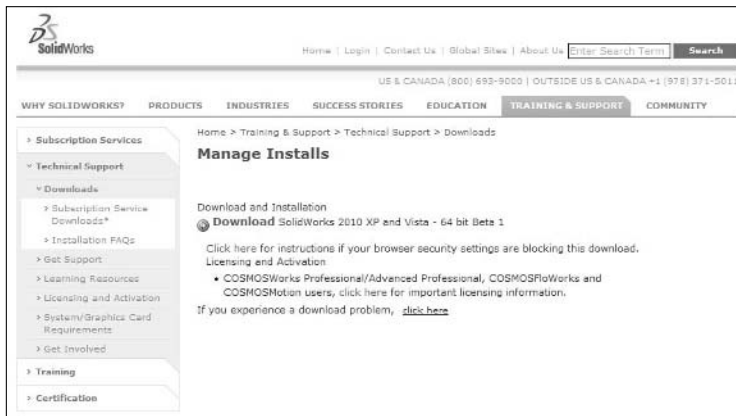
* Full access requires an active Subscription Service contract.

Chapter 22: Installing and Configuring the Vault

Click the Accept button on the SolidWorks End User License Agreement (EULA) after reading it thoroughly. The Manage Installs page opens. If you click the Download SolidWorks 2010 XP and Vista – 64 bit Beta 1 link, you download the Installation Manager, and then have the option to download other SolidWorks software. This is the front door. For what you are doing here, you do not need the formal introductions that come with being greeted at the front door. For your purpose, you want the back door. Click the link on the last line, where it says, “If you experience a download problem, click here,” as shown in Figure 22.2.

FIGURE 22.2

Finding the back door on the Manage Installs page



The back door brings you to the Manual Installs page, where you should select your language and the level of the SolidWorks product, which should be either Professional or Premium.

Clicking Continue takes you to the Download Packages page, which has a large list of download options. Find the box called Step 5, which is the SolidWorks Workgroup PDM Server with a download of just over 500MB. Download this selection.

Once the file is downloaded, extract and run it. Figure 22.3 shows the starting point. Make sure that you are running the extracted installation files on the server machine where you want to install the vault server.

When you click through the EULA again, you get the Setup Type screen, shown in Figure 22.4.

Typically, you do not want the vault data to go onto the default drive, which is usually the C: drive. You need to select a drive with a good amount of space. A good starting size for the vault is 50GB, and it should last you for a while, depending on other folders that are also on that drive and may grow. You can change the location of the vault data later if need be.

Part IV: SolidWorks Workgroup PDM

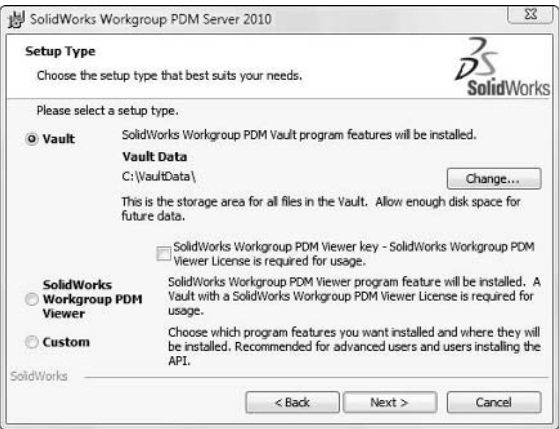
FIGURE 22.3

Installing the Workgroup PDM Server



FIGURE 22.4

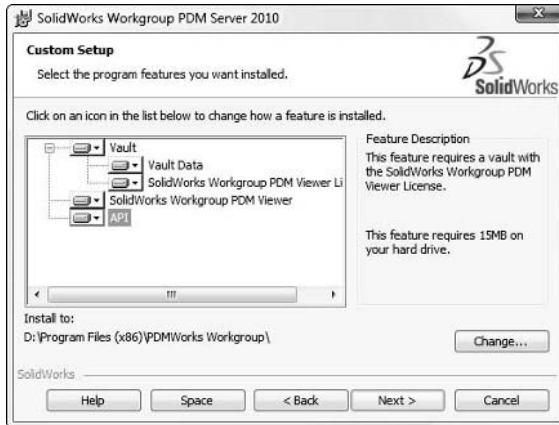
Select the Setup Type



Just for thoroughness, select the Custom option and click Next. You see the Custom Setup screen shown in Figure 22.5.

FIGURE 22.5

Customize the Setup with a location with a lot of open disk space for the Vault Data.



Notice here that there is a difference between the vault and the vault data. The vault is being installed to a Program Files folder, and the vault data is going somewhere else. This is an important distinction that you need to understand. The executable files that make the vault work as a software program are not located in the same place as all the data that you are going to store in the vault. When the vault server service runs, it is running the files located in the path indicated by the vault path shown in Figure 22.5.

Note

Notice that the names for the program files paths have not been changed since the name of the software was officially changed from PDMWorks to SolidWorks Workgroup PDM. Workgroup PDM has not received much development attention since the addition of the product formerly known as Conisio, now known as SolidWorks Enterprise PDM. ■

In this example, I did not have Internet Information Services (IIS) turned on, as you can see in Figure 22.6. IIS enables the SolidWorks Workgroup PDM Viewer (formerly known as the PDMWorks Advanced Server Web Portal). The Workgroup PDM Viewer is Web-based at an additional cost. If you do not have a license for Workgroup PDM Viewer, you can skip to Figure 22.12.

To install IIS, choose Start ➤ Control Panel and then click the Programs link. Under Programs and Features heading, click the Turn Windows features on or off link. Next, select the Internet Information Services option in the Windows Features dialog box, shown in Figure 22.7.

Part IV: SolidWorks Workgroup PDM

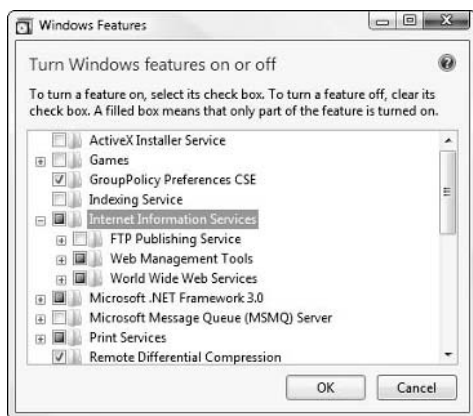
FIGURE 22.6

IIS not detected so a warning is issued



FIGURE 22.7

Turning on IIS in the Windows Features dialog



If you have a license for the Workgroup PDM Viewer, you can enter it when you get to the screen shown in Figure 22.8; if not, you can just leave the boxes blank.

The Workgroup PDM Viewer was formerly called the PDMWorks Web Portal. A Web-based interface requires Microsoft Internet Explorer 5.5 or later to run. If you have it installed on your local computer, you can access it through Internet Explorer (IE) using the address `localhost/pdmweb`. If you want to access it from another computer on your network, enter the computer name in place of `localhost` in the IE address bar.

SolidWorks Corp. has a sample setup available to all users on the Internet. This sample can be useful in evaluating if the product is worth the additional expense. Figure 22.9 shows this interface. Three public logins are available, where the user name and the password are the same. They are `guesta`, `guestb`, and `guestc`. To access this sample Workgroup PDM Viewer, go to `http://pdmdemo.solidworks.com/pdmweb/frame.html`.

Chapter 22: Installing and Configuring the Vault

FIGURE 22.8

Enter your serial number and computer info for the Workgroup PDM Viewer.

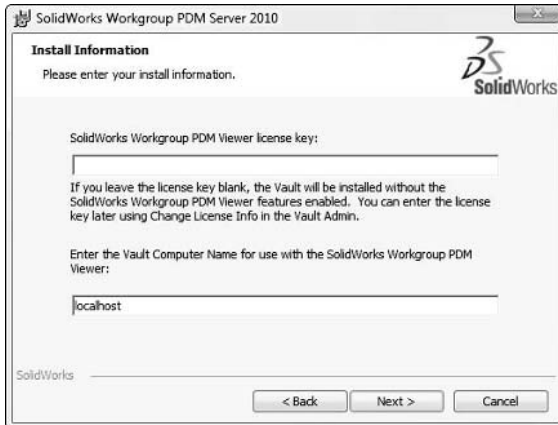


FIGURE 22.9

SolidWorks Corp. sample Workgroup PDM Viewer



At this point, you are ready to install the vault software, so initiate the install by clicking the Install button in the dialog box shown in Figure 22.10.

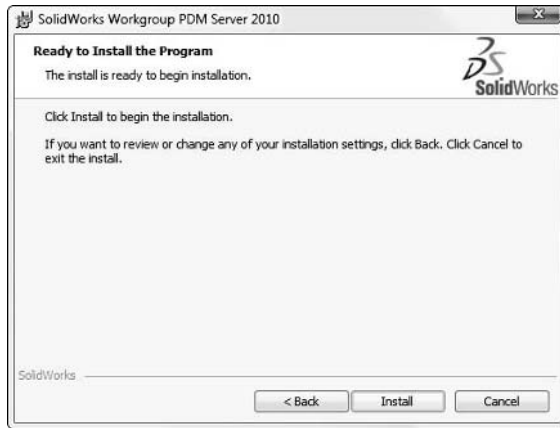
After the vault is installed, you need to verify that it is there. Right-click My Computer, select Manage ➤ Services; then verify the SolidWorks Workgroup PDM Server service is running.

Cross-Ref

To view this service running in the Services interface, see Chapter 21 (Figure 21.4). ■

FIGURE 22.10

Installing the vault



The next step is to install the VaultAdmin in order to set up the vault with the appropriate settings. Interestingly, the VaultAdmin is part of the client install, not the vault install. Because the Workgroup PDM client is actually the SolidWorks Explorer interface, the client is already installed with SolidWorks. To get the VaultAdmin tool, you have to modify the SolidWorks installation.

To initiate the installation modification, choose the Control Panel ➤ Add/Remove Programs (for XP) or Features and Programs (for Vista), and click the SolidWorks entry you want to add the VaultAdmin to, and select Change from the options.

This initiates the SolidWorks Installation Manager (SWIM). In the SWIM, indicate that you want to modify the installation, as shown in Figure 22.11, then click Next.

The next screen should show your serial number. Just click the Next button on this screen. At this point, the SWIM contacts the SolidWorks database across the Web to check your serial number's eligibility for the software, after which you should see the screen shown in Figure 22.12. This Product Selection window enables you to select what software you want to install. During the default installation, the VaultAdmin tool is not installed.

FIGURE 22.11

Modify the SolidWorks installation

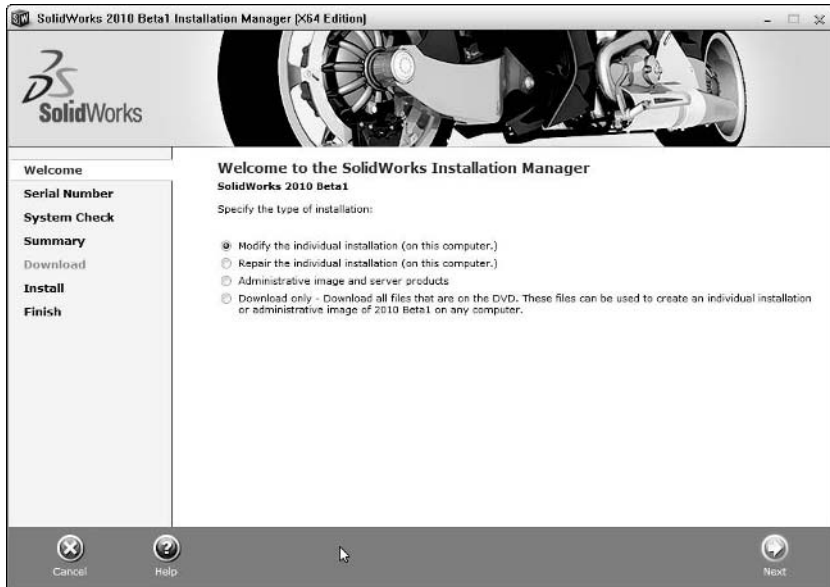
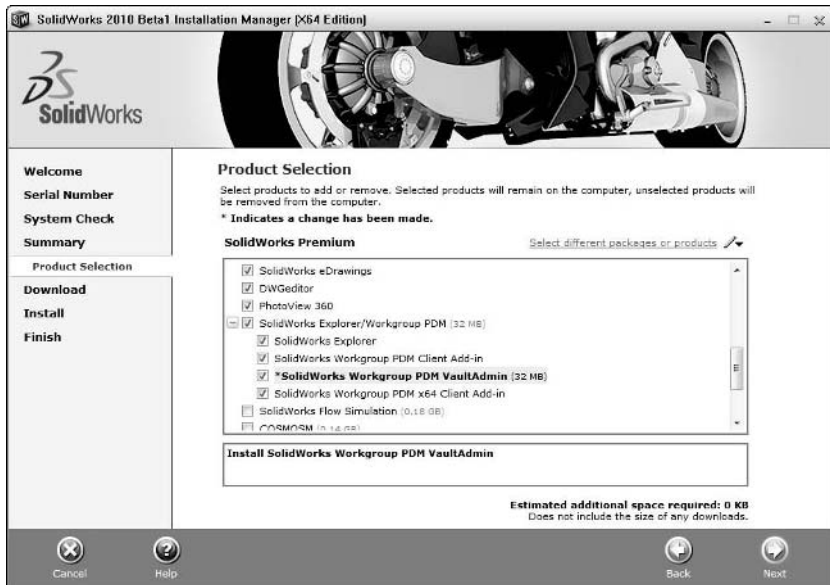


FIGURE 22.12

Adding in the VaultAdmin to your SolidWorks installation



Part IV: SolidWorks Workgroup PDM

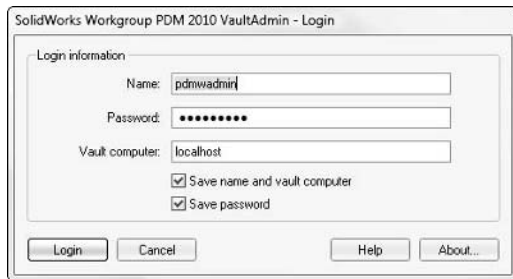
You probably do not want the VaultAdmin tool installed on every end user machine. However, I recommend you install it on the server as a convenience; on the Administrator's computer; and install the VaultAdmin tool on a remote user's computer as well, not for this user to use, but for you to use when you are away from your desk and need to make a change. If your company has SolidWorks users scattered throughout your facility, and it can be a bit of a walk back to your Administrator's machine, you might also want to install the latter.

At this point, using the VaultAdmin, you can verify that the vault is installed correctly and that the admin tool is installed correctly. Access the shortcut to start the VaultAdmin through the Start Menu by choosing Start → All Programs → SolidWorks → SolidWorks Workgroup PDM VaultAdmin.

The default admin user name is pdmwadmin, and the default password is the same as the user name. The first time you log in, you will also need to enter the computer name for the location of the vault. If you are logging in from the same computer that the vault is located on, just use the vault computer name localhost. If you are logging in from a different computer than the one the vault is installed on, make sure that both computers are available on the network, and enter the name of the vault computer. The login screen is shown in Figure 22.13.

FIGURE 22.13

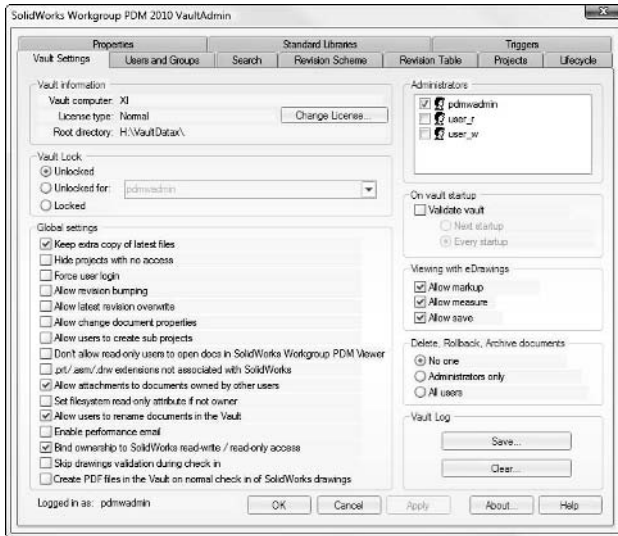
Logging in to the VaultAdmin tool



The default settings for the VaultAdmin are shown in Figure 22.14. This is the starting point for examining all of the settings for the vault. For some, it is obvious what they do from the descriptions, but for others, you will need to do some experimenting before deciding how you plan to use the options.

FIGURE 22.14

Default settings on Vault Settings tab of VaultAdmin



Configuring the Vault

In this section, I go through all the settings in the vault, except for the Users and Groups tab, which I cover in Chapter 23 on users, groups, and permissions. You may want to test out some of these vault settings on your practice install with a couple of SolidWorks users to help you. You may feel the need to experiment with some of the settings to see how they work in practice before setting them up on your production vault for all users to use. Reading through this section will give you a good idea of what the settings do, and include some of my recommendations, but it is also a good idea to try them for yourself to make sure the settings will work correctly in your business.

You can copy a vault configuration from one vault to another. This does not involve copying keys out of the Windows Registry, just text files from the vault folder. In the folder where you installed the vault data, by default called VaultData, there is a folder called `pdmrc`. The settings controlled from the vault admin tool are all stored in these text files. To copy settings from one vault to another, copy the following items:

- `revision_def` folder. Defines the revision scheme
- `globals.pdmw`. Defines the settings on the Vault Settings page
- `projects.pdmw`. Defines the project folder hierarchy
- `ole_mapping.pdmw`. Defines metadata properties in Workgroup PDM

- `sw_mapping.pdmw`. Defines which SolidWorks custom properties map to which Workgroup PDM properties
- `tboptions.pdmw`. Defines how Workgroup PDM deals with Toolbox data
- `cust_prop_defs.pdmw`. Defines the custom properties used in Workgroup PDM
- `status_defs.pdmw`. Defines lifecycle status levels
- `users.pdmw`. Defines the list of users
- `admin.pdmw`. Defines which users are administrators

These files are all readable in Notebook, but the contents may not make much sense to you without some reverse-engineering of the data.

Caution

I do not recommend hacking the Workgroup PDM text files. Copying vault settings is useful and safe if done correctly, but editing the contents of the files should be left alone. ■

Vault Settings

The default page of the Vault Admin tool contains many general settings for the vault. These settings all refer to the settings shown in Figure 22.14. Each of the headings that follow in this section include related settings that are grouped together visually in a box.

Vault information

The information in the vault information section is useful when someone is helping you troubleshoot the vault. Vault computer is simply the name of the computer on which the vault is installed. The license type can be either Normal or Advanced Server. If you have a license for the SolidWorks Workgroup PDM Viewer, you can click the Change License button to activate that part of the license. The Root directory is the top-level folder for the vault.

The Change License button enables you to input a license string for the Workgroup PDM Viewer, which is a separately licensed, additional cost browser-based viewer that enables people inside and outside of your network and company to gain read-only access to data inside the vault.

Vault Lock

When you make changes in the vault to users, groups, projects, or permissions, it is advisable to use LockVault button to keep users from logging in or initiating vault operations that may be interrupted by the change. It also can be convenient to unlock it for the Admin, but keep everyone else locked out if you are running some tests on the changes being implemented. Remember to unlock it when you are done.

Global settings

The settings that appear in this section are general-purpose settings covering behavior of miscellaneous items in Workgroup PDM from user access to vault validation, and many other topics.

- **Keep extra copy of latest files.** This is a good setting if you have plenty of space on the vault server and want a backup copy of the latest versions of the parts. These latest copies are not renamed (parts in the vault are always renamed to “_SLDPRT”) and are rarely used, but can be retrieved when necessary. You should not have users accessing these files to circumvent the PDM system unless you have a temporary emergency.
- **Hide Projects with no access.** This setting toggles projects with no access between grayed out (when off) and invisible (when on) for users.
- **Force user login.** This setting forces users to type their password to get into the vault. When it is deselected, as long as the user has typed his user name and password at least once, it will allow access automatically unless the password has been changed.
- **Allow revision bumping.** This setting enables a user with ownership to uprev a document without making changes to the file. This is recommended only for the Administrator to use as appropriate, and generally to turn it back off when done. You cannot allow access to this functionality by user name; it is either globally on or off.
- **Allow latest revision overwrite.** This setting enables a user with ownership who is checking an existing file back into the vault to put an updated document into the vault without upreving it. This setting is recommended only for the Administrator to use as appropriate, and generally to turn it back off when done. This may be necessary when inputting existing data that has a later revision, for example, Rev E, than the initial default revision.
- **Allow change document properties.** This setting enables users who have ownership of documents to change document properties such as Description without needing to check out and edit the SolidWorks file. This is very handy, and a record is kept of who makes what change when, so it is fully traceable. The changes are visible in Workgroup PDM and in the native SolidWorks files as well.
- **Allow users to create subprojects.** The vault structure is an integral part of the document control process, which needs to be guided by a single clear vision. The Allow users to create subprojects setting can make this vulnerable. Use the setting sparingly, and only for well-disciplined users. It is better practice for the Administrator to determine if the new project is a benefit or not, and to add it via the VaultAdmin tool where permissions can be assigned. User-created projects inherit the permissions of the parent project.
- **Don't allow read-only users to open docs in the SolidWorks Workgroup PDM Viewer.** This setting enables read-only users to view documents with eDrawings and does not allow them to save the files to their local computers. Non-eDrawings viewable documents cannot be viewed.
- **.prt/.asm/.drw extensions not associated with SolidWorks.** This setting is usually used when companies have applications like Pro/ENGINEER, Unigraphics, or CADKEY that also use the .prt, .asm, or .drw file extensions. Selecting this setting means that SolidWorks will not open these extensions.
- **Allow attachments to documents owned by other users.** This setting enables one user to put a part owned by another user into his assembly. I can't think of a reason to turn this setting off.

Part IV: SolidWorks Workgroup PDM

- **Set filesystem read-only if not owner.** This setting actually sets the Windows permissions for a file to read-only if it is opened from the vault without ownership.
- **Allow users to rename documents in the Vault.** This setting allows users with ownership to rename a document in the vault. The name is changed in Workgroup PDM and in the native SolidWorks file when retrieved from the vault. When a dependent document is renamed, Workgroup PDM does a Where Used search and updates all the references, so the nightmare of renaming SolidWorks files, which are used in multiple assemblies and drawings, is solved with this toggle.
- **Enable performance email.** This setting sends an e-mail to SolidWorks to help them determine how the program is being used and if there are problems, which are occurring.
- **Bind ownership to SolidWorks read-write / read-only access.** This setting is similar to the previous operating system read-only setting, but is only for SolidWorks read-only. This works in conjunction with collaboration settings in SolidWorks. This setting is also necessary if you intend to allow users to use the SolidWorks Collaboration options.
- **Skip drawings validation during check in.** This setting speeds up the check in process for large groups of files where drawings and their referenced documents may or may not be available.
- **Create PDF files in the Vault on normal check in of SolidWorks drawings.** This setting automatically generates a PDF file, which is placed in the same project folder as the drawing.

Administrators

Only Administrators have access to the VaultAdmin tool. If you grant another user Administrator access, there is no master Administrator, and this user would then have the same power as you to make changes. It is good practice to both have a main Administrator and a backup Administrator who are well versed in the operation of the software just in case the main Administrator is not available. I normally recommend that one Administrator come from engineering and the other come from IT.

I usually set up a group called Admin, and put Admin users into that group, then just assign that group Admin privileges. Users can go in or out of the group. You will see later that assigning privileges to groups is more efficient than assigning them to individuals.

On vault startup

Validate vault and Every startup should be selected except when it is necessary to restart the Workgroup PDM server service of a large vault quickly. Validation goes through the vault, reads all the text-based metadata into memory, and checks it to make sure it has the correct structure. If it does not have the correct structure, it is corrected automatically. It takes approx one minute per gigabyte of vault size to restart the vault, depending on the vault itself and the vault hardware.

Viewing with eDrawings

Viewing with eDrawings enables the Administrator to enable eDrawings Pro type functionality through the client interfaces even if the users do not have a Pro license. eDrawings is a 2D and 3D viewer for SolidWorks data that enables advanced visualization techniques for parts, assemblies, and drawings without needing to have SolidWorks available. It also enables threaded markup capabilities within the eDrawings document, so comments can be sent back and forth between reviewers with a record of who said what.

Delete, Rollback, Archive documents

Delete, Rollback, and Archive documents should be restricted to Administrators only, or turned off entirely and given to Administrators as needed. A recommended practice is to create a special project called “Delete” or “Trash” where users move documents they want deleted. The Admin should go through the folder periodically to clear it out. Folders are best cleared by using the VaultAdmin tool to delete the folder and re-create it.

Vault Log

The vault log is a chronological record of automatic and user-driven actions that take place in the log. It is text based, and human readable. The log can be very useful when troubleshooting problems with your vault. You will be asked for your vault log if you call for tech support. You can tell which user performed which action at what time. You can also use it to tell how long it takes the vault to start up. You can also find error codes for errors recorded in the log. Your reseller tech support should be able to help you decipher vault error codes. The Save button enables you to save a copy to an external location. The Clear button clears all of the data out of the log.

Search

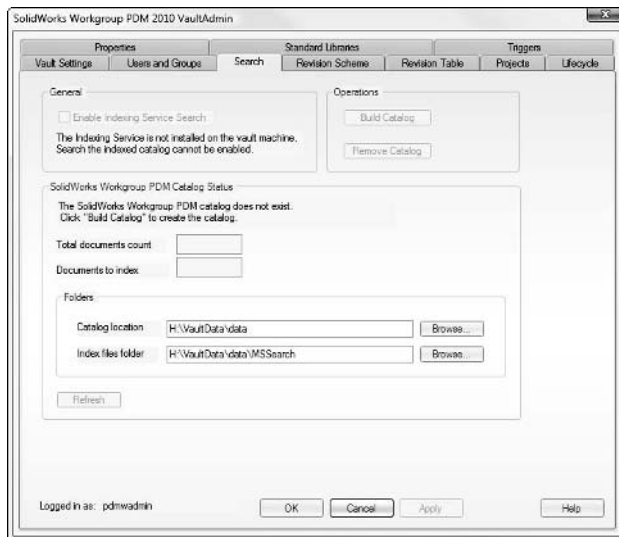
Search in Workgroup PDM is driven by the Windows Indexing Service. The controls on this page enable you build or remove catalogs, and locate where their data is stored. This is not the same as the Windows Desktop Search. Find the Indexing Service in the Services and Applications heading of the Computer Management dialog. You can get to Computer Management by right-clicking My Computer and selecting Manage. Figure 22.15 shows the contents of the Search tab in the VaultAdmin tool.

Once Indexing is enabled, you must create a catalog. You can have different catalogs for different purposes. You must also tell Indexing what locations on your hard drive to index. It is best if you can point to a single top-level folder or drive to accomplish this.

Search in Workgroup PDM can be a very effective replacement for browsing, because search indexes on custom properties as well as filenames. It is a good idea to set up and use metadata inside SolidWorks documents to maximize the search capabilities.

FIGURE 22.15

The Search tab enables you to configure Indexing Service Search.



Revision Scheme

Revision Scheme allows for three revision schemes and a working copy designator. Revision schemes can be concatenated (a revision scheme with two or three levels, such as A.1 or A.1.1), or they can be sequential (after the last rev in the Primary scheme comes the first rev in the Secondary scheme, and so on). If they are concatenated, the last revision could be optional, so you could have one or two schemes, or two or three schemes. You cannot have one, two, or three available at the same time. Figure 22.16 shows the Revision Scheme tab of the VaultAdmin tool.

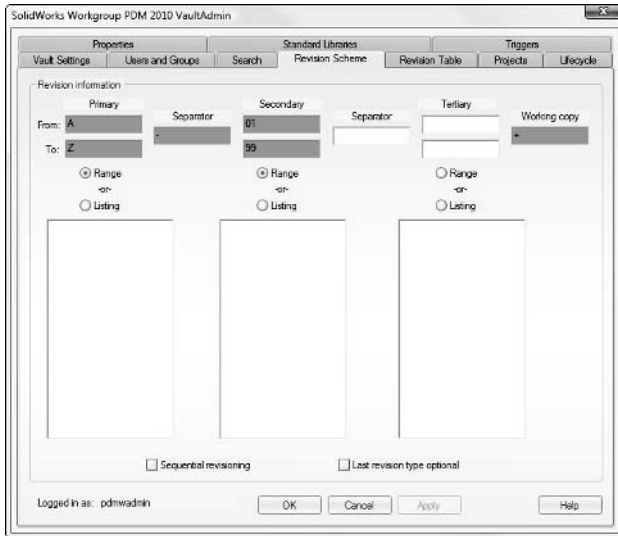
To concatenate schemes, set up the schemes you plan to use, and then make sure Sequential revisioning is unselected. The Last revision type optional check box option only works for concatenated schemes.

The actual revision levels in revision schemes can be spelled out using a list or an automatically generated range. All the revision names for a range must have the same number of characters. If you wanted to go from 1 to 99, you would have to go 01 to 99. If that is not acceptable, the only alternative is to use the listing, where each level would need to be typed in. Fortunately, it is possible to create a list in Excel using the cell paint function, and copy it into the listing box, making long lists easy to create.

If you need to skip letters or numbers, you cannot use the range, and must use the listing. To build a list you can enter data in by hand or use Excel and copy the vertical list into the box. This does not follow standard ASME rules, but it does enable you to establish your own rules, ASME or not.

FIGURE 22.16

The Revision Scheme tab enables you to configure revisions by list or range.



The Working copy designator is used to indicate revisions that are in progress. For example, if you start with rev A, and modify it but you're not ready to release it to B, you could set it to A+ working copy. Each time it is checked in as a working copy, it overwrites the previous A+. Working copy and Latest revision overwrite are the only times this can happen. When the user is ready to check it in at rev B, rev A+ is deleted, and only A and B remain. Working copy is a good way to maintain visibility of incremental changes to documents in progress while still keeping the vault size to a reasonable level. In the end, vault size is the main reason for using Working copy.

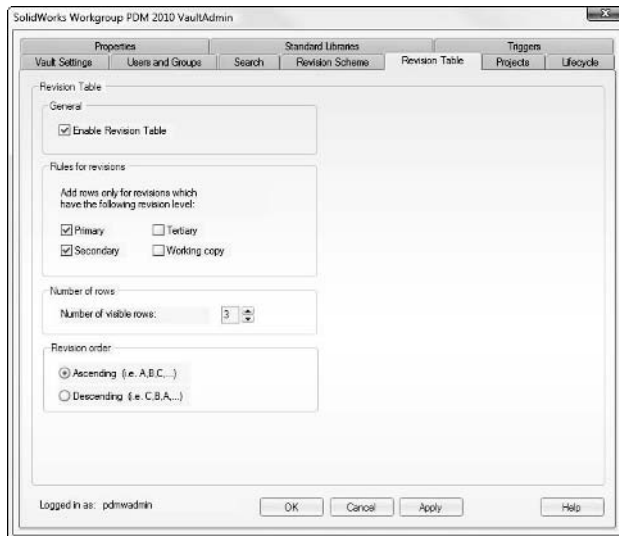
Revision Table

Workgroup PDM can connect with the SolidWorks revision table. Here's how it works: Set up drawing templates with revision tables. Make a new drawing, but don't fill in the table. When you check it in, Workgroup PDM takes the Notes information and puts it in the Description column of the table, and the revision and the date are automatically filled out as well. Figure 22.17 shows the Revision Table tab of the VaultAdmin tool.

The nice thing is that your revision table descriptions become visible inside Workgroup PDM without viewing the actual drawing. Notes can be made to wrap, so there is less likelihood that the description will cause problems during check in. Regardless of what the SolidWorks Tools Options look like, the Revision Table functionality follows your Workgroup PDM revision scheme.

FIGURE 22.17

The Workgroup PDM Revision Table function drives the SolidWorks Revision Table.



Projects

Project structure of the vault should be established with an eye toward browsability and permissions. There is no advantage to having a complex hierarchy. The hierarchy should be no more than three or four levels deep. It is not necessarily a good idea to try to re-create a Windows Explorer folder structure in the vault. In Windows, the primary way of finding a document is to browse for the filename. In Workgroup PDM, you can use a search function to find the filename or one of several properties. Figure 22.18 shows the Projects tab of the VaultAdmin tool.

Try not to duplicate in the project structure data that already exists in the custom properties. For example, creating a project structure that has folders for ranges of part numbers is redundant, and adds no new information. The only exception to this is when the folders are required to set permissions.

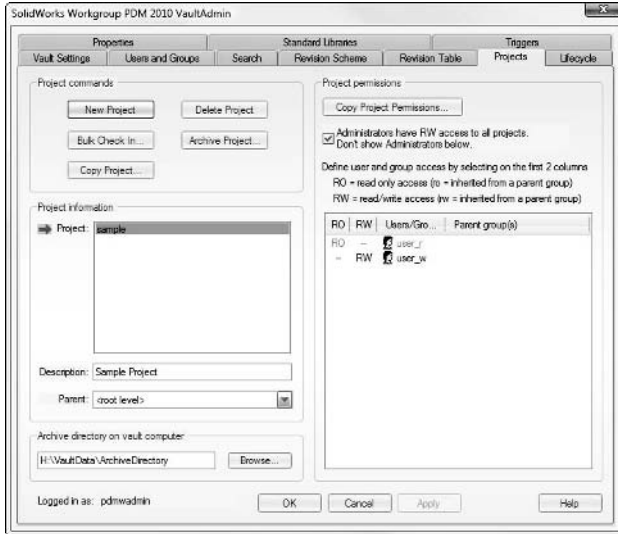
Project names must be unique, but project descriptions can be duplicated. You can work around this by adding the upper-level project name in parenthesis after the branch project name, for example, Drawings (Products). Project names must not use Windows reserved names (such as aux, prn, or com1) or characters illegal for Windows filenames (such as /, ?, *, and so on). The danger of this is that Workgroup PDM can potentially create folders that it (and you) cannot remove. In the vault, Workgroup PDM is creating folders for the projects

Best Practice

It is a recommended best practice to select the option giving Administrators read-write access to all projects. ■

FIGURE 22.18

The Projects tab is where you establish the folder structure of your vault.



Cross-Ref

Users, Groups, and Permissions are covered in detail in Chapter 23, and will not be covered here. ■

Archive

Archiving and recovering old data is usually driven by hard drive space or vault startup time considerations. I recommend removing the archived files to long-term storage media of your choosing. Archives can be driven by the age of files or number of revisions. Restore enables you to bring files back into the vault that have been archived out.

Bulk Check In

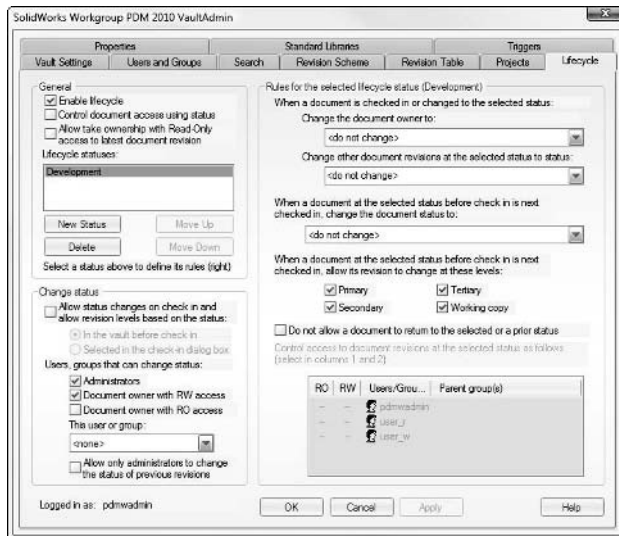
This will probably only be done when a company is starting to populate the vault with legacy data or if you are merging two separate vaults. Bulk Check In can read files from a drive, pull them into Workgroup PDM, create projects according to the Windows folder structure, assign revisions, and several other functions while checking in. This makes pulling in existing folder structures of unmanaged data fast and easy.

Lifecycle

Lifecycle can be complex to understand and set up, but if you take the time to understand its capabilities, it can also be a very useful addition to Workgroup PDM. Figure 22.19 shows the Lifecycle tab of the VaultAdmin tool.

FIGURE 22.19

Lifecycle adds permission and workflow capabilities to Workgroup PDM.



Lifecycle enables you to set up status levels and control access to the files based on the Lifecycle status. You can determine if the status can go back to a prior level or if it can only go on to the next one. You can control who can change the status, for example, the document owner, the Workgroup PDM Admin, or a dedicated documentation person. You can even set it up so if it is a Pending status that is being worked on for an engineering change order (ECO), the next time the user checks it in, it is automatically set to the document manager as the owner. You can do several things here.

The settings on the right-hand side are per status. Therefore, your settings for the first status will be different from your settings for the second status. Each status can be allowed to use certain revision schemes.

Lifecycle works best when there is a central document administrator to manage the ECO process and the release of documents. If there is no document administrator, each user will be given the permission to change the lifecycle status, and if this is done, it is unlikely that user permissions can change with each status, because all users would require write access for every lifecycle status.

It is also possible to set up three groups, such as Users, Checkers, and Approvers, and establish status levels to allow the documents to flow between groups. Lifecycle is not really a workflow direction application, but it can be used to mimic workflow in some ways.

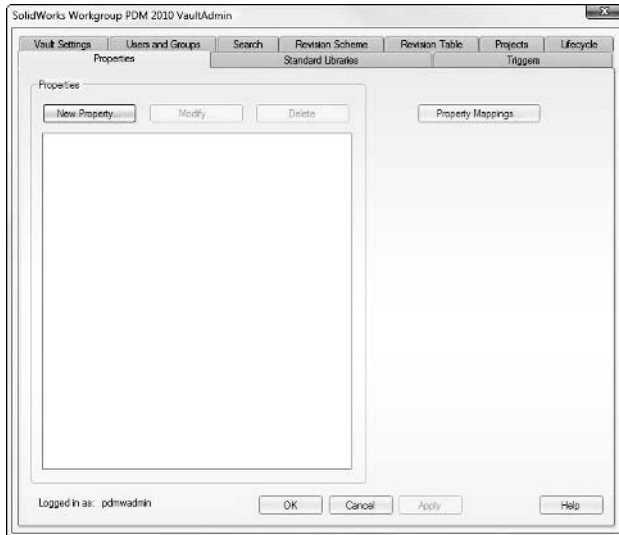
Permissions are still also controlled on a per-project basis. A user must have access to a project or the lifecycle permissions don't mean anything.

Properties

Properties are an underused area of SolidWorks software. If the metadata is entered into the SolidWorks parts, it can be reused in many places and wind up saving you a lot of time. SolidWorks custom properties can be used in PDM display, PDM searches, and PDM reports. Figure 22.20 shows the Properties tab of the VaultAdmin tool.

FIGURE 22.20

The Properties tab enables mapping Workgroup PDM properties to SolidWorks properties.



Workgroup PDM and SolidWorks work very nicely together when it comes to using Custom Properties. If the properties are filled out in SolidWorks, they transfer directly to Workgroup PDM, so that when the document is checked in, the properties are already there. This works if the property names are spelled the same.

Many companies want to limit choices when inputting properties to maintain consistency. A drop-down list would handle this request perfectly. Unfortunately, drop-down lists cannot be created in Workgroup PDM, but they can be created easily in SolidWorks Property Tab builder. These properties are bidirectional, meaning changes in either Workgroup PDM or SolidWorks will show up in the other application as well.

It is also possible to map dissimilar property names, so you can map a property “desc” in SolidWorks to the “Description” field in Workgroup PDM. This might be important if you have a large amount of legacy data that didn’t use the same property naming convention as Workgroup PDM. Mapping also works with AutoCAD files when you use Attributes.

Part IV: SolidWorks Workgroup PDM

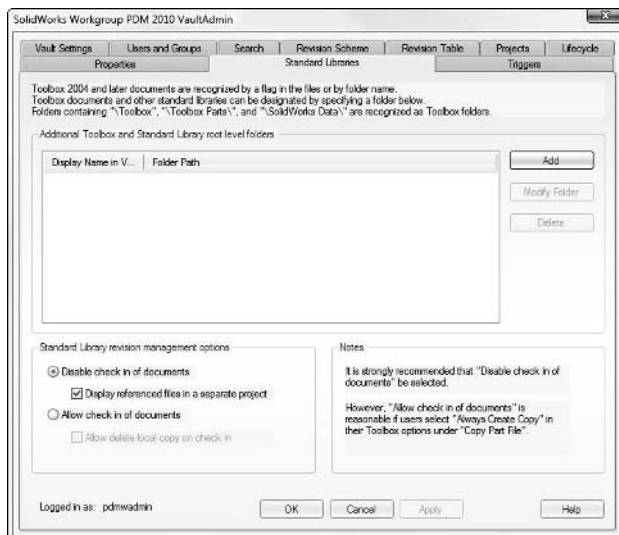
A scenario that you may want to be careful of is when you have SolidWorks custom properties linked to notes or other annotations or tables on a drawing. If the content of the property changes, the next time the drawing is opened, the text from the property changes, which would require a revision. Be careful when updating custom properties that you know your changes will affect.

Standard Libraries

This tab is used when libraries of parts are kept external from the vault. Library Features cannot be used from inside the vault; it is possible to keep them in the vault and managed as non-SolidWorks files, although they will need to be taken out of the vault to be used. Figure 22.21 shows the Standard Libraries tab of the VaultAdmin tool.

FIGURE 22.21

The Standard Libraries tab



If you want the parts in the libraries to remain external to the vault when assemblies they are referenced by are checked in, select the Disable check in of documents option. If you want the library parts to show up into a special folder as shown to the right, then also select the Display referenced files in a separate project check box. Otherwise the library parts will only be shown as links under the assembly. In both cases, the library parts show up on a References list for the assembly, but in neither case will the library parts display a Where Used list. If you need the library parts to show the assemblies in the Where Used list, you can only do it if the library parts are checked into the vault.

Any folder that is added to the list of library folders can be given a special name, and that is how it will be displayed in the vault. "Library Components" will be the top-level folder, but each folder under that takes on the name assigned in the Standard Libraries path box.

Allow check in of documents does what it says. If this option is selected, all library parts will be checked into the vault with assemblies that reference them. You can override the Delete local copies option in Workgroup PDM for library components by selecting the Allow delete local copy on check in check box. Optionally, and this might be a better idea, you can use Bulk Check in to pre-load your libraries into the vault.

Using vaulted libraries

Vaulted libraries work very much like the Design Library in that you can drag and drop them from the vault into the assembly. The vault does not have preview icons, however. The client option Ctrl key to drag requires that Ctrl be used when the user wants a part to be dropped into the assembly. All the standard functionality with configurations and mate references will work when dragging from the vault.

Setting up Toolbox Multi user Network Installation

There are a couple of valid ways of setting up Toolbox so that it works properly with Workgroup PDM. I will spell out a few possible scenarios:

- **Toolbox completely separate.** In this scenario, the Toolbox parts never go inside the Workgroup PDM vault, and Workgroup PDM doesn't know about the parts. To achieve this, select Disable check in and deselect View referenced files. Toolbox will continue to exist as it was set up, either as an individual installation or a shared networked install. A word of caution on individual installs: If you are going to do this for whatever reason (such as network speed issues), make sure to use the Copy Parts setting so that you don't create a situation where configurations for sizes are not available to other users.

The advantage of this method is that you get to do what you want. The disadvantage is that you are likely to run into missing size problems and the "huge screws" problems. In addition, Workgroup PDM will not be able to report on anything about the hardware.

- **Toolbox only for new parts.** In this scenario, Toolbox is used to create only the parts that aren't already in the vault. You can drag and drop parts out of the vault right into your assemblies, which begins to make Toolbox unnecessary. This involves checking the Toolbox parts into the vault. This scenario cannot be used with the configurations setting, so Copy Parts is how it has to go. To use this scenario, put the path to your Toolbox folders in the big window, and select Allow check in. Remember to set the Toolbox setting to Copy Parts. This has to be done on a fresh installation. If your parts already have some configurations created, and you start using Copy Parts, you will wind up with a mess.
- **A hybrid.** Toolbox parts can be created and used from outside of the vault, but still be tracked inside the vault. Use Disable check in, and View referenced files. This has the advantage that you can track your hardware in Workgroups PDM, and use either local, network, config, or copy parts. It has all the standard disadvantages of using Toolbox.

After you have all the parts, bulk check them in to a special hardware directory, and use Workgroup PDM as your library.

IsFastener and IsToolbox properties

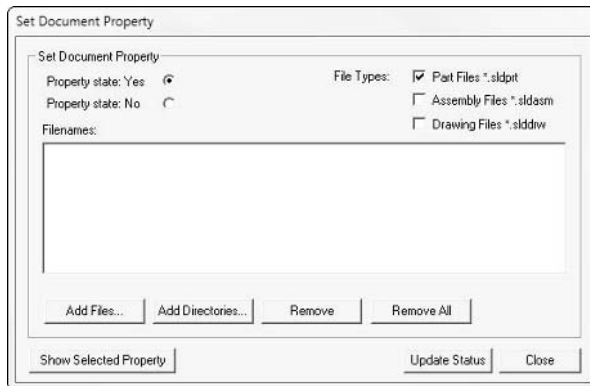
If you have library parts that you want Workgroup PDM to treat like Toolbox parts, you can use the IsToolbox flag. This is similar to the IsFastener custom property that excludes parts from being sectioned on drawings, and from interference detection. There is a fair amount of confusion or misunderstanding about these because they look similar.

IsFastener is a custom property that when it has a value of 1, it excludes the part it is in from being sectioned on a drawing or being included in interference detection in an assembly. To enable, add the custom property IsFastener to a SolidWorks part and then set its value to 1.

IsToolbox is a hidden internal flag that can do what IsFastener does plus tell Workgroup and Enterprise PDM that the part should receive the established treatment for Toolbox parts. You can control this internal setting through a utility that SolidWorks provides found in your SolidWorks install directory, \Toolbox\data utilities\slidsetdocprop.exe. Figure 22.22 shows the utility dialog box.

FIGURE 22.22

Using slidsetdocprop.exe to control the IsToolbox flag



Triggers

Triggers functionality generates an e-mail every time a particular event happens. It requires the MSMQ (Microsoft Message Queuing) service to be set up. The Hostname referred to in Figure 22.23 is the machine where MSMQ is running. Figure 22.23 shows the Triggers tab of the VaultAdmin tool.

To turn on MSMQ, choose Start ⇨ Control Panel. In the Control Panel window, click the link in the left panel to Turn Windows Features On or Off. In the Windows features dialog box, shown in Figure 22.24, expand Microsoft Message Queue Server, then expand Microsoft Message Queue Server Core, and activate MSMQ Triggers.

FIGURE 22.23

The Triggers tab

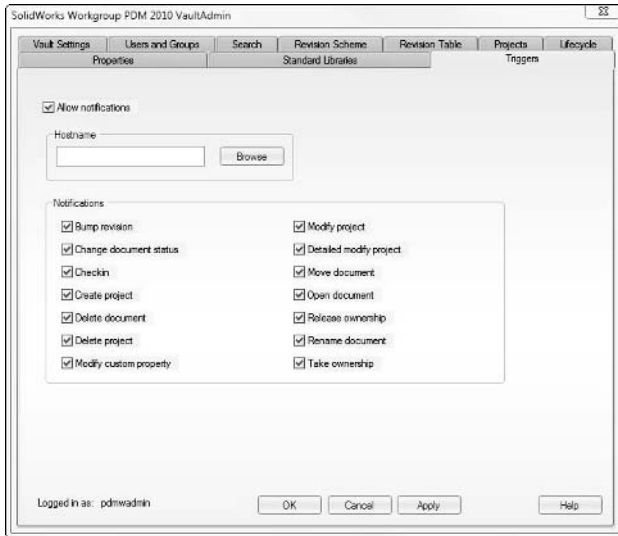
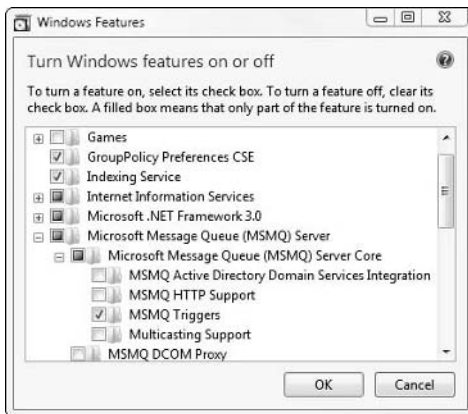


FIGURE 22.24

Activating the MSMQ triggers



In order to work with MSMQ and Workgroup PDM Triggers you must have some programming experience. The programming required to get results from the Workgroup PDM Triggers is beyond the scope of this book. Your best bet for help on this topic is the SolidWorks forum, in the Workgroup PDM discussion group.

Summary

Installing the vault and VaultAdmin tool is relatively easy, but the configuration of the vault requires a lot of decision making. You may not be able to make good decisions about the settings until you have some practice with their functionality in the vault. I encourage you to get together with another user, start using different methods to populate the vault, and start using the data, after which you should come back to this chapter and review the settings to see what makes the most sense.

Vault configuration is not easy to change after the vault has been in use for some time, so it is best to get it right the first time. The only way to ensure that is to install, set up, and practice on a demo vault. Make your experiments and mistakes there, before moving into a permanent production vault.

Establishing Permissions and Managing Toolbox

Along with managing your project folders, establishing how you will handle users, groups, permissions, and lifecycle are very important tasks. These issues are intertwined, and your approach in one area affects the other areas as well.

As Administrator, you assign permissions for users to access data in project folders to either individuals or groups. You can also assign permissions to groups based on lifecycle status in addition to project folders. Lifecycle can be complicated, and changing your mind on how you implement it when the vault is already populated can cause difficulties.

Assigning Users and Groups

When you start putting together your first vault, you need to imagine all the possible roles to which you might need to assign different types of permissions. For example, SolidWorks users are a definite group, along with Administrators, but you might also need to give vault permissions to documentation personnel. Managers might also need a special level of permissions. The permissions you can assign are read only, read /write, and no access. When you get to the Lifecycle tab, you will also be able to assign permission to users or groups to change lifecycle status or allow access to documents based on lifecycle status.

Note

Depending on the size of your SolidWorks installation, it may be advisable to convene a committee to oversee the implementation of Workgroup PDM to make sure the needs of everyone in your organization are being met. ■

IN THIS CHAPTER

Designating users and groups

Creating a project folder scheme

Establishing lifecycle status levels

Allocating permissions

Working within Toolbox and Workgroup PDM

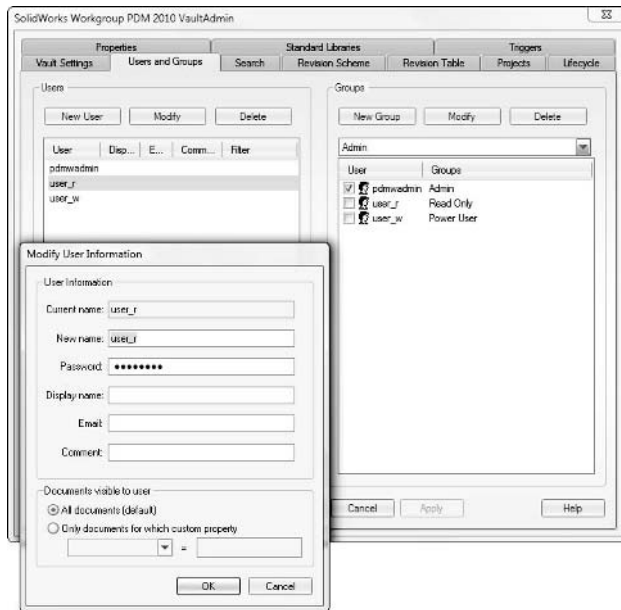
Setting up libraries inside the Workgroup PDM vault

Part IV: SolidWorks Workgroup PDM

Figure 23.1 shows the Users and Groups tab in the VaultAdmin dialog box, along with the Modify User Information dialog box, where you can change user information.

FIGURE 23.1

You control user's membership in groups in the Users and Groups tab.



Once you decide who needs permissions, and what type, you need to establish groups right upfront to make assigning permissions easy. You can make changes to the group structure later if need be, but if you add groups after you establish the project folder list, permission assignment will be more tedious.

Assigning users

Each person who accesses the vault should have his or her own user name and password. Users should not be encouraged to share user names. It doesn't cost anything extra to give everyone their own login, and you get the added benefit of being able to track people's actions in the vault for audit or troubleshooting purposes. The vault logs each action of each user, and in situations where you need to put together a string of events, having one actual person for each login is crucial. For example, if you have several assembly workers and one of them prints out an assembly instruction sheet from the vault, and you have some quality issues with the resulting assembly, someone will want to know which user printed out the document and which document it was. This sort of information can be extremely valuable for understanding how to fix and avoid future problems.

Users can be renamed and their passwords reassigned by the Admin. When a user is renamed, all the documents the original user held ownership of are transferred to the new user. You should change the default admin (pdmwadmin) password as soon as the Vault Admin tool is installed.

Assigning groups

Establish groups with an eye toward functional roles and permissions. By adding a user to a group, the user inherits all the permissions assigned to that group. These permissions are additive, so if the Engineering group has no access to the Purchased Part project folder, and the Purchasing group has read/write access to it, and Joe is a member of both Engineering and Purchasing groups, Joe will have read/write access to the Purchased Part project.

It may be useful to make a group called “Everyone” to assign universal access to a particular project, and a group called “Admin” group if you have users who will administer the vault but will not have the pdmwadmin password. Consider making the default Admin (pdmwadmin) a member of every group. An option exists to give Administrators read/write access to all projects, which accomplishes the same thing more easily, however. This option is in the Project Permissions area of the Projects tab of the VaultAdmin tool.

In the Add New User interface, which is the same as the Modify User Information interface shown in Figure 23.1, you have the ability to make documents visible when a certain custom property has a certain value. For example, if documents have a custom property called “visibility,” the user could be set up so he only sees documents when “visibility” is equal to “manufacturing.” There are several uses for this option, but you should consider its use carefully since document visibility becomes dependent on the use of custom properties.

With all the available options for controlling permissions, it can be tough to come up with a single clear plan. Here is a summary of the permissions possibilities:

- Project folder access assigned directly to users
- Project folder access assigned to groups and transferred to users
- Document access assigned by custom property value
- Document access assigned per lifecycle status

As you can see, you have two types of permissions: folder permissions and document permissions. The folder permissions are applied first, followed by the document permissions. This means if you cannot access a folder, you cannot access whatever document is in it, regardless of the document permissions. If you have access to a folder, but no access to a document that is in it, then you cannot access the document. In order to have access to a document, you must have access to both the folder and the document. Any other combination of permissions does not grant access.

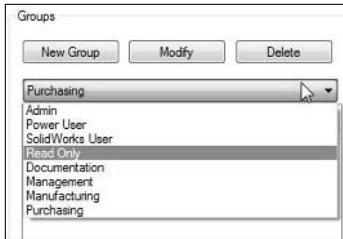
Of these methods, the methods controlling document access directly are less frequently used because direct document permission control is usually used for special cases, such as possibly a lead designer might have access to scheduling documents, but other users would not.

Part IV: SolidWorks Workgroup PDM

Figure 23.2 shows a sample set of groups that I like to use as a starting point for implementations at companies that hire me to help install and set up Workgroup PDM (Product Data Management).

FIGURE 23.2

A sample set of groups establishing users by functions



The set of groups shown in Figure 23.2 establishes users by function. If you organize the folders for the documents by project, you can control permissions to access documents according to a person's function within the company and the particular project to which the document is associated. This makes the most sense. Doubling up on control, such as having groups of users named for projects and folders also named for projects, for example, leaves out a level of control. You could do it the opposite way, however; that is, assign groups of users by project name, and organize documents into folders by document function.

The main point is that by organizing users (in groups) differently from the way you organize documents (in project folders), you get an additional level of control without adding a level of complexity.

Establishing a Project Folder Scheme

Establishing a project folder structure is an important decision. If you make a scheme too detailed, assigning permissions can take a long time. If you don't make the structure detailed enough, you can wind up with too many documents in a single folder. In addition to assigning permissions and limiting the number of documents in any single folder, there are also other considerations involved in this decision.

I have been involved in implementing Workgroup PDM for companies from time to time. I have done several dozen implementations, mainly for smaller organizations, but also some for larger groups. Most of the companies I work with are in the "sweet spot" for Workgroup PDM as far as number of users and size of projects. Sometimes I am asked to help implement the software in a way that is less than ideal. In situations like this, the customer is usually trying to solve a problem with the wrong tool, and it might be making things worse rather than better. The reason for feeling

pressured into selecting the wrong tool is usually simple: cost. If you hire a consultant to do any sort of implementation for you, listen to him or her. You are paying for experience and expertise, and you should be receiving good advice. It could be that there is something about the situation that you don't understand as well as you think you do, or possibly the person who sold you tools has left you with an incorrect impression about how to apply them.

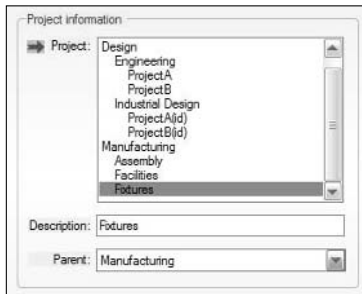
One of the things I talk customers out of regularly is re-creating a Windows Explorer-type folder structure inside the Workgroup PDM vault using the project folders. PDM is an improvement over Windows Explorer, not an extension of it, so the techniques you use in PDM are likely superior to whatever you did, or do, in Windows Explorer. In Windows Explorer, you likely browse for your files. Even Windows has become more sophisticated these days in terms of file access. Finding documents is mostly about search, and little to do with browsing. Browsing can only look for the file name. That's all you see, so that's all you can browse for. When you introduce searches into the mix, you can find things based on other criteria, such as custom properties. In today's CAD world, the custom property metadata is very important and is reused in a number of ways. The more ways in which you can leverage this data, the more valuable it is to you. Custom properties can be reused in Bills of Materials (BOMs), drawing formats, tables, notes, and several other places, including searches.

Therefore, search is the thing. However, how do you set your project folder scheme up to accommodate search? Search can be negatively affected by very large vault sizes, and because SolidWorks Workgroup PDM is not based on a "real" database, searching can take some time for large vaults. You can use the project folder structure to even out the load for search. It is best to have a more even distribution of files rather than having them all clumped into a single folder. For large vaults, you can improve search performance by narrowing down the search to maybe a single top-level folder rather than the entire vault. Search works regardless of the folder scheme. Search would work if there were *no* folder scheme. However, search works better (faster) if you have a folder scheme that narrows it down a little bit. Be aware that these comments are directed at very large vaults. Small vaults are not as affected by speed problems. The definition of large or small can vary, but I define it this way: If your vault speed is noticeably affected by the number of documents in the vault, you have a large vault. If not, it is small. This fuzzy line probably comes up at around 200,000 documents, depending on your server and network. Is there any other type of organization that is not necessarily reflected in the custom properties or other metadata that might be of some value? If you don't need the folder structure to aid search, could it reflect something that you can't express any other way? You can probably think of a few, but my favorite is having the folder structure represent the function of the document. If you have a drawing of a manufacturing fixture, it might be listed under Manufacturing ↔ Fixtures. A model for a part of a consumer product might be listed under Design ↔ Product A. An example of this type of layout, which I might use as a starting point to recommend to an implementation customer, is shown in Figure 23.3.

This layout immediately reminds you that the biggest function of the project folder layout is to organize documents by permission. As I mentioned previously, permissions are the single most important factor to consider when setting up a project folder structure.

FIGURE 23.3

An example of a project folder layout for a working vault



Recognizing limits

One of the things that a consultant will be able to help you with that a salesperson may not be able to see is that different tools have different limits. SolidWorks Workgroup PDM is a *workgroup-based* tool. You should probably not try to push it to encompass a scope that is too large. For example, you should not try to implement SolidWorks Workgroup PDM in a situation in which you have groups working in different facilities (groups on different LANs). Workgroup PDM works best in a setting that has 2 to 25 SolidWorks users, with maybe more read-only access users, and with fewer than 100,000 documents. Don't panic if you have 26 people; these are ballpark numbers, but you should certainly be thinking of a different option if you have 100 people.

These limits of course are flexible, and depend heavily on how your company actually uses the tool. For example, using the vault validation option is great for troubleshooting, but can severely hamper the performance of a large vault. Windows indexing and antivirus scanning of the vault during working hours can also have substantial negative impacts. If you make use of the archiving functionality, removing old versions from the vault, you can improve the performance of your vault and keep it viable for a longer period of time. Using the Working Copy designator instead of many incremental revisions can also help keep the size of the vault down somewhat.

The obvious option for larger user bases is SolidWorks Enterprise PDM, formerly known as Conisio. This book does not cover Enterprise PDM except to mention it as an upgrade path from Workgroup PDM. You may find several other companies other than SolidWorks, or its parent Dassault Systemes, supply high-end file management tools. You may see them called PLM (Product Lifecycle Management), PPM (Product Process Management), or any one of a number of other acronyms that do not add to the functionality of the tool, but do add to the marketing mystique around them. A useful working definition of PLM is a top-down strategy of integrated engineering, documentation, manufacturing, accounting, planning, and project management tools in an enterprise setting. Workgroup PDM would not be considered a part of a PLM system because it cannot be totally integrated into an enterprise tool like SAP (Systeme, Anwendungen, Produkte — German for Systems, Applications, and Products).

When you start out an implementation by pushing a product beyond its limits, the result will not be a good one. The first step in implementing Workgroup PDM is to recognize its limitations, which show up mainly in terms of customizability and scalability (number of users and documents). It is best to implement a tool you can grow into, rather than one you can shoehorn into your process.

The advantages of Workgroup PDM are that it is inexpensive, easy to install, easy to implement, and easy to maintain. If you have a more complex installation, or more demanding requirements, you may not be able to take advantage of the easy option. Advantages of the higher end options are that they are highly customizable, built on a sophisticated database, have capabilities in multi-site implementations, offer workflow management (document routing), and more.

Looking to future development

SolidWorks Workgroup PDM comes with SolidWorks Professional, which is just one level above the base product, SolidWorks Standard. SolidWorks Enterprise PDM is a separately priced item that does not come with any prepackaged bundle.

What this means to you is that SolidWorks Corp. is likely to put more development into the Enterprise product than into the Workgroup product. This can be seen as good or bad, because while it is nice to see new features and additional functionality, stability is also an important thing. The majority of the information in these chapters on Workgroup PDM was also true for the product in 2006, when it had a different name. While this is not a guarantee that the Workgroup PDM product will continue to exist substantially in its current form, it is safe to say that the bulk of development for the past several releases has been focused on the Enterprise product.

Using Lifecycle

The Lifecycle options in Workgroup PDM enable you to establish lifecycle status levels and assign them to documents, and then to assign permissions to groups on a per document basis. For example, you could create status levels such as Concept, Development, Engineering, Manufacturing, Sustaining, Obsolete, Pending, or others to fit your company's business practice and product development process. You can set up the lifecycle options to allow documents to move in only one direction along the list of status levels, or you can allow documents to move back and forth along the list. An example of a situation in which you might want to go back a level might be if you use a "pending" status, where between status levels, or between revisions within a status, you might want to label a document as pending to avoid the Purchasing department going out and making a large order right before a change is made to the part.

You also need to understand that the Lifecycle options exist primarily to support permissions and revisions. You can turn lifecycle on or off, and it is off by default. If you turn lifecycle on after you have started populating the vault, documents that are already in the vault will not get a lifecycle status until they have gone through a revision change. This can pose some difficulties, especially if you assign permissions according to lifecycle status.

Part IV: SolidWorks Workgroup PDM

The lifecycle status setup is probably the most confusing portion of the Workgroup PDM setup. So many possibilities exist. It pays off to create a practice vault and get a feel for different ways to use this very powerful functionality.

Besides assigning a lifecycle status and granting access to documents, you can also use lifecycle to assign different revision schemes to documents at different lifecycle status levels. For example, documents in the lifecycle status Concept might be assigned a Secondary revision scheme, which assigns numerical revisions. The Production lifecycle status might assign Primary revision scheme, which assigns alphabetical revisions. While this way of doing things may not be able to replicate any revision scheme you may have run into in real-world documentation practice, it is flexible enough to give you several useful options.

Cross-Ref

I recommend reading through all the available check box options in the Lifecycle tab as you test settings in your practice vault. These options are found in Chapter 22. ■

For each lifecycle status, you can define the rules for that status level for revision scheme, document permissions, which can change lifecycle status, and what to do when the status does change. To make the settings, you will first select the status level for which you wish to establish rules.

I have found it beneficial to have a status called Pending to indicate that the document is in an approval phase. If you select the Do not allow a document to return to the selected or a prior status option, you will have to use status levels like Pending1, Pending2 because the status time line becomes one-directional (you cannot return to a previous status), and you will need a pending status in between other status levels.

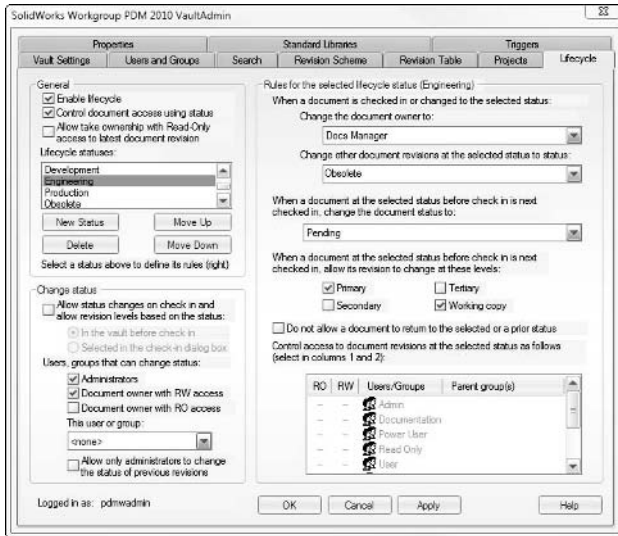
Figure 23.4 shows a sample setup of the Lifecycle tab. It is important to go through the settings on the right to determine what happens to documents at one status that are checked in after a change. I have chosen to have each document go through a Pending status after being checked in, as they wait for the approval of the document manager to be moved forward to the next status.

Once you spend some time with lifecycle, I think you will see its value, but the main question you will need to ask is whether it is worth the extra complication. Generally, I have found that lifecycle works best if your company has a dedicated documentation manager with primary responsibility for the items on this tab.

One very real danger with the lifecycle setup is that you can get lost in the detail and over-specify your process. Be careful to keep an eye on using only functionality that will make you more productive, and shun functionality that amounts to semiautomated bureaucracy.

FIGURE 23.4

A sample lifecycle setup used as a default template



Assigning Permissions

The permission levels you can assign to project folders are read only, read/write, and no access. Ideally, you assign these permissions to groups. If you assign permissions to users, and then add a project folder, you may find yourself going through the permissions to assign or reassign permissions to each user individually, although a Copy Project Permissions option on the Projects tab will help with this type of problem. Users acquire the permissions assigned to the groups to which they are assigned. In the case where a user is assigned to multiple groups, and one group has access and another does not, the more permissive permissions take precedence.

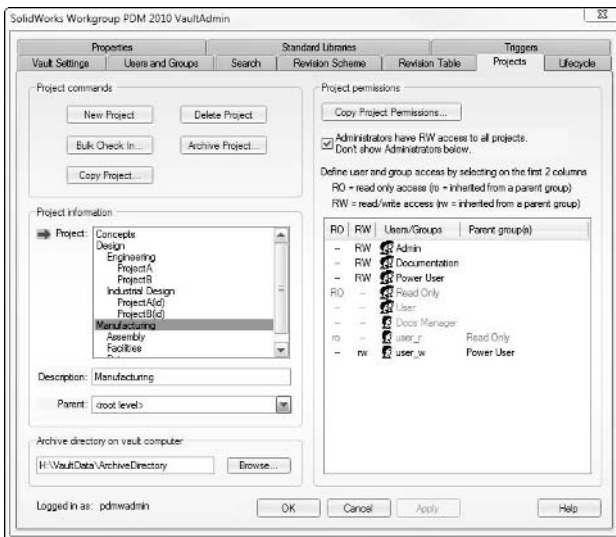
In the window to the right of Figure 23.5, the icons with two heads represent groups and the single heads represent users. The uppercase letters are permissions assigned directly to the user or group, and the lowercase letters are permissions assigned indirectly through group membership. File permissions are color-coded. You may see filenames and properties listed in orange, black, or gray. Orange is read-only, black is read/write, and gray means no access. An option that the vault administrator controls can make files that the user cannot access invisible to that user. The trade-off between not being able to see or not being able to access files is the trade-off between confusion for not being able to find files, or frustration for seeing them but not being able to access them.

Note

The times when you would assign no access should be rare. In most cases, preventing unauthorized individuals from making changes should be enough. If no access is used, I recommend allowing users to see the grayed-out filename, since they may need to know other information about the file, such as its lifecycle status. ■

FIGURE 23.5

A sample projects setup used as a default template



Assigning permissions is essentially a measure of how much you trust your users. If all users were perfect, and did only what you told them to do, then there would be no reason to lock some users out. The more self-disciplined your users are, the less restrictive your permissions need to be. If your users have been using manual file management for some time already, they are probably well disciplined. Users new to the concept of document management may require more restrictive settings to keep them from making unwanted changes to vaulted documents.

Using Toolbox with Workgroup PDM

SolidWorks Toolbox has always raised questions about file management, and when it is combined with Workgroup PDM, additional questions come to mind. Users have a wide range of experiences and preferred solutions when it comes to Toolbox on its own, and more when they are combined with Workgroup PDM. If you ask users about which settings they recommend, you will hear everything from “Don’t touch it” to “It has worked great for me for years.” In this section, I explain the specialized techniques for dealing with Toolbox within Workgroup PDM, and recommended settings.

One of the most frequently heard questions about Toolbox comes from those who are trying to implement Toolbox and make it work with Workgroup PDM. Each package is a field of study unto itself, but combining them can be tricky and introduces options that can affect the rest of your design and modeling workflow.

The biggest difficulties with Toolbox are generally file management ones. Given this, you might think that using Toolbox in conjunction with file management software would solve those problems. To an extent, it does, but it is not automatic nor is it guaranteed.

Specifically, Toolbox by design violates what I consider one of the primary rules of file management: You should never have files with the same name and different content. When you use the configuration setting in Toolbox, it constantly builds configurations for each size of a fastener used and saves them in the same file. For example, you could have one Toolbox file with a given name that on one computer may only have a single configuration or on another computer may have thousands of configurations.

Handling configurations in Workgroup PDM

Part of the issue is how Workgroup PDM deals with configurations; namely, it doesn't do anything special. Workgroup PDM will allow you to search for configuration names, but it doesn't manage configurations as separate documents. (Other PDM products do this.) With Toolbox documents, managing configurations is a concept that would be helpful, but you will not find it in this PDM software.

The best way to use Toolbox parts with or without the Workgroup PDM vault is to have them as a static library rather than a library where the parts keep updating by adding more and more configurations as you require new sizes. Creating a static library means one of two things. You can either pre-create all the configuration sizes or set Toolbox to create separate parts instead of using configurations. Given the fact that Workgroup PDM does not treat configurations as individual documents, it is probably best to use the create parts setting, where each size is an individual part, and each individual part has only a single configuration. If this is not the case, then you may want to consider managing Toolbox with a small group of administrators. All other users would then be required to request any new configurations, which would be built and added to the library by a library administrator.

Recognizing Toolbox parts

SolidWorks Workgroup PDM automatically recognizes files that come from any path including the word "Toolbox" or "SolidWorks Data" as Toolbox parts. It also recognizes documents where the internal flag called `IsToolbox` is set (you can use the utility `sldsetdocprop.exe` found in the SolidWorks director, `\Toolbox\data utilities\` directory). Using the Standard Libraries tab in the VaultAdmin, shown in Figure 23.1, you can also specify additional library folders to be recognized as Toolbox folders.

Part IV: SolidWorks Workgroup PDM

Any documents from these folders are put into a special project folder called Library Components. If you accept the default settings in the Standard Libraries tab of the VaultAdmin dialog box, those parts do not go into the vault (only a shortcut icon is placed in the vault) but are still tracked by Workgroup PDM. This is a nice situation because the files don't need to be in the vault. They are not revision managed. Further, if you were using Toolbox with configurations, putting them in the vault would be a disaster anyway.

The only way you could practically put Toolbox parts in the vault would be to use the Create Parts setting in the Toolbox setup. Unfortunately, Create Parts is not the default setting. Create Parts is one of those settings that you need to have set at the beginning of the life of the Toolbox library. If you start using parts in the library and creating configurations, then switch to the Create Parts setting, Toolbox copies the parts with the configurations in them, and you will have a big mess waiting to happen.

Cross-Ref

Chapter 10 covers the implementation of Toolbox, along with the Create Parts setting. ■

Figure 23.6 shows a view of the vault with an assembly loaded into the vault that has some Toolbox hardware. The hardware text is in a different color (you may not be able to see this in the book's gray scale print), has <Not Revision Managed> after it, and is stored in a separate project folder.

The biggest surprise about the parts in this folder is that they are not really in the vault. Workgroup PDM knows where the parts are: they are tracked within Workgroup PDM to enable them to be added to BOMs, reports, and Where Used lists. The icons are distinctly different from the other revision-managed parts in the vault.

Notice also that the Toolbox parts have a special icon. If you play with this a little you will find that Toolbox parts that you have tried to divorce from Toolbox still show up with the Toolbox icon. This is because the parts retain the internal IsToolbox flag discussed earlier in this section. The `slidsetdocprop.exe` utility can be used to turn the property on or off.

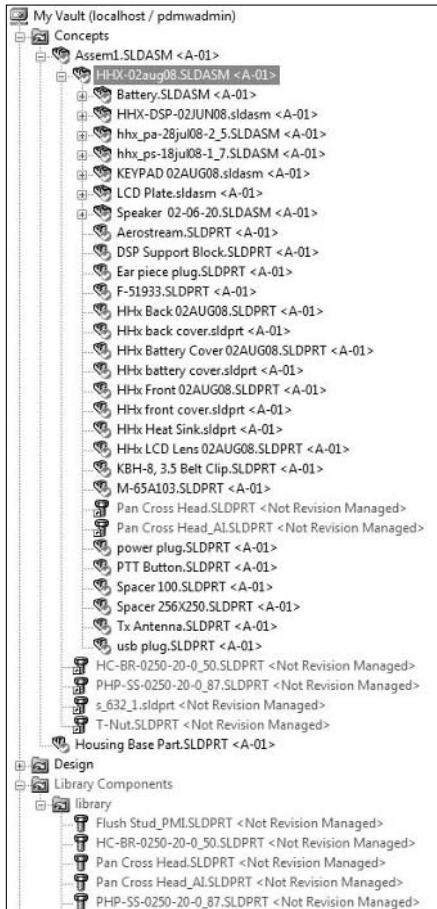
This scenario works fine with configurations in Toolbox, mainly because it is not making a copy of the parts with different numbers of configurations. I still recommend using the Create Parts option in Toolbox, especially for use with Workgroup PDM. This is the foolproof way of running Toolbox.

Dynamic libraries, or libraries that are really configurators, cannot run from within a vault. These libraries must either be created outright into a static collection of files or be managed from outside the vault.

One aspect of this arrangement that is not ideal is that if a Workgroup PDM user has access to the vault but does not have access to the library used for the parts, she cannot get copies of the hardware. For this reason, I think the ideal usage is to use Toolbox in Create Parts mode and put all libraries into the vault. This ensures that anyone who can get to the vault and copy an assembly with all its parts can also get the hardware and library components that go with it.

FIGURE 23.6

A view of the vault with Toolbox hardware in it



Having libraries inside the vault is not a completely bad idea. If you ever move the vault, your libraries go right along with everything else.

Using the IsFastener and IsToolbox properties

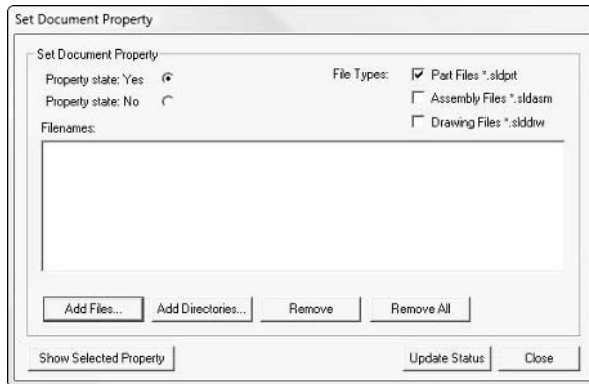
Situations may exist when either you want to remove Toolbox parts from Toolbox and no longer have them recognized as special, or you may wish to have some of your own custom-made parts treated as Toolbox library parts. Parts that are seen as fasteners get the special treatment described earlier by Workgroup PDM, but are also not cut by drawing section views, or included in interference checks. In these cases, you need to use or establish either the IsFastener or IsToolbox property in your library documents.

Part IV: SolidWorks Workgroup PDM

IsToolbox is an internal property that Toolbox uses to let SolidWorks and Workgroup PDM know that a part should be treated as a Toolbox part. If you add this custom property to your own custom parts, your parts will also be treated like Toolbox parts. This is the property mentioned earlier that you can use the `sldsetdocprop.exe` utility to turn on or off. The property cannot be accessed through the normal user interface. This utility is shown in Figure 23.7.

FIGURE 23.7

Set Document Property is used to set the IsFastener property flag.



IsFastener is a custom property that only accomplishes half of what IsToolbox accomplishes. IsFastener only excludes parts from drawing sections and interference analysis and can be turned on by creating a custom property of the name IsFastener with a value of 1. IsFastener and IsToolbox are different types of tools that have overlapping functionality. Do not confuse these.

If you have a large number of files to update with the IsFastener property, you can use the Update Custom Properties function found in the SolidWorks Task Scheduler, or more appropriately, you can use the Set Document Property utility.

Note

The IsToolboxPart property is also effective in preventing parts from being cut in drawing section views. ■

In summary, the special functions of Toolbox parts in Workgroup PDM are as follows:

- Toolbox parts shortcuts go to a special folder, without the actual file (depends on setting).
- Toolbox parts use special icons in the Workgroup PDM vault.
- The IsFastener or IsToolboxPart custom property prevents parts from being cut in section views on drawings.
- The IsFastener or IsToolboxPart custom property prevents parts from being included in interference detection.

Establishing Libraries inside the Workgroup PDM Vault

CAD Administrators have varied opinions on using libraries from inside PDM vaults. You can approach this from several different points of view. One is that the library data is not revision-controlled data, so it does not need to be inside the vault. Another is that revision control is not the only reason to put documents inside a vault. The vault can also be used simply to keep documents in a secure central location where a structured method of access exists. Accessing files from multiple locations does not make much sense, especially when you already have to control write access to library data that no one changes.

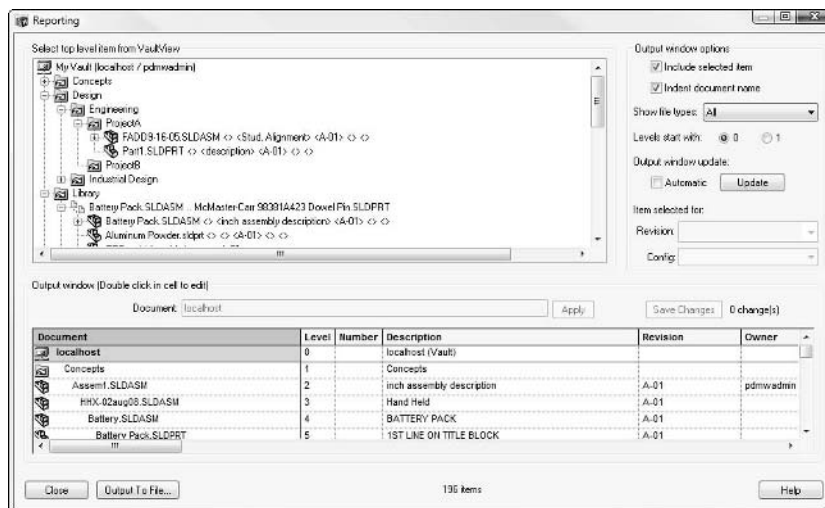
I subscribe to this second way of thinking because it gives your document control a more consistent approach. In addition, the Workgroup PDM interface works well enough as a library interface, with drag-and-drop capabilities. It is an improvement over the search functions available to libraries, and you only have to create and enforce one set of rules for all data, including library data, rather than inventing a completely different set of rules that have to enforce some of what Workgroup PDM would be enforcing anyway.

The downside to the method of putting libraries in the vault is that it only works for library parts and assemblies; it doesn't work for library features. Library features must still exist outside of the vault.

In addition, if your library parts are all in the vault, reports can include them as well. Figure 23.8 shows the Reporting interface.

FIGURE 23.8

Creating a report through the Workgroup PDM interface

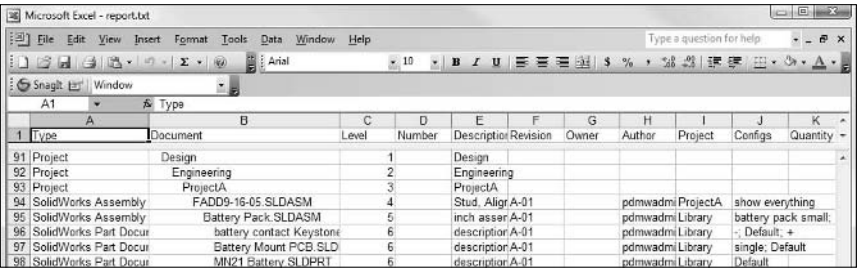


Part IV: SolidWorks Workgroup PDM

You can run reports for the entire vault, or an assembly with all its children, or just a single document. The output formats are text or comma-separated or delimited values. Figure 23.9 shows an assembly with several parts in different folders, including some library parts. This report is shown in Excel, although you can use any text or CSV (comma-separated value) file reader.

FIGURE 23.9

A report showing some library components

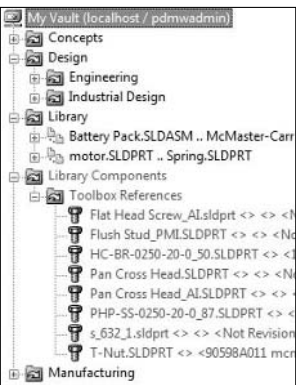


| Type | Document | Level | Number | Description/Revision | Owner | Author | Project | Configs | Quantity |
|----------------------|--------------------------|-------|--------|----------------------|---------|----------|---------|---------------------|----------|
| Project | Design | 1 | | Design | | | | | |
| Project | Engineering | 2 | | Engineering | | | | | |
| Project | ProjectA | 3 | | ProjectA | | | | | |
| SolidWorks Assembly | FADD9-16-05.SLDASM | 4 | | Stud, Aligr.A-01 | pdmwadm | ProjectA | | show everything | |
| SolidWorks Assembly | Battery Pack.SLDASM | 5 | | inch asser.A-01 | pdmwadm | Library | | battery pack small. | |
| SolidWorks Part Docu | battery contact Keystone | 6 | | description:A-01 | pdmwadm | Library | | -. Default; + | |
| SolidWorks Part Docu | Battery Mount PCB.SLD | 6 | | description:A-01 | pdmwadm | Library | | single; Default | |
| SolidWorks Part Docu | MN21 Battery.SLDPR | 6 | | description:A-01 | pdmwadm | Library | | Default | |

One thing you should take note of when using Toolbox with other libraries is that Toolbox will not put links into a folder you can interact with like normal folders. The Display referenced files in a separate project option on the Standard Libraries tab of the Vault Admin tool creates a separate folder called Library Components, which is colored blue and does not show up in the Project Information area of the Projects tab of the VaultAdmin dialog box. This means you cannot add subprojects to that area, and you cannot put documents into it. If you have an area for libraries, it has to be separate from the area reserved for Toolbox parts. Figure 23.10 shows a section of the vault with both regular library and Toolbox library folders.

FIGURE 23.10

User-managed library and Toolbox library storage shown in the vault



You can use items from the library inside the vault as drag-and drop parts into assemblies, the same as if you were using them from the Design Library. Mate References even work such that the parts you drag from the vault folder into an assembly will snap into place on appropriate Mate References or geometry.

Summary

Overall, SolidWorks Workgroup PDM is easy to use, install, and administer. This is especially true when you compare it to a larger, more established enterprise system. Still, for as simple as it is, Workgroup PDM gives you enough options that you can't just install it and turn it on, you have to intentionally plan the implementation.

When establishing vault practices, strive for simplicity with the project folder structure and be especially careful with lifecycle. Be thorough with your usage of SolidWorks custom properties. Remember that the vault structure setup is mostly about permissions, not about browsing. Search eliminates much of the need for browsing, although it can take time for users to form new habits when old ones are deeply ingrained.

Establishing Client Best Practice

Many of the ways end users will use the Workgroup PDM (Product Data Management) software depend on how the Administrator has set it up. As the implementer, you may choose to train your users after you have completed the implementation. You will probably need a four-hour session to teach users how to use the software the way you have set it up. You can use some concepts from this chapter to help build your training material.

In this chapter, I cover concepts that all users should be able to take advantage of, such as the best ways to get the most out of Workgroup PDM, including local setup that end users can control. I also aim to help users avoid some of the common mistakes that allow for local file management problems, and provide some file management troubleshooting skills. If you can recognize some common problems when they happen, you can avoid using incorrect data you may receive at those times.

IN THIS CHAPTER

Gaining access to the Workgroup PDM Client interface

Setting up Workgroup PDM Client options

Accessing the Workgroup PDM User Interface

Workgroup PDM is an easy-to-use document version manager for SolidWorks files and general documents. The user interface consists of the vault view and local view windows, a small toolbar, and the right mouse button (RMB) menu.

SolidWorks provides three ways to access Workgroup PDM Client interfaces: SolidWorks Explorer, the SolidWorks Add-In (Integrated) Client, and the Workgroup PDM Viewer (formerly called the PDMWorks Advanced Server Web Portal). The SolidWorks Explorer client is primarily designed to give

Part IV: SolidWorks Workgroup PDM

non-SolidWorks users, such as IT, Documentation, Production, or management personnel, access to the Workgroup PDM vault, although SolidWorks users often use it as well because it is the most intuitive of all the client interfaces. This interface can check in and out non-SolidWorks documents and check out SolidWorks documents, but it cannot check in SolidWorks documents because it doesn't recognize relationships between these documents. It also has the bitmap preview and eDrawing View window. The interfaces are all different, but similar enough that training on one should apply to all, with the exception that the Viewer will not be able to check in documents or make other changes.

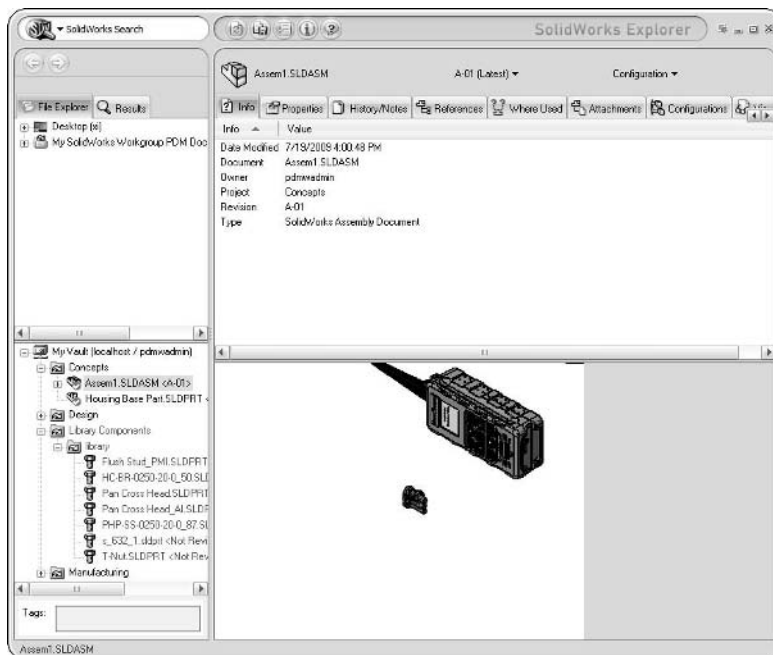
Using SolidWorks Explorer

SolidWorks Explorer serves multiple purposes. It has become the main viewer for SolidWorks data, as well as providing capabilities in basic file management, such as renaming, updating references, and editing custom properties and such. It also serves as the Workgroup PDM stand-alone client to provide users without SolidWorks licenses access to the vault.

When non-SolidWorks users use the SolidWorks Explorer, it requires a separate registration. However, for SolidWorks Office Pro/Premium users, the SolidWorks Explorer is installed with SolidWorks, and no additional registration is required. Figure 24.1 shows the SolidWorks Explorer interface.

FIGURE 24.1

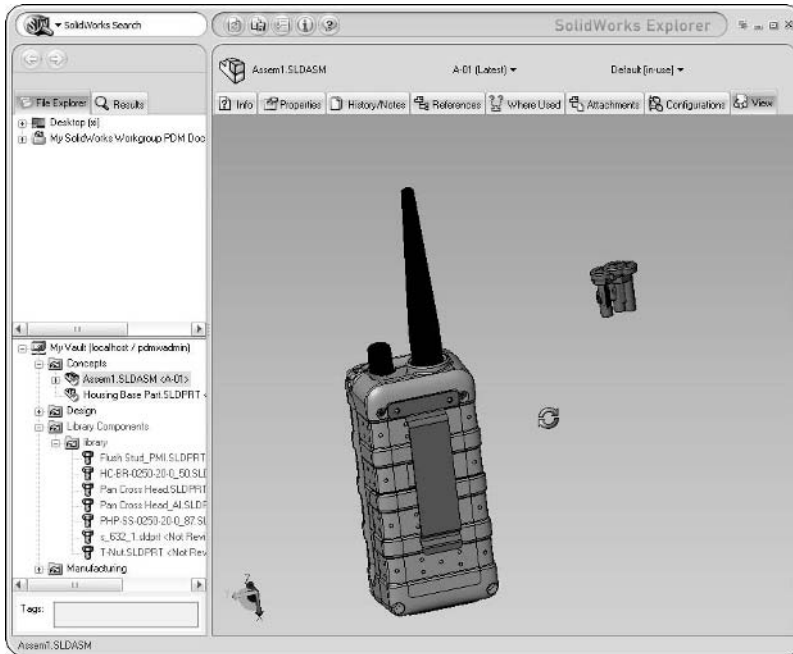
The SolidWorks Explorer interface is the most versatile Workgroup PDM interface.



The upper-left area is the “Local View,” for files on the local computer as well as a view of search results when SolidWorks Search is used. The lower-left area is for the vault. The upper-right window is for various text-based information, and includes tabs for Where Used and References, which enable you to browse up and down the hierarchy of the assembly. The lower-right corner has a View tab that turns into a dynamic eDrawings-based 3D viewer. Figure 24.2 shows the SolidWorks Explorer window being used by the View tab as an eDrawing-based viewer.

FIGURE 24.2

SolidWorks Explorer uses an eDrawing style viewer.



Using the SolidWorks add-in client

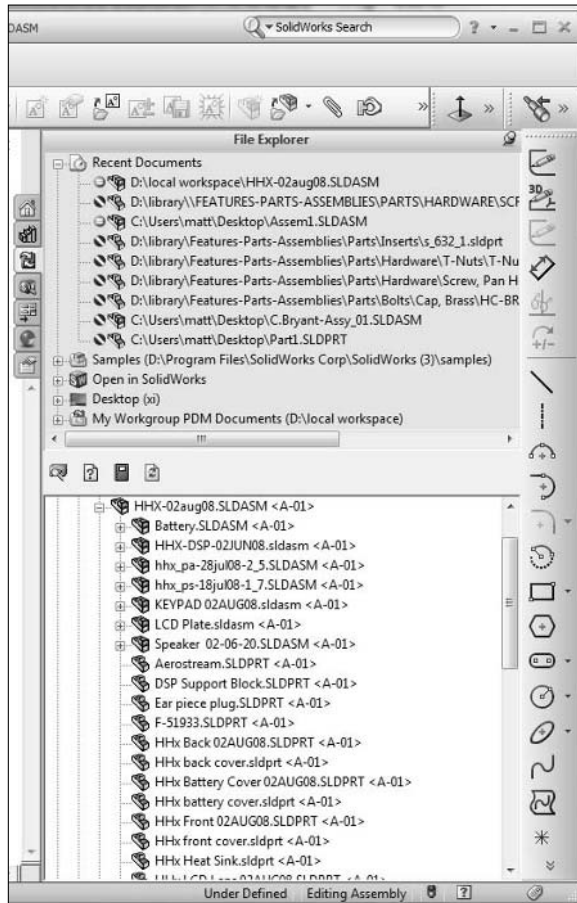
For convenience, the Workgroup PDM SolidWorks add-in client runs in the SolidWorks application window. Because it is an add-in, you must make sure you turn it on using the Add-ins interface. You can access this feature by choosing Tools ➤ Add-ins.

Its window shows up in the Task Pane area, on the File Explorer tab. The tab gets a Workgroup PDM icon, as you can see in Figure 24.3. Like the SolidWorks Explorer, the top section of the window is the Local View and you can use it like Windows Explorer to browse your hard drive. The Local View displays icons that show if documents are in the vault or not, and if so, if they are equal to, older, or newer than what's in the vault. It can also show recent documents and network locations.

Part IV: SolidWorks Workgroup PDM

FIGURE 24.3

The SolidWorks add-in client runs inside the SolidWorks application window.

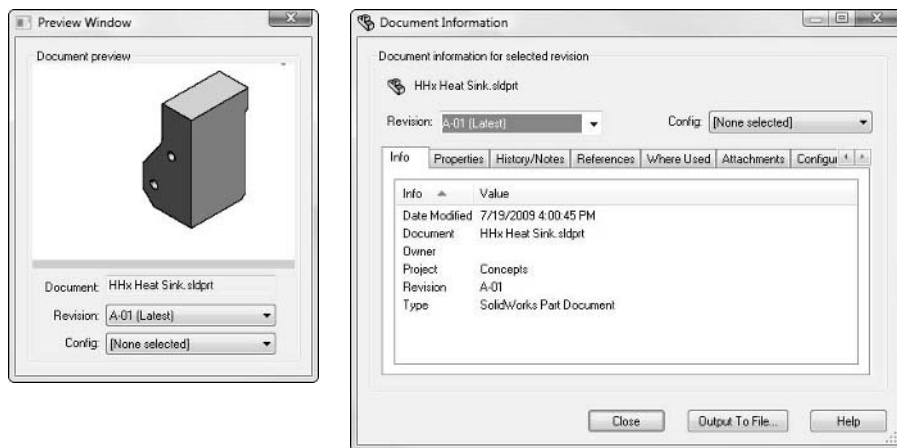


The Preview Window and Document Information window in the add-in client are detached windows, shown in Figure 24.4. They are available via RMB menus. Regardless of which client you use, the entire interface is available through the RMB menus and a toolbar, described next. The add-in client is the only place from which SolidWorks documents can be checked in. This is because being connected to SolidWorks, the add-in client understands the inter-relations between the various SolidWorks documents.

In the Workgroup PDM interface, you can access most functions from the RMB menu or the small toolbars available in both interfaces. This keeps the interface simple and intuitive. If you want to perform some action on a document, just right-click on it, and you should be able to find a command in the list to do what you need to do.

FIGURE 24.4

The Preview Window and Document Information window provide part information.



Establishing Workgroup PDM Client Options

You can access the Workgroup PDM client options from both the add-in interface as well as the SolidWorks Explorer interface. In SolidWorks Explorer, you can click the Options button on the top bar. From the add-in, click the Options link at the bottom of the RMB menu.

The Local tab

Figure 24.5 shows the SolidWorks Workgroup PDM Options dialog box with its various tabs. The Local tab settings affect only the operation of the local client, not the files or the larger Workgroup PDM vault application. Most of these settings can be adjusted by the user mainly for convenience, but there are some that the administrator should have a hand in recommending.

The options are:

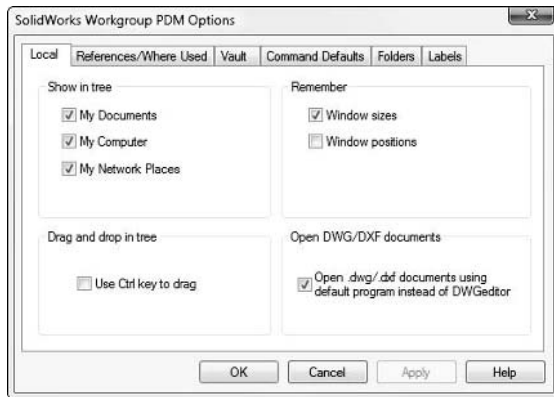
- **Show in tree.** The items in the Show in tree box enable you to control what is shown in the local view box in both the SolidWorks Explorer and the add-in client.
- **Drag and drop in tree.** The Use Ctrl key to drag requires the user to Ctrl+drag a part out of the vault and into an assembly. If this switch is not set, the default behavior is to allow any drag from the vault window to drop a part into the assembly. This can work like Design Library parts, aside from the preview image. Mate references are used when dragging from the vault.

Part IV: SolidWorks Workgroup PDM

- **Remember.** The Remember box enables you to set whether the software should remember window sizes or positions after you close them.
- **Open DWG/DXF documents.** This feature determines if you want to use the DWG editor or another program for DWG and DXF files.

FIGURE 24.5

The Local tab in the SolidWorks Workgroup PDM Options dialog box houses all options that a client can change.



The References/Where Used tab

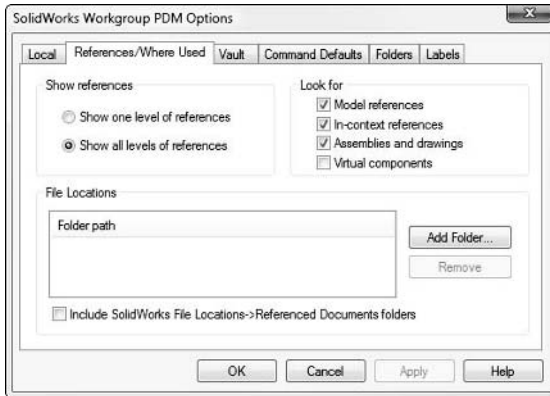
The References/Where Used tab enables the user to indicate where SolidWorks should look for referenced documents. Figure 24.6 shows the References/Where Used tab of the SolidWorks Workgroup PDM Options dialog box.

The options are:

- **Show references.** This box enables you to select how many levels deep you want to show of parts and subassemblies within a top-level assembly.
- **Look for.** These options address the type of references you want Workgroup PDM to look for and report.
- **File Locations.** This tells the Where Used and References functions where they should be looking for parts and assemblies referenced by other SolidWorks documents. At the minimum, users should have the Include SolidWorks File Locations option enabled. This calls search paths from the SolidWorks settings for the same thing. Users should establish a local workspace where all of the vault files go, and where all of the work on the local machine is done. This workspace path should be listed in the File Locations box.

FIGURE 24.6

Settings on the References/Where Used tab

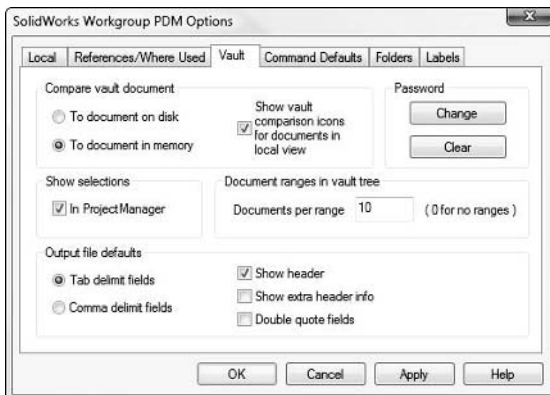


The Vault tab

The settings on the Vault tab of the client options dialog establish how the user sees the data in the vault, and is the location for the user to change his or her password. Figure 24.7 shows the Vault settings on the client SolidWorks Workgroup PDM Options dialog box.

FIGURE 24.7

The Vault tab of SolidWorks Workgroup PDM Options dialog box enables users to control how they visualize the vault.



The options are:

- **Compare vault document.** This refers to what the vault file should be compared against. If you have opened a part and made changes to it, but not saved the file, then the changes are in memory and not on disk. It is safest to make comparisons against files in memory, because parts that have not been saved can still be saved and checked into the vault. Workgroup PDM uses symbols to stand for newer than the vault, older than the vault, and the same as the vault.
- **Password.** These options enable you to force Workgroup PDM to forget your password or to change it altogether. This is essentially a security feature, so that no one accesses your account through the automatic settings on your computer. A person would need to enter your password directly each time she accesses your account.
- **Document ranges in vault tree.** This option limits the number of top-level documents that are displayed. It will create temporary display-only folders limited to the number specified in this box. One of the odd things about this setting is that when it is used and documents and subprojects are seen together at the same level, the subprojects are listed inside the range folders, after the documents. In typical usage, actual documents and subprojects are generally not at the same level inside a Workgroup PDM project.
A setting of 0 disables the function. This function is predominantly intended for display performance when there are a large number of documents.
- **Output file defaults.** Output files come from reports of tabulated data, and these settings control their format.

The Command Defaults tab

The Command Defaults tab enables users to set the default settings for documents when they are checked in or out of the vault. These settings can save time and avoid mistakes, if you understand the settings properly. Figure 24.8 shows the Command Defaults tab of the SolidWorks Workgroup PDM Options dialog.

The options are:

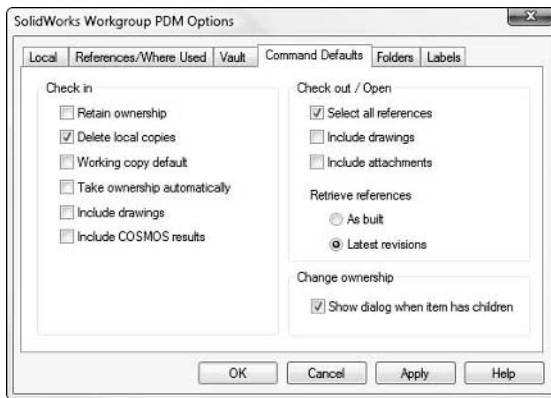
- **Check in.** When you check in a document, you are creating a later revision to put into the vault, or possibly overwriting the latest revision in the vault, depending on settings. You may retain ownership of the document or release it. You may keep the local copy or delete it.
- **Retain ownership.** If this option is selected, it means that after a check-in operation, you will still have ownership of the document(s). If this is unselected, checking in will automatically release ownership.
- **Delete local copies.** This option deletes files from the hard drive when they are checked into the vault. One of the leading causes of version problems with SolidWorks is caused by having multiple copies of the same filename. Selecting this option helps prevent that situation.

Sometimes selecting Delete local copies might cause some difficulties, such as when you are checking in parts that are used in multiple assemblies. If the other assemblies the part is used in are not already checked in, the next time the other assembly is opened, it will not be able to find the part.

Another situation where this could cause problems is when a drawing and the associated part are open, but only the part is being checked in. Workgroup PDM tries to delete the part, but can't do it because the drawing has it locked. In a situation like this, you need to remember to clear out the part when the drawing is checked in.

FIGURE 24.8

The Command Defaults tab enables users to establish defaults.



- **Working copy default.** This option is used in conjunction with Revision Schemes where you add a Working Copy designator to the end of the revision. If you keep designating documents you check in as Working Copies, they will overwrite one another until you deselect the Working copy option.

This is one technique users can employ to save server space from being unnecessarily taken up by many intermediate revisions, while still keeping incremental data secure.

- **Include drawings.** This option enables Workgroup PDM to check in drawings associated with parts and assemblies. For example, if you check in an assembly with its parts and subassemblies, and this option is selected, Workgroup PDM will search the SolidWorks search path (see SW help Index for Search file locations) and include drawings in the checkin.
- **Include COSMOS (Simulation) results.** This option manages the results of SolidWorks Simulation studies, when available.

Part IV: SolidWorks Workgroup PDM

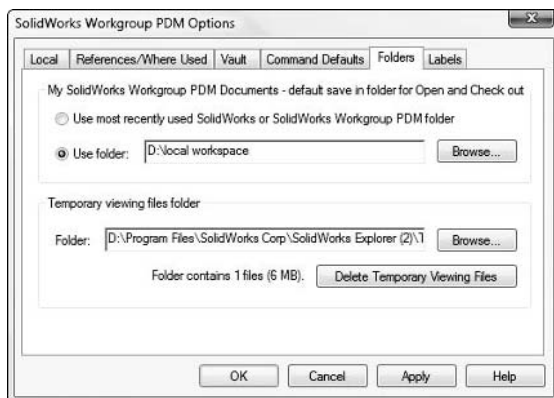
- **Check out/Open.** When you check out or open an assembly, you choose if you want to select all the references, only associated drawings, or other documents attached to the model or drawing files.
- **Select all references.** When checking out a document with references such as an assembly or drawing, all references will also be selected for check out or open as well. Frequently you may want to edit the upper level document while just having read-only access to the references. Leaving this option cleared is the best option so you do not lock other users out of the referenced documents automatically.
- **Include drawings.** When you include referenced documents, the referenced documents may also have drawings. Therefore, an assembly drawing references an assembly and parts, and those parts may also be referenced by other drawings. These other drawings are the ones this option is addressing.
- **Retrieve references.** As built and latest versions are important options in the Retrieve references section. Say an assembly is checked in at rev A, with parts 1 and 2 both at rev A. Then part 1 is bumped to rev B. When you get the assembly, what revision do you want of part 1? The Latest Versions option gives you rev B, and the As Built options gives you rev A. Rarely do companies up-rev assemblies when a part revision changes. If, however, part 1 were swapped out with a new part 3, the assembly would have to be up-revved for this change.

The Folders tab

The folders tab is where the local workspace folder is specified. Administrators should ensure that everyone working in a vault uses the same name and drive for their local workspace so the SolidWorks search path functionality works correctly for finding referenced documents. Figure 24.9 shows the Folders tab of the SolidWorks Workgroup PDM Options dialog box.

FIGURE 24.9

The Folders tab of the SolidWorks Workgroup PDM Options dialog box enables users to control folder display.



The options are:

- **My SolidWorks Workgroup PDM Documents.** This option is often called the local workspace. These files are copied out of the vault onto your local computer for you to use locally, make changes to, and then check back into the vault.

From the administration point of view, it is best to have this folder set to the same local location on each individual Workgroup PDM workstation, so that every user has the same path for the workspace. This is to assist the internal file management scheme in SolidWorks. It always guarantees it is finding the correct files, especially in situations where technique has been sloppy and there are multiple copies of files with the same name scattered on local and network drives.

The Referenced Documents setting, which works hand in hand with this one, is found in SolidWorks at Tools ⇨ Options ⇨ File Locations ⇨ Referenced Documents. That setting should point to the location of the local workspace as well. Again, these Admin functions are addressed in the client section because the settings are done through the client interface rather than through the Admin interface.

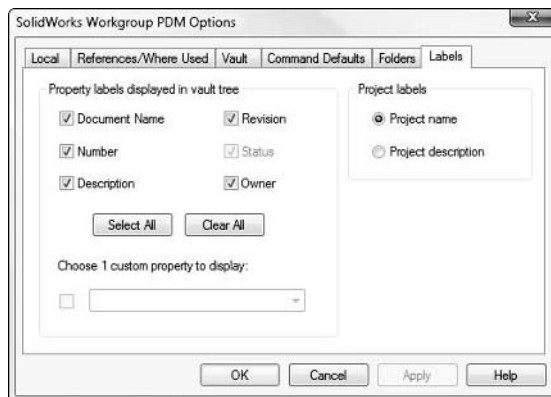
- **Temporary Viewing files folder.** This option enables you to locate and clear the temporary files.

The Labels tab

Figure 24.10 shows the Labels tab of the SolidWorks Workgroup PDM Options dialog box. Labels are what are displayed in place of the project folder names in the Workgroup PDM tree. It is best to be selective here to avoid a confusing display. Project folders can be listed by name or description.

FIGURE 24.10

The Labels tab of the Workgroup PDM Options dialog box controls how documents are displayed in the vault.



The options are:

- **Property labels displayed in vault tree.** Property labels are standard Workgroup PDM properties that can display after the document name in the vault view. Using some of these may make it easier to find information you are looking for such as the latest revision of a document, but showing all of them might result in unnecessary clutter.
- **Choose 1 custom property to display.** The properties in this list are established by the Administrator in the Properties tab of the VaultAdmin tool. The property value displays after the name of the document in the vault view.
- **Project labels.** The order of the projects in the vault view is alphabetized by the project name, even if the description is shown. To show projects in an order other than alphabetical order, you can name projects with numbers in front of them (for example, 1 Pneumatics, 2 Hydraulics), create the description without the number, or show the project descriptive name instead of the actual name. This would cause Pneumatics to be listed before Hydraulics.

You can also display one custom property in the vault view. This enables you to later search on that displayed custom property.

Project folders cannot be renamed. If you want to change the name of the project in which a set of documents are located, you have to create a new project with the desired name, take ownership of all the documents you want to move, and then use the Change Project command from the RMB menu to move the documents to the new project.

Using the Right Mouse Button menu

As with other sections of the SolidWorks software, and more generally with Windows-based software, the RMB context-sensitive menus are very important and play a key role in the intuitive function of the software. The RMB menu in Workgroup PDM contains most of the functionality that you need. Figure 24.11 shows the menu and its options.

Searching in Workgroup PDM

Search is a very important function in Workgroup PDM. In fact, you might say that searching can replace browsing in many situations. You can use two methods for Search. One method is simply to click the Find or Find Next button at the bottom of the Search for Document dialog box, which will highlight documents in the vault view window, one by one. Another method is to click the Generate Report button, which creates a separate window with all the results. Use the first method if you are looking for a single document to open and work on, and use the second method if you are looking for a list of documents. You can set up searches using compound conditions, and then save and reuse them. The search can present the results one by one or in a mass report. If presented one by one, you can select whether to search forward or backward from the current selection in the vault. You can limit the search to the children of a particular assembly or to files within a particular project. Figure 24.12 shows the Advanced Search dialog box.

FIGURE 24.11

The Workgroup PDM RMB menu contains most of the functionality.

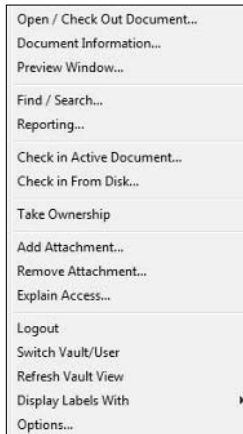
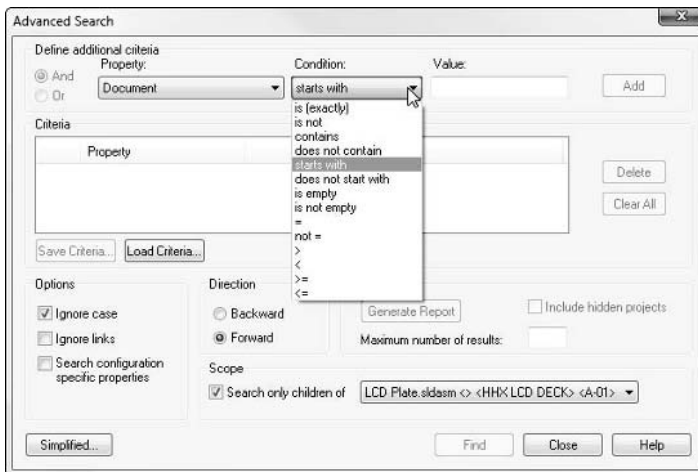


FIGURE 24.12

The Workgroup PDM Advanced Search dialog box can replace most browsing.



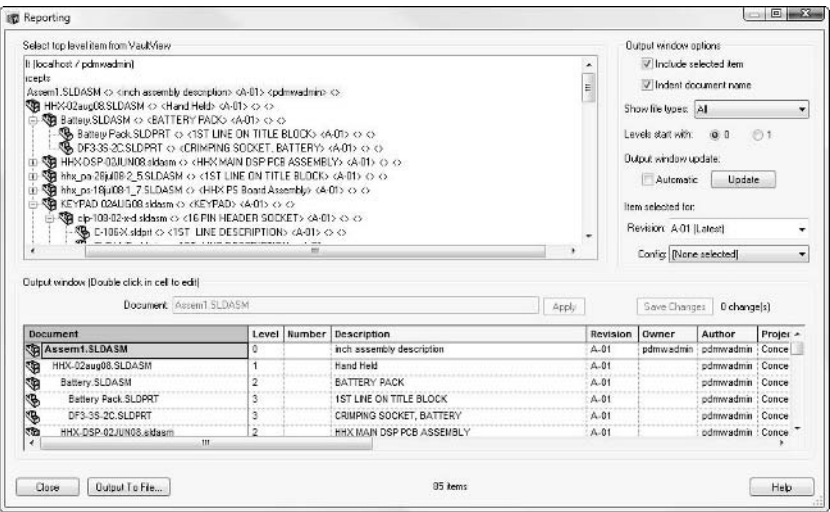
Search criteria are saved in files with the extension *.sqy, and can be saved in network locations for multiple people to use. These make the most sense for specific searches with several criteria that are frequently run and may be tedious to set up.

Running reports in Workgroup PDM

You can run reports on their own or in conjunction with searches. Reports enable you to get tabulated information that can be used like a Bill of Materials (BOM), saved to a text file, csv, or printed. I find the Report function in the Search to be more useful than the one-by-one display. Figure 24.13 shows the Reporting dialog box in Workgroup PDM.

FIGURE 24.13

The Reporting dialog box in Workgroup PDM shows a wide range of data.



A report can list the results of a search, the contents of an entire project, or the children of a particular assembly document. In some ways, the report of the assembly can be used like a BOM, because it reports all the metadata (text-based information) associated with any particular document in the vault.

When you are not using the report for searches, you can use the Report dialog box option to update automatically, which is more convenient than refreshing the report manually. Every time you select a new document or project, Workgroup PDM Report redisplayes the new information. This option is only available when you initiate the report by right-clicking a document in the vault and selecting Reporting.

One of the most frequently asked questions about reports is how to get the Quantity column to display a number. To get this to happen, you have to have an assembly selected, and in the middle right of the dialog box, select a configuration and a revision for the assembly. This is the case even if the assembly has only one configuration and one revision.

Checking in associated documents

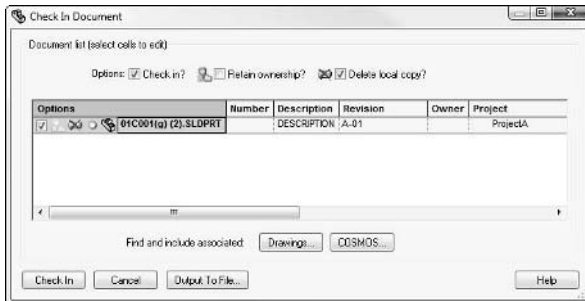
SolidWorks Workgroup PDM enables you to find and include drawings and SolidWorks Simulation (COSMOS) results associated with parts and assemblies being checked in. The best practice here is to have all the documents in the same folder when checking in. If you have to use the searching functions, they can take a long time to find files. In addition, working from your workspace or My Workgroup PDM Documents folder is the best practice recommendation anyway.

SolidWorks documents frequently reference one another, such that assemblies reference parts and other assemblies, and drawings reference parts and assemblies. Parts can even reference other parts for things like inserted parts or mirrored parts. If you check in a top-level document, such as a drawing, Workgroup PDM attempts to check in all the referenced documents as well. Therefore, if you check in a top-level assembly drawing, Workgroup PDM tries to check in all subassemblies and parts. The Find and include associated option, shown in Figure 24.14, enables Workgroup PDM to look for drawings or Cosmos data associated with parts or subassemblies, essentially looking back up the referenced document tree rather than just down. It can only find these associated documents if they are in the same folder as the original SolidWorks part or assembly document that they are based on.

Figure 24.14 shows the Check In Document dialog box in progress, with the options to attach drawings and Cosmos data at the bottom.

FIGURE 24.14

Attaching drawings and Cosmos data



Creating attachments

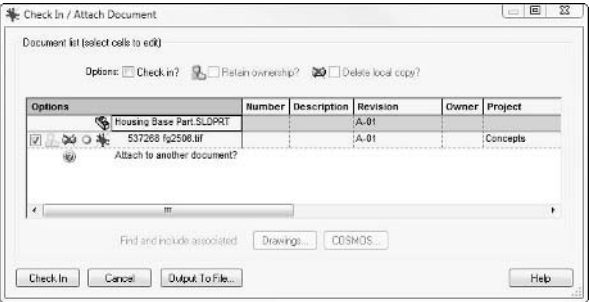
Attachments are non-SolidWorks documents that are attached to SolidWorks documents inside the Workgroup PDM vault. You can create an attachment by dragging a document such as a PDF file from the local view onto the document you want to attach it to, or dragging it into the vault and then manually specifying which document you want to attach it to by clicking the “?” icon.

Part IV: SolidWorks Workgroup PDM

Figure 24.15 shows the Check In/Attach Document dialog box. This dialog box appeared after I dragged an image file from a Windows Explorer window onto a SolidWorks document inside the add-in client vault view.

FIGURE 24.15

Attaching an image to a SolidWorks document is as easy as drag and drop.



Collaborating with other users

Collaboration can mean simply “working together” or it can refer to specific collaboration settings available in SolidWorks. In this section, I’m referring to the specific collaboration settings in SolidWorks. If the Administrator has enabled collaboration, Workgroup PDM can check periodically for new versions of parts in the vault. You can find the collaborations settings in SolidWorks by choosing Tools ⇨ Options ⇨ Collaboration.

Cross-Ref

Chapter 17 has more information on the collaboration functionality in SolidWorks. ■

For the Workgroup PDM user, all the user has to do is keep an eye on the changing status symbols of documents to tell if a particular document is checked out by another user, or if a newer version of the file has arrived in the vault.

Using Lifecycle settings

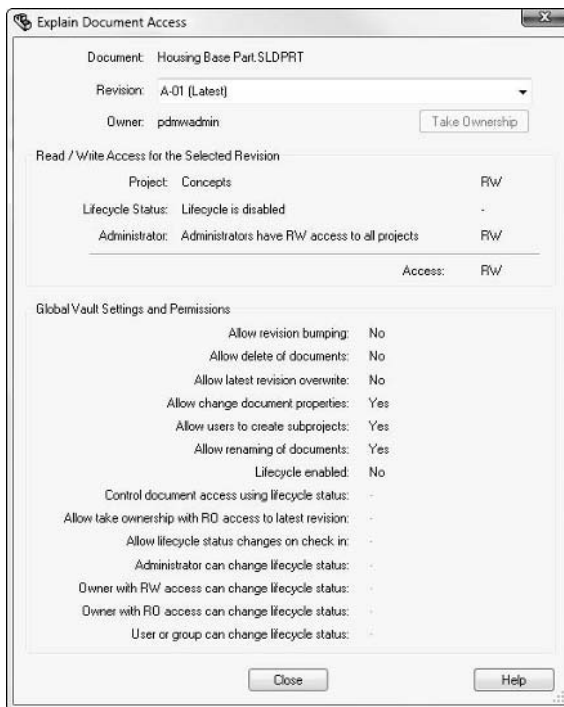
The functionality associated with the Lifecycle settings in Workgroup PDM is so variable from one installation to another that it is difficult to give any generalized recommendations to users about its use. Probably the best advice I can give is to understand how the Administrator has set up the lifecycles and work with the process as intended. The Workgroup PDM Administrator will have to brief you on how it is intended to work. Lifecycle settings generally control permissions to documents by lifecycle status, and there may not be anything for users to do other than check in files to

the vault and allow the system to control the rest of the process automatically. Using Lifecycle to control document access can sometimes result in users not being able to access files. If this happens, users need to contact the Administrator to make sure the process is running as designed.

You may also ask the Administrator to chart out permissions by group and lifecycle status. Many times access problems in Workgroup PDM can be traced to unintentional lifecycle permissions settings. If you are having access difficulties, try using the Explain Document Access dialog box, shown in Figure 24.16. You can access this through the Workgroup PDM RMB menu.

FIGURE 24.16

The Explain Document Access dialog box shows how the current permissions were assigned.



Distilling recommended SolidWorks settings

Most of the SolidWorks default settings that install out of the box are good for general purposes. However, if you or your coworkers become accustomed to working in a particular way, you may want to have your custom settings handy (by using the Copy Settings Wizard in the Windows Start menu).

Part IV: SolidWorks Workgroup PDM

One setting that reflects a particular way of working is the Referenced Document setting. The paths at Tools ⇨ Options ⇨ File Locations ⇨ Referenced Documents should be directed to your Workgroup PDM workspace folder. This works best for people who make use of a local workspace. The local workspace is a great idea for a best practice, because it helps you avoid common file management mistakes with the files you have retrieved from the Workgroup PDM vault.

The SolidWorks Collaboration settings are useful when multiple people need to work with the same files. You can find these settings by choosing Tools ⇨ Options ⇨ Collaboration.

Cross-Ref

You can find more information on the Collaboration settings in Chapter 17. For more general, recommended SolidWorks settings, see Appendix C, where all the Tools ⇨ Options settings are listed and defined. ■

Getting the most from the software through general options

The best way to use software is rarely obvious. You never seem to learn the best way until you have used it for a while and made some mistakes. The tips that follow are intended to help you shorten that learning curve somewhat. While Workgroup PDM is a relatively easy software program to learn and use, you may find these tips helpful.

- During check-in or in dialog boxes that show the grid view, you can enter in a value for all documents at once by clicking a column header. For example, to change the check-in project for all documents, double-click on the project column header and enter a value. This value is propagated to all documents being checked in with this operation. To change individual documents, click in individual cells.
- Use the RMB Refresh Vault View menu option to make sure you are seeing the latest versions of documents in the vault and the latest settings done in the VaultAdmin tool. Remember the SolidWorks collaboration options enable you to set how frequently SolidWorks checks for updated documents.
- Open the Preview window before you do a Search. This enables you to see previews as you toggle through found documents.
- Click the Generate Report button in the search dialog box, which enables you to do a secondary search by double-clicking on any of the column headers.
- The Rename Document function on the RMB menu in the ProjectManager updates references inside the vault without checking out the documents (must be enabled in VaultAdmin). This means that changing names while documents are inside the vault is far easier than renaming them outside the vault.
- If a function appears to be unavailable (grayed out), it may be that you need to have ownership of the document to perform the function or the function has been disabled in the VaultAdmin. If you are unable to take ownership, it might be that you do not have write access to the project or folder, or that there is some lifecycle logic for an access restriction. Try using the Explain Access option in the RMB menu.

- If you can't find a function, try the RMB menu. If you still can't find it, it may be that ownership is required to use that function, or it has been disabled by the Administrator.
- You can do multiple selections the same way you do in Windows Explorer, using combinations of Ctrl+ and Shift+ select.

Summary

End-user training is going to require some customization on the part of the Administrator after you have done a detailed implementation of the production vault you intend to use. Most of the end-user interface can be found on the RMB menus.

End users may run into some local file management problems if they don't follow some general advice. Users should always use a local workspace and download everything they use from the vault into that workspace. The user should keep the workspace clean by using the Delete local copies option, and clearing it out from time to time.

Part V

Appendixes

The appendixes in this book cover a range of ancillary data from where to find help and how to implement the software to a detailed and complete list of the available options. The Tools⇌Options list in Appendix C is a comprehensive look at the available settings in SolidWorks, something every administrator needs to have handy.

IN THIS PART

Appendix A
Online Help Resources

Appendix B
Sources for Standards

Appendix C
Tools, Options Menu Settings



Online Help Resources

As a CAD Administrator, one of the most valuable commodities you have is information. The Internet is a wealth of easily searchable information, but it can also be information overload if you don't have a guide to help make sense of it all.

The resources listed here all are of some value to CAD Administrators, and in particular SolidWorks Administrators. I encourage you to browse through the resources, sign up for membership in various forums, and participate in any available social media that seems relevant to you.

Accessing the SolidWorks Customer Portal

The Customer Portal (see Figure A.1) has several areas of use to end users and SolidWorks Administrators alike. This area of the SolidWorks Web site requires a valid login. Some areas of the portal are available to everyone, and other areas are only available to customers with valid subscription service. You can access the SolidWorks Customer Portal at <http://customer.portal.solidworks.com>.

IN THIS APPENDIX

Researching the SolidWorks Customer Portal

Accessing information from SolidWorks blogs

Getting help through non-SolidWorks blogs

Connecting with SolidWorks users through social media sites

FIGURE A.1

The SolidWorks Customer Portal home page



The Customer Portal was used to research topics for this and my other books, as well as blog articles and general SolidWorks information.

Here are some of the options on the home page:

- **Knowledge Base.** The SolidWorks Knowledge Base, under My Support category, enables you to search on words and phrases found in technical articles, software performance requests (SPRs), or SolidWorks Web sites.
- **My SRs/My SPRs/Fixed SPR List.** SRs are known as service requests, and SPRs are software problem reports. SRs are the report that is initially created when you make a tech support call to your reseller. If they identify a problem that requires further action (a bug fix), the SR becomes an SPR. In all cases, SRs and SPRs are linked to your SolidWorks serial number so you, your reseller, and SolidWorks can track whom to notify when the fix is completed.

The My SRs and My SPRs sections of the SolidWorks Customer Portal enable you to see a list of all the reports for your serial number. This is useful because if you have reported a bug, this list shows the status of the bug and if it has been fixed, and what version of SolidWorks it was fixed in.

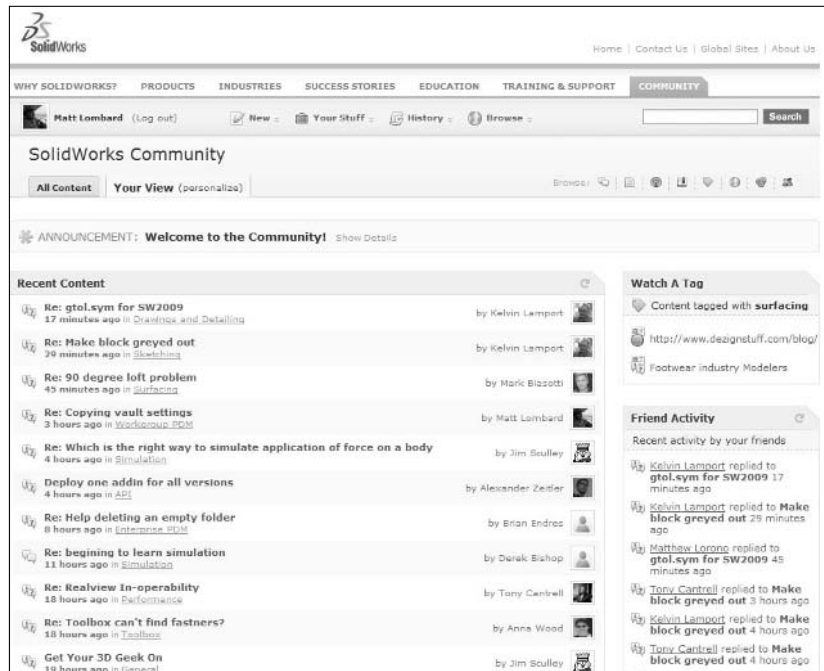
- **My Products.** The My Products section of the Customer Portal helps you see what serial numbers you have and which level of SolidWorks they are for, who the end user is, and registration dates.
- **Register My Products.** The Register My Products area of the Customer Portal enables you to do the initial registration on a new serial number, and to associate a user with it.
- **Downloads and Updates.** Downloads and Updates is the area you browse to if you need to manually download service pack files or other downloadable data. Ideally, you access this data from the SolidWorks interface through the SolidWorks Installation Manager.
- **3D Content Central.** 3D Content Central is the location from which you can download all sorts of vendor-supplied models of standard catalog components. It also has an area for user-submitted parts.

- **System Requirements.** The System Requirements typically list the bare bones minimum under which SolidWorks can be expected to work. It is a good place to go to find out about the availability of a version of SolidWorks for a specific version of Windows. This area also has the graphics card requirements, which is necessary to look through before buying a new workstation or SolidWorks laptop.
- **Admin Guides.** The Admin Guides area has several sections for installing some of the associated product documentation such as What's New, Release Notes, and Enterprise PDM documentation. These are not extensive, but simple overviews mainly of obvious or basic information.
- **Forums.** The SolidWorks Forums are my favorite destination within the Customer Portal. The Forums are full of great advice and examples for a wide range of functionality. Sometimes SolidWorks employees answer questions. For more application specific questions, expert users are happy to answer many of your questions. The Forums cover just about anything you can think of in the SolidWorks software. This is the most active forum on the SolidWorks topic available anywhere on the Web, and I believe it has the highest quality information as well.

Figure A.2 shows the main page for the Discussion Forums. It doesn't show everything available, but you get the idea that there are plenty of discussion topics and a lot of participation.

FIGURE A.2

The front page of the Discussion Forums area



- **Certification.** The SolidWorks certification program is growing to include tests that are more specialized. As of this writing, the available certifications include CSWA (Certified SolidWorks Associate), which is primarily intended for beginners and students, and CSWP (Certified SolidWorks Professional), which consists of a three-hour test of basic modeling skills. Tests in sheet metal, surfacing, and FEA are also available. An Elite CSWP exam is said to be in the works as well, which would offer a more advanced exam. The specialized tests seem to be the better tests and the more valuable certifications, and they cost less. Specialized areas will distinguish your skills from other users the best.

As a CAD Administrator, certification in specialty areas can be a great goal to help motivate CAD users. This test can also be a great way to evaluate skills, as long as you take them for what they are. For the certification site, go to www.solidworks.com/sw/mcad-certification-programs.htm.

Getting Information from SolidWorks User Blogs

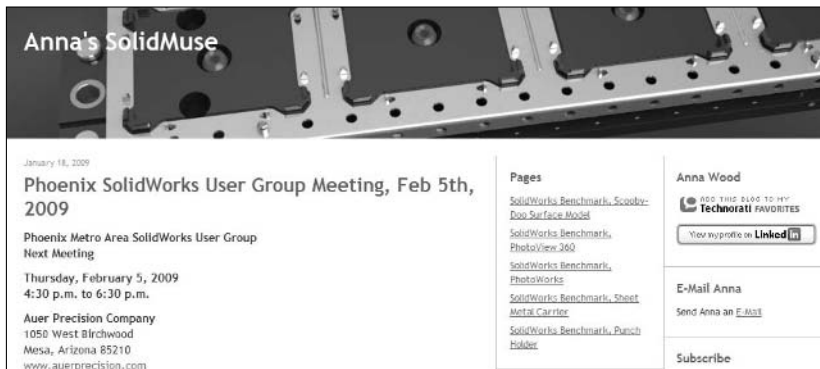
SolidWorks users write several blogs that cover different types of information. The amount of original or relevant material in each blog varies, and you can subscribe to the ones you like to read. You can use many mechanisms to subscribe from NetVibes to Google Reader to Mozilla Thunderbird to Outlook. RSS (simple syndication) is one of the options for subscription, and the blog posts can be sent directly to your e-mail program or on-line RSS reader.

Following is a list of some of the SolidWorks user blogs that are the most active and have been around the longest.

- **Anna's SolidMuse.** Go to www.solidmuse.com/ where Anna Wood writes about benchmarking and computer hardware issues, as shown in Figure A.3.

FIGURE A.3

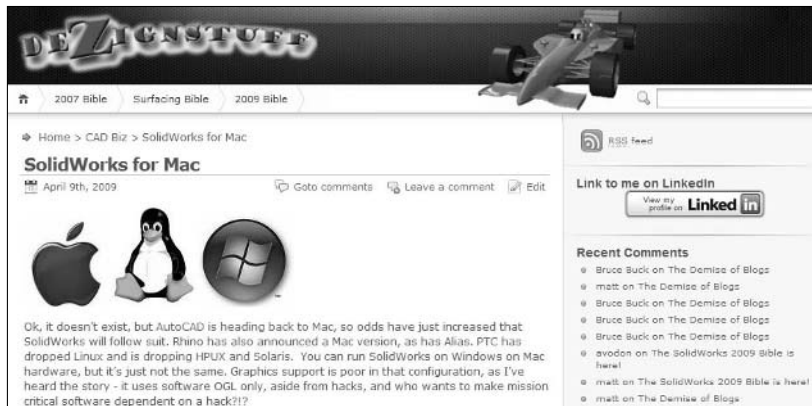
Anna Wood's SolidMuse blog



- **CAD Fanatic.** Go to www.cadfanatic.com/ where Brian McElyea writes about various SolidWorks technical issues.
- **Devon Sowell.** Go to <http://designsmarter.typepad.com/devonsowell/> where Devon writes about general SolidWorks and PDM topics.
- **Dezignstuff.** Go to <http://Dezignstuff.com/blog> and check out my blog, specializing in surfacing and CAD Administration topics (see Figure A.4).

FIGURE A.4

The Dezignstuff blog



- **Ricky Jordan.** Go to www.rickyjordan.com/ where Ricky writes about advanced SolidWorks issues (see Figure A.5).
- **Rob Rodriguez.** Go to <http://robrodriguez.com/wordpress/> where Rob writes primarily on rendering topics (see Figure A.6).
- **SolidSmack.** Go to www.solidsmack.com/ where Josh Mings writes about random SolidWorks topics and entertaining bits from the world of tech and techno-fiction.

Blogs also range into the world of opinion and controversy where other types of media cannot go. This can be a benefit or a detractor because sometimes the information is something you cannot get any other way, and sometimes it simply devolves into silliness. While blogs are not necessarily the best choice for technical assistance, they do from time to time post tutorials or challenges where multiple people contribute different techniques and styles.

One of the best things about blogs is that news and event reports are reported immediately, as opposed to traditional media outlets, which might take a month to report on an event like SolidWorks World.

Part V: Appendixes

FIGURE A.5

Ricky Jordan’s blog front page

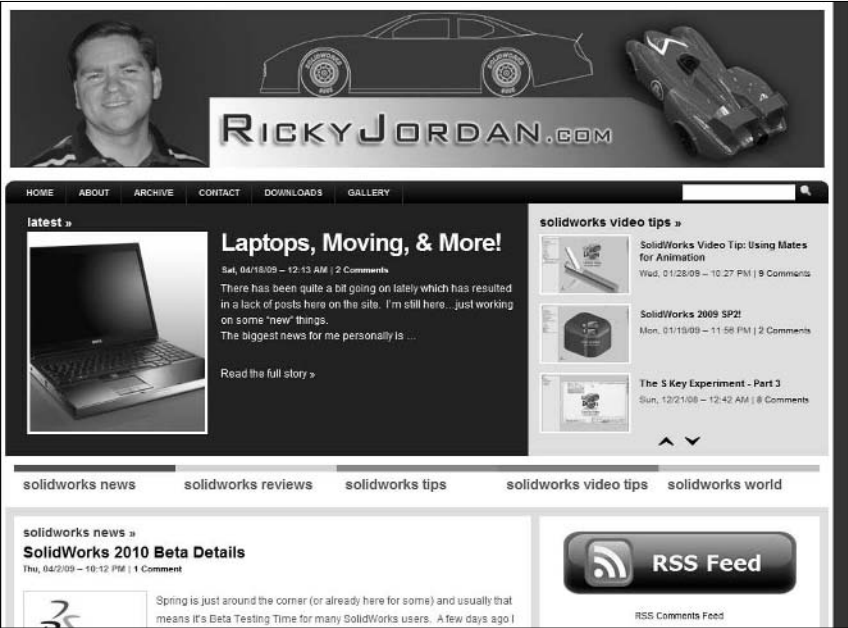


FIGURE A.6

The front page of Rob Rodriguez’s rendering blog



Blogs also serve as a jumping-off point into a diverse world of information on engineering, technology, and design. Each blogger maintains a list of links to other interesting sites. There is a lot of overlap between the various bloggers, but you are sure to find something different on each blog.

Accessing Help on Non-SolidWorks CAD Administration Blogs

You can find many non-SolidWorks blogs that deal with the topic of CAD Administration. They are mostly AutoCAD or Revit blogs, but some of the techniques mentioned have cross-platform value.

- **Robert Green CAD Manager blog.** Go to www.cad-manager.com/ and chat with Robert Green who is well known in AutoCAD Administration circles.
- **Mark Kiker CADDManager blog.** Go to www.caddmanager.com/CMB/ where you will discover a lot of information on standards.

For links to many non-SolidWorks CAD related blogs, visit the Novedge Pulse. This site is called an aggregator, which pulls together snippets from many blogs on topics relevant to CAD users. The URL for Novedge Pulse is www.novedge.com/pulse.asp.

Using Social Media Sites to Connect

Social media is something that cannot be ignored by people who need technical connections to manage technology. Whether Twitter itself has any technical SolidWorks content, you will at least be able to find people there who have the information, knowledge, or skills you may need at some point. Facebook and LinkedIn may not include a lot of explicit technical content, but, again, you can connect with groups of people affiliated with the software and tools you need.

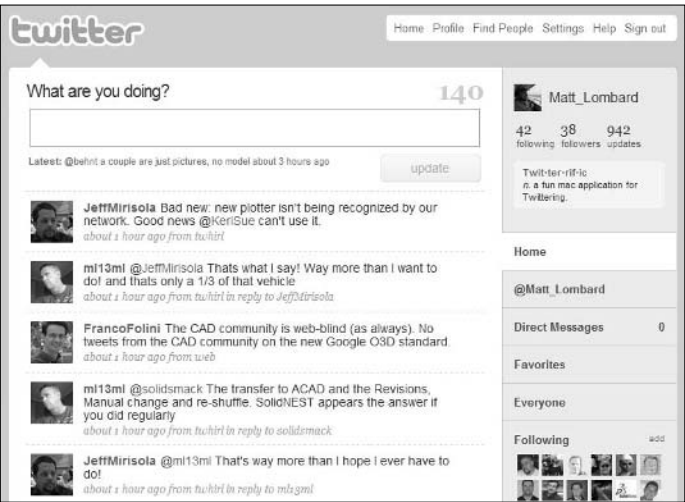
Twitter

Twitter has value for connecting with an active community of SolidWorks users (see Figure A.7). You will not find a lot of direct SolidWorks activity or discussion on Twitter, but you will certainly be able to meet people from across CAD platforms who are experts in their fields.

Part V: Appendixes

FIGURE A.7

A sample Twitter session



Facebook

Again, Facebook does not have a lot of direct technical content, but it has groups of SolidWorks users, Certified SolidWorks Professionals (CSWPs), SolidWorks Surfacing, and so on. Facebook leans toward personal networking rather than professional networking, so be careful not to mix the two inappropriately, but certainly take advantage of the opportunities for introductions that the site is so well known for (see Figure A.8).

FIGURE A.8

A sample home page on Facebook



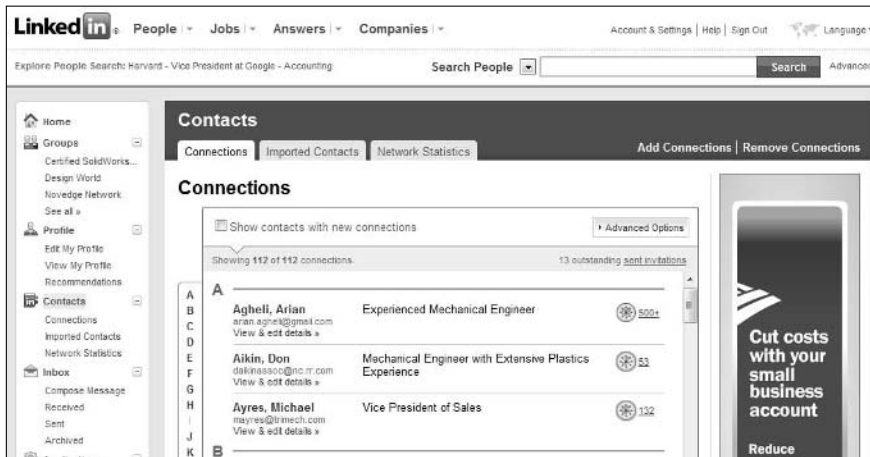
LinkedIn

LinkedIn is the professional social media site of choice. It is much like Facebook except that it is professional and more conservative. It is a great place to ask questions about specific topics, and get introductions and recommendations. You will find plenty of SolidWorks, CAD, and engineering professionals on LinkedIn. If you are familiar with the concept of six degrees of separation, this is the place to test that out. Your contacts have contacts, who know someone, who... and on and on.

LinkedIn is a site that does not have to be addictive or take all your time like Twitter or Facebook (see Figure A.9). I use it for making connections with people, reading about backgrounds, and seeing who has moved to a new job in the last couple of months.

FIGURE A.9

A sample Connections page from LinkedIn



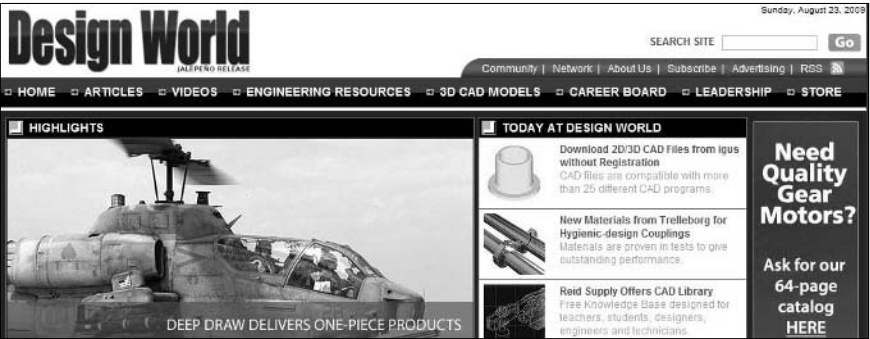
Design World

Design World (www.designworldonline.com/) has an entire suite of online content for several different CAD packages (see Figure A.10). The Engineering Exchange (www.engineeringexchange.com/) offers forums on CAD and technical design topics, 3D CAD Tips (www.3dcadtips.com/). Design World offers a wide range of online and print publications in various design fields. You're sure to find something of interest to you on this site.

Part V: Appendixes

FIGURE A.10

The front page of the Design World suite of sites full of content



Sources for Standards

CAD standards are designed to make interpretation of drawings uniform across an industry. Standards for architecture, schematic, and mechanical drawings have different goals, and each type of drawing looks different. Traditionally, standards are called *drafting standards*, and usually refer to the presentation of data and symbols on paper drawings, and may refer to layers, colors, and line weights on AutoCAD documents.

As a SolidWorks Administrator, you will need to find a way to apply standards to SolidWorks 3D data in addition to the 2D documents SolidWorks can produce. Remember that when you establish a standard, that standard is supposed to serve you, not the other way around. Don't create a standard for the sake of a standard; create it to make your job easier, more repeatable, possibly more automatable, and easier for your customers or suppliers to interpret consistently. Standards shouldn't be hard; they should be easy and intuitive.

You will likely need to adapt any standard you find from the sources that follow for your purposes. You can also probably trim out a significant portion of existing standards for use at your company. It all depends on what you are trying to achieve. Most CAD standards are structured around AutoCAD layers, text sizes, line thicknesses and colors, and so forth, which you can certainly output from SolidWorks; but SolidWorks standards should include far more than just DWG output formats.

Although layering for fonts, colors, linefonts, view data, and so on were critical for segregating data in 2D environments like AutoCAD, depending on your requirements, you may not need to use any layering scheme at all. You can manipulate SolidWorks data, especially at the drawing level, using selection tools and filters to hide and show just about any SolidWorks object.

IN THIS APPENDIX

**General Services
Administration**

**American Society of
Mechanical Engineers**

**American National Standards
Institute**

**International Standards
Organization**

**National Institute of Standards
and Testing**

If you are looking for existing standards upon which to base your own, here are some suggestions.

General Services Administration

The General Services Administration (GSA) has a Web site from which you can find CAD standards for different purposes (www.gsa.gov/cad). Many of the standards that you find from here are architectural or civil engineering related, and come from the Public Buildings Service in the form of AutoCAD drawings, acceptable drawing scales, and AutoCAD R14 color, layer, and line-type standards. This site also includes title blocks, again in *.dwg format.

American Society of Mechanical Engineers

The American Society of Mechanical Engineers (ASME) has a lot of helpful information on their Web site, and it is especially directed at students. A publication in PDF format on their Web site (www.asme.org) offers a nice definition of a standard:

A standard can be defined as a set of technical definitions and guidelines — “how to” instructions for designers and manufacturers. Standards, which can run from a few paragraphs to hundreds of pages, are written by experts.

The ASME is the keeper of many standards that are used very frequently in industry, so a general familiarity with these standards is a good idea.

The following are basic Y14 standards:

ASME Y14.1 — Drawing sheet size and format

ASME Y14.1M — Metric sheet size and format

ASME Y14.100 — Engineering drawing and practices (lists or references all “Y14” STDs)

ASME Y14.2 — Line conventions and lettering

ASME Y14.3 — Multi-view and sectional view drawings

ASME Y14.4 — Pictorial drawings

ASME Y14.5 — Dimensioning and tolerancing

ASME Y14.24 — Types and applications of Engineering drawings

ASME Y14.34 — Associated lists

ASME Y14.35 — Drawing revisions

ASME Y14.38 — Abbreviations

ASME Y14.41 — Digital product definition drawing practices

ASME Y14.42 — Electronic approval systems

The following are Y14 specialty standards:

ASME Y14.6 — Screw threads

ASME Y14.7 — Gears and splines

ASME Y14.8 — Casting and forgings

ASME Y14.13 — Springs

ASME Y14.18 — Drawings for optical parts

ASME Y14.31 — Undimensioned drawings

ASME Y14.32 — Ground vehicle drawing practices

ASME Y14.36 — Surface texture symbols

ASME Y14.37 — Composite part drawings

ASME Y14.40 — Graphic symbols

ASME Y14.43 — Dimensioning and tolerancing of functional gages

ASME Y14.44 — Reference designations

American National Standards Institute

The American National Standards Institute (ANSI) Web site (www.ansi.org) has a section called the Standards Store (click the Access Standards link) that enables you to search and purchase specific printed standards. An example of a standard you can get here is the International Organization for Standardization (ISO) 9001 standard, which is used as a certification for design, manufacture, and quality documentation. It is helpful to become familiar with popular standards such as ISO 9000.

American Society of Testing and Materials

The American Society of Testing and Materials (ASTM) Web site (www.astm.org/) is dedicated to material specifications and standards, including test requirements.

Deutsches Institut für Normung (DIN)

The German equivalent of ASME is located at www.din.de/cmd?lang=en&level=tpl-home&contextid=din&languageid=en.

International Organization for Standardization

You can also find the International Organization for Standardization (ISO) standards on the ISO Web site (www.iso.org). It is helpful to find particular standards if you have their numbers, but you can also search on text such as “geometric dimensioning.”

Japanese Industrial Standards (JIS)

The Japanese equivalent of ASME is located at www.jisc.go.jp/eng/pj/index.html.

National Institute of Standards and Testing

The National Institute of Standards and Testing (NIST) Web site (www.nist.gov/index.html) has a lot of information on standards and certifications.

Society of Automotive Engineers International

The Society of Automotive Engineers (SAE) Web site is at www.sae.org/servlets/index. The organization now covers many other aspects of engineering. Along with automotive standards, it covers aerospace. One of the biggest changes is the collaboration between SAE and other organizations. This aspect of SAE has produced documents identified as SAE-AMS (Aerospace Material Specification), SAE-MS (Military Specification), SAE-AS (Aerospace Standard), and so on. The collaborative efforts result in “industrialized” standards and specifications.

Concerning ASME or Other Standards

As you work your way through the material in this book, you are preparing to lay the groundwork for your CAD environment. During this setup phase, you must include and satisfy your company’s existing processes, standards, and procedures, and they may or may not adhere to, or be based on, military and/or industrial standards, practices, or methodologies.

Establishing the Need for Standards

Throughout this book, I have listed reasons for creating standards. The following is a more comprehensive list of reasons for creating and adhering to standards, which may apply to some companies and not to others.

- You won't have to continually "reinvent the wheel."
- Why spend the time defining all your company requirements when you can follow tried and true ways that have been tested and proven over time?
- To settle disagreements.
- If ever you need to defend or explain how or why something was done a certain way, you can define chapter and verse from the specification, or standard to which you adhere. You are providing yourself with credibility.
- You may be under contract.
- This is true for many companies, but not all. However, you may receive a job that has a Statement of Work (SOW) that specifies certain standards and specifications must be followed. Maybe all drawings must be generated and supplied using geometric dimensioning and tolerancing (GD&T) and that all materials and finishes must be per ASTM or other recognized standards. If you are already there, then you're all set. If you do not conform to most standards, then you may have lost opportunities.
- Provide for consistency and to portray professionalism.
- If drawings are consistently created, then they can be consistently interpreted and, thus, consistently produced or manufactured.
- You must create clear and concise documentation that all drawings, from all employees, look uniform — like they come from the same company.
- Depending on your company's requirements, certain questions may already be answered while others may provide input or direction, and in some cases, changes will need to be made.

Tools, Options Menu Settings

A program with the complexity of SolidWorks is going to have many settings. You can access most of these settings in the Tools ⇨ Options menu. These include system settings that affect only your local computer and document settings (template-based) that are saved with the file. You will not likely need to know all of the settings unless you are doing CAD administration, consulting work, or writing a *SolidWorks Bible*, but this reference will familiarize you with the most important settings. All settings shown in the figures are the recommended settings for general use. Recommendations always change with the situation, and so you should only use these recommendations as starting points.

Be aware that from version to version, the options may change names, change locations, or be removed altogether. I have not documented settings that have been moved or removed.

System Options

System Options apply only to your local computer, but they also apply to every document that you open. The settings are not saved with the files, so be aware that when changing the System Options, other users on different computers will not necessarily see things the way that you see them.

Settings that are saved with the file and used on other computers are called Document Properties, and are discussed later in this appendix.

IN THIS APPENDIX

System Options

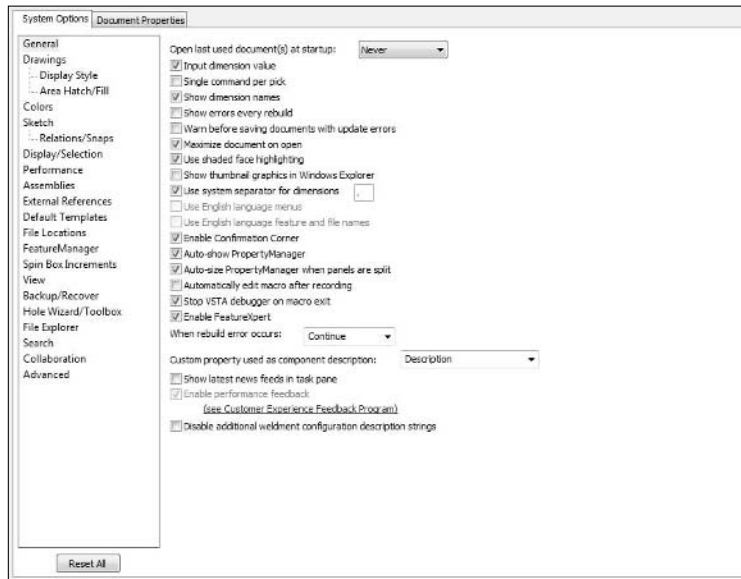
Document Properties

General Settings

The General System Options are for items that affect all of the software. One of the most used settings on this page is showing the dimension names, but several others are important. Figure C.1 shows the list of options on this page.

FIGURE C.1

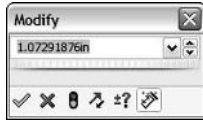
The System Options ⇨ General page



- **Open last used document(s) at startup.** The options are Never or Always. If you select Always, then when you start SolidWorks, it automatically opens the last document that you had open. Default is Never.
- **Input dimension value.** When you place driving dimensions in model sketches, the Modify dialog box appears so that you can change the dimension value immediately, as shown in Figure C.2. The one place this does not work is when you set sketch entities to be automatically dimensioned, which is controlled by a setting at Tools ⇨ Options ⇨ Sketch ⇨ Enable On Screen Numeric Input On Entity Creation.
- **Single command per pick.** This option is often overlooked. It allows only one application of a tool for each selection, after which the tool is automatically turned off. For example, if you turn on the Single Command Per Pick option and you click the Line tool, then after you draw one line, the Line tool turns itself off. If you double-click the Line icon from the toolbar, then you can draw multiple lines as usual. For selections from the Right Mouse Button menu and drop-down menus, the setting is always followed and cannot be overridden. Default is off.

FIGURE C.2

The Modify dialog box



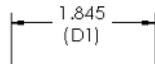
Tip

This double-clicking function also works in other Windows applications such as Word and PowerPoint, in various functions such as drawing figures, and in text boxes. ■

- **Show dimension names.** This option displays the dimension name in parentheses below the actual dimension. It is most useful when you use Design Tables, Link Values, or Equations (see Figure C.3). Default is off.

FIGURE C.3

Using the Show dimension names option



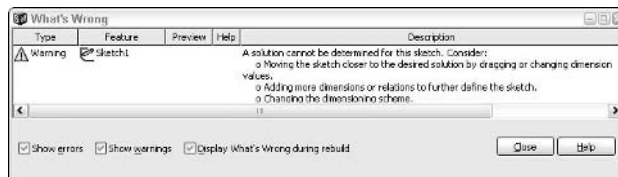
Note

The Show dimension names setting in SolidWorks 2010 has been removed from the General Settings to the View menu. ■

- **Show errors every rebuild.** Every time the model rebuilds, if any errors exist, then SolidWorks displays error messages, as shown in Figure C.4. This option can also be controlled in the What's Wrong dialog box that appears when errors exist in the active document. Default is off.

FIGURE C.4

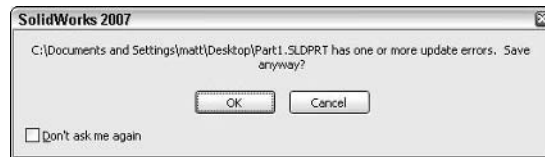
A warning message for a sketch error



- **Warn before saving documents with update errors.** The message given by this option has changed in SolidWorks 2010. In 2009, the message warned that there was an error and asked if you wanted to fix it. In 2010, it asks if you want to rebuild the model, as shown in Figure C.5. The new functionality does not reflect the purpose of the setting, so it is possible that this will be classified as a bug and fixed at some point after the release of SolidWorks 2010 SP0. Default is on. Clicking the Don't ask me again option turns this option off.

FIGURE C.5

The Update Error warning



Tip

The Don't ask me again toggle is a nice option if you find specific messages annoying. Messages that are permanently dismissed are listed and may be retrieved at **Tools** ⇨ **Options** ⇨ **Advanced**. ■

- **Maximize document on open.** This option causes all of the new windows in SolidWorks to be maximized within the SolidWorks window, even if the actual SolidWorks window is not maximized. Default is on.
- **Use shaded face highlighting.** When the model is shaded, any selected face turns entirely to the selection color (blue by default). If this option is turned off, then only the edges of the face are highlighted. Default is on.
- **Show thumbnail graphics in Windows Explorer.** This setting turns the small SolidWorks icons in front of the filename shown in the Windows Explorer detail view into a very small preview of the part (see Figure C.6).

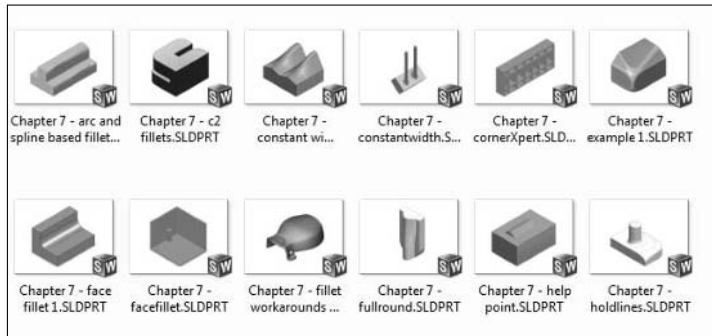
If you are using the Vista or Windows 7 Operating Systems, use the Views functionality in Windows Explorer, Open or Save dialogs. Default is on.

- **Use system separator for dimensions.** This setting establishes the character used to separate decimal values from whole number values. In Europe, a comma is most frequently used, while in the United States a period is used. This setting depends on the Windows setting that is found in the Windows Control Panel. Default is on.

When the setting is on, SolidWorks uses the default Windows setting, and the box to the right where you enter the separator character is disabled. When the setting is off, the box to the right is enabled.

FIGURE C.6

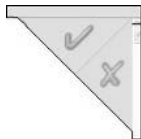
Thumbnail graphics in Windows Explorer



- **Use English language menus.** This setting is active only when SolidWorks has been installed in a language other than English. English is always available, regardless of what language was installed.
- **Use English language feature and file names.** Again, this setting is active only when a language other than English has been installed. This setting does not change the displayed names of existing documents and features; it only controls the default names assigned to new documents and features.
- **Enable Confirmation Corner.** All SolidWorks functions that use the Confirmation Corner (triangular area of the upper-right corner of the graphics window containing the green check mark and the red X; see Figure C.7) should also have a green check mark icon in the Property Manager or on the Right Mouse Button menu. Turning this option off could save some interface space, although it is not recommended. Default is on.

FIGURE C.7

The Confirmation Corner



- **Auto-show PropertyManager.** This option automatically shows the PropertyManager for items such as sketch entities and dimensions. If this setting is off, then you need to manually select the PropertyManager tab to access settings for these items. Default is on.
- **Auto-size PropertyManager when panels are split.** If you split the FeatureManager panel, and the PropertyManager is shown in one panel, that panel will autosize to accommodate the PropertyManager.

Note

You can detach the PropertyManager from the FeatureManager area and float it in the graphics window or dock it to a number of places. It can even be put on a second monitor. ■

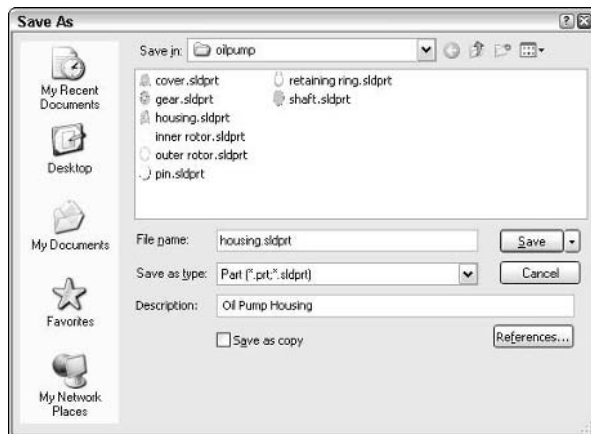
- **Automatically edit macro after recording.** This setting saves you from manually opening macros so that you can edit them immediately after recording them. Default is off.
- **Stop VSTA debugger on macro exit.** VSTA stands for Visual Studio Tools for Applications.
- **Enable FeatureXpert.** This option enables the Mate, Fillet, and Draft Xperts. The Xperts are intended for two purposes, either for novice users or for situations where a large number of entities are involved and you want the system to sort out errors automatically. Default is on.
- **When rebuild error occurs.** The available options are Continue, Stop, or Prompt. Continue ignores the error and rebuilds what it can. Stop causes the rebuild to stop at the first error and allows you to correct the error before moving down the tree. Prompt asks you what to do if there is an error. Default is Prompt.
- **Custom property used as component description.** The available options are any custom property. This setting affects which custom property is used in the Save As and Open dialog boxes. You can use the drop-down menu to replace the word Description and the value that follows it with any custom property name and value, as shown in Figure C.8.

Note

The custom property used as component description does not apply to the Description used in the BOM (Bill of Materials). ■

FIGURE C.8

The Description custom property is shown in the Save As dialog box.



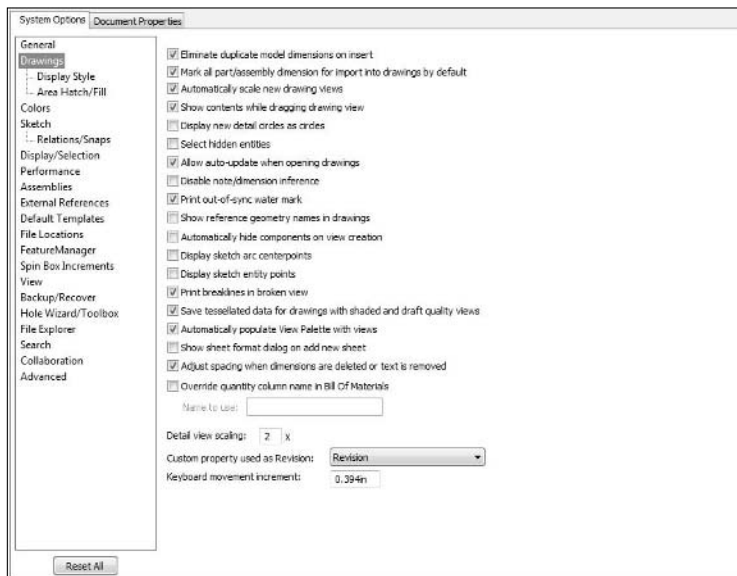
- **Show latest news feeds in task pane.** When SolidWorks starts up, you can see really simple syndication (RSS) feeds (blog entries) in the Task Pane. You can get back to these on the SolidWorks Resources tab of the Task Pane. Default is on.
- **Enable performance feedback.** Performance feedback is data about how you use SolidWorks, how often it crashes, and what was happening when it crashed. This feedback is in the form of an e-mail, and works best with Outlook. Default is set during installation.
- **Disable additional weldment configuration description strings.** When activated, this setting does not add the As Welded and As Machined configuration descriptions to weldment parts. Default is off.

Drawings

The Drawings page has many general settings that affect the display and default settings used on your computer. If you want to affect specific geometry or data types, you may want to look on the Document Properties page. Figure C.9 shows the Drawings page of settings.

FIGURE C.9

The System Options ⇄ Drawings page



- **Eliminate duplicate model dimensions on insert.** Prevents duplicate dimensions when Insert, Model Items is used. This setting can be overridden by the Eliminate Duplicates toggle in the Model Items PropertyManager. Default is on.

- **Mark all part/assembly dimensions for import into drawings by default.** Model dimensions use a setting called “Mark for drawing” to determine if a dimension should be inserted when using Insert Model Items. This option marks all new dimensions with the Mark for drawing option by default. Default is on.
- **Automatically scale new drawing views.** This option automatically scales new drawing views, based on the size of the view geometry relative to the size of the sheet. Default is 2X.
- **Show contents while dragging drawing view.** When moving drawing views, it is often useful to see the view geometry. The downside of this option is obviously system performance, particularly on large assembly views. The alternative is to only show the rectangular view border when dragging. Default is on.
- **Display new detail circles as circles.** No, you are probably not the only one to whom this sounds like double-speak. A detail circle refers to any closed loop that is used to create a detail view. This option displays the loop as a circle, regardless of whether it is drawn as a hexagon, rectangle, ellipse, spline, or any other closed profile. Turning it off enables it to be shown as it was drawn. The behavior of this setting is affected by the Document Property drafting standard in use. Default is off.
- **Select hidden entities.** This option enables you to select edges that have been manually or automatically hidden by moving the cursor over them and using the Right Mouse Button menu. Default is off.
- **Allow auto-update when opening drawings.** This option rebuilds drawings and associated models when you open the drawing. Default is on.
- **Disable note/dimension inference.** When placing or moving notes and dimensions, inference lines help you align to other notes or dimensions. Default is off.
- **Print out-of-sync water mark.** Detached drawings that have not been updated since the model was last changed can be printed with a watermark to make sure that users know that what was printed was not the latest available version. Default is on.
- **Show reference geometry names in drawings.** Reference geometry, such as planes and axes, may be assigned names that are important, or may just have generic default names such as Plane1. This option enables you to control whether or not these entities are shown on drawings. Default is off.
- **Automatically hide components on view creation.** If an assembly component is not visible when a view is created, then it will be set to hidden. This helps reduce drawing rebuild times. Default is off.

Caution

The Automatically hide components on view creation option can leave parts hidden when they should be shown; for example, when you are using exploded and section views. ■

- **Display sketch arc centerpoints.** This is the same as the setting under the Sketch heading. Sketched arc centerpoints (not edge centerpoints) can be shown or hidden using this setting. This setting only applies to drawings. Default is off.

- **Display sketch entity points.** This refers to endpoints, ellipse, parabola, and spline control points. When these points are off, it can be difficult to tell where the ends of sketch entities are in a chain of entities. Troubleshooting sketches can also become more difficult when the points are turned off. However, some people find the display of the points to be distracting. This setting only applies to drawings. Default is off.
- **Save tessellated data for drawings with shaded and draft quality views.** You can use tessellated data in View-only mode and in eDrawings. If this data is not saved, then the viewer and eDrawings will show blank views where shaded or draft quality views are called for. The setting has no effect on high-quality drawing views. Tessellated data increases drawing file size significantly. Default is on.

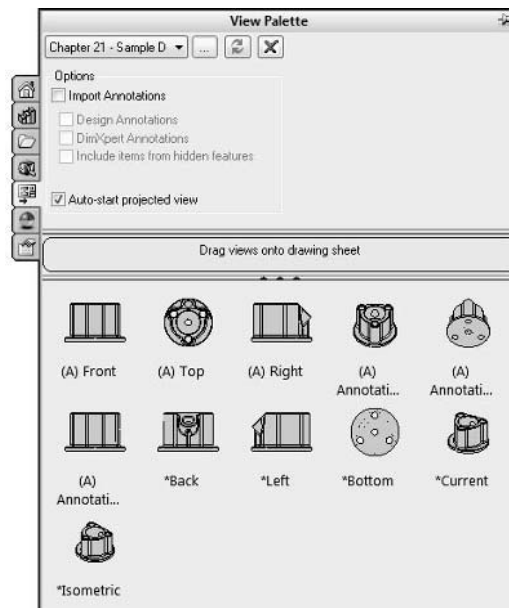
Best Practice

Best practice for the Save tessellated data setting is to turn it on. There is a constant trade-off between file size and performance (speed). Reading large files is slow compared to reading small files, but reading data is faster than recomputing the data. ■

- **Automatically populate View Palette with views.** The View Palette is located in the Task Pane, and you can use it to quickly drag-and-drop drawing views onto the drawing (see Figure C.10). Default is on.

FIGURE C.10

The View Palette



- **Show sheet format dialog on add new sheet.** When starting a new drawing from a template without a format, SolidWorks prompts you to select a sheet format. Default is off.
- **Adjust spacing when dimensions are deleted or text is removed.** When dimensions are stacked and spaced, if a dimension is altered or removed, the spacing will update. Default is on.
- **Override quantity column name on Bill Of Materials.** The default QTY column in the BOM can be difficult to manage, so SolidWorks offers the option to create your own column for quantity. Default is off.
- **Detail view scaling.** This option controls the default scale at which detail views will be created. This is most commonly set as shown. Default is 2.
- **Custom property used as Revision.** This option controls which custom property is used to drive the Revision Table. If it is blank, a property called Revision is used. Default is blank.
- **Keyboard movement increment.** You can nudge drawing views and annotations by pressing an arrow key on the keyboard. This setting controls the distance. Default is 10mm.

Tip

SolidWorks tends to display many default values as “.394 in” or some multiple variant. This is because SolidWorks thinks in metric units, and .394 inches corresponds to 10 millimeters. ■

Display Style

The Display Style page sets the defaults for display style, tangent edge display, and display quality, as shown in Figure C.11. These values apply only to newly created views, not views that already exist.

- **Display style for new views.** This setting applies to newly created views. It does not affect existing views. Default is Hidden lines removed.
- **Tangent edges in new views.** Edges between tangent faces display using the selected style. The Removed option does not typically work well with parts without sharp edges, although it would be a good way to display an image of only the silhouette outline of a part. Default is Visible.
- **Display quality for new views.** Draft quality views are created when the model is lightweight, and are typically used for large or complex assemblies to improve performance. Default is High quality.

Area Hatch/Fill

This page enables you to set the defaults for hatching and area fills, as shown in Figure C.12.

- **Area Hatch/Fill.** This setting determines the default hatch, scale, and angle for newly created hatch fills. Default is ANSI 31, scale:1, angle:0deg.

FIGURE C.11

The System Options ⇨ Drawings ⇨ Display Style page

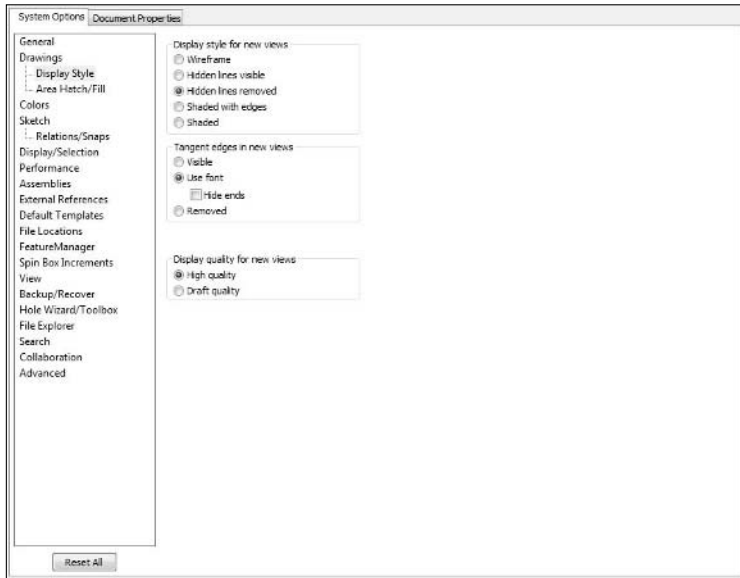
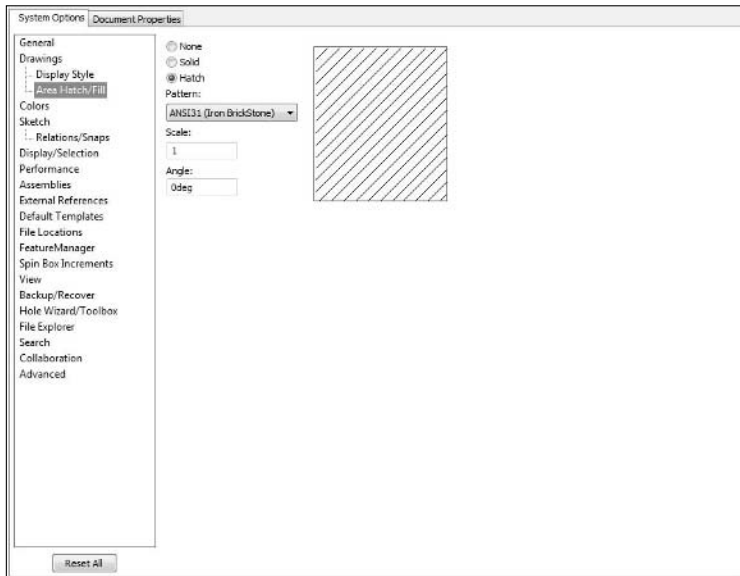


FIGURE C.12

The System Options ⇨ Drawings ⇨ Area Hatch/Fill page

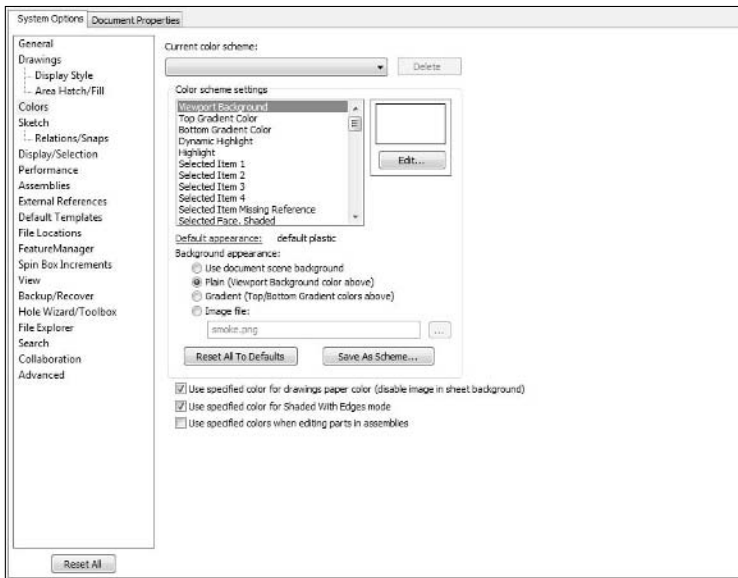


Colors

You can establish color schemes for various items on the interface by using the settings shown in Figure C.13.

FIGURE C.13

The System Options ⇄ Colors page



- **Current color scheme.** By default, SolidWorks installs with several color schemes, and you can make your own custom schemes and save those as well. You can use this drop-down menu to select a color scheme.
- **Color scheme settings.** Figure C.14 shows the list of entity colors that you can change. Some colors stand for multiple entities that may not be listed.

Caution

If you change the background to the same color as another object, you may not be able to see the object against the background. SolidWorks does not automatically change text color if it is set to the same color as the background. ■

- **Background appearance.** You can select a document scene, from a plain color, a gradient color, or an image file background. Background images must be in one of the following formats: BMP, GIF, JPG, JPEG, TIF, WMF, and PNG. Not all compression formats are allowed. Be aware that **Use document scene background** is the default and creates a document specific background that overrides system settings.

FIGURE C.14

List of entity colors that can be changed

| | | |
|--|---|---|
| Viewport Background Top Gradient Color Bottom Gradient Color Dynamic Highlight Highlight Selected Item 1 Selected Item 2 Selected Item 3 Selected Item 4 Selected Item Missing Reference Selected Face, Shaded | Drawings, Paper Color Drawings, Background Drawings, Visible Model Edges Drawings, Hidden Model Edges Drawings, Model Edges (SpeedPak) Drawings, Model Tangent Edges Dimensions, Imported (Driving) Dimensions, Non Imported (Driven) Dimensions, Dangling Dimensions, Not Marked for Drawing Dimensions, Controlled by Design Table | Text Sketch, Over Defined Sketch, Fully Defined Sketch, Under Defined Sketch, Invalid Geometry Sketch, Not Solved Sketch, Inactive Grid Lines, Minor Grid Lines, Major Construction Geometry Assembly, Edit Part |
| Assembly, Hidden Lines of Edit Part Assembly, Non-Edit Parts Inactive Entities Inactive Handles Temporary Graphics Temporary Graphics, Shaded Surfaces, Open Edges Edges in Shaded With Edges Mode Feature Tree, Normal Item Feature Tree, Selected Item Annotations, Imported | Annotations, Non Imported Assembly Interference Volume Hidden Edge Selection Show Color Mate Callout, Healthy Mate Callout, Warning Mate Callout, Error Annotations, DimXpert DimXpert, Under Constrained DimXpert, Fully Constrained DimXpert, Over Constrained Annotations, TolAnalyst Dimension | |

Tip

A free image application, such as IrfanView (www.irfanview.com), can be very helpful in changing formats, cropping images, or editing colors or resolutions. If you need more Photoshop-like power, try the free application Gimp at www.gimp.org. ■

- **Reset All To Defaults.** If you create a visual mess that you cannot repair, you can easily return the settings back to their defaults.
- **Save As Scheme.** If you have multiple users on one Windows login or you just like a change of pace from day to day, you can save your schemes. I use a scheme with a white background for capturing screen images and a gradient background for working.
- **Use specified color for drawings paper color (disable image in sheet background).** The color of the sheet of paper is controlled separately from the color of the background behind the paper. Paper color can be overridden by an image. To remove the image, turn this setting on. Default is off.
- **Use specified color for Shaded With Edges mode.** The Shaded With Edges display mode can use either a single color for edges on all models (black is the default) or a contrasting shade (the same hue, but either lighter or darker). Default is on

Tip

When displaying an assembly in Wireframe mode, it is most useful for each component to be displayed in its own color, rather than having all of the parts use the same color. The wireframe color can be related to the shaded color for a part through a document-specific setting, Document Properties ⇨ Colors ⇨ Apply Same Color to Wireframe, HLR, and Shaded. This setting is only available for part documents. ■

- **Use Specified colors when editing parts in assemblies.** When editing parts in the context of an assembly, you can specify that the colors of parts being edited and parts not being edited to the colors specified in the object color list for Assembly, Edit Part and Assembly, Non-Edit Part. Default is off.

Part V: Appendixes

Caution

Administrators and users alike should be careful about changes to system colors. If you set up a dark background, you may also set up light-colored dimensions and annotations, which will be difficult for a person with a light background to read. On the other hand, changing a curve display color from light yellow to medium orange for visibility is subtle and helpful. As Administrator and creator of standards for SolidWorks, you should establish standardized colors to ensure everyone who receives documents that your team produces can read and work with the documents easily. The default colors are a reasonable starting point. ■

Tip

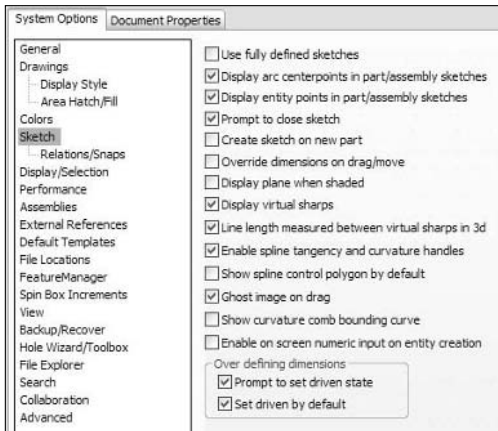
You can change the transparency of the parts by selecting **System Options** ⇨ **Display/Selection** ⇨ **Assembly transparency** for in-context edit. ■

Sketch

You can control the default sketch behavior and display by using the settings shown in Figure C.15.

FIGURE C.15

The System Options ⇨ Sketch page



- **Use fully defined sketches.** If this setting is turned on, then you cannot exit a sketch or create a feature from a sketch unless the sketch is fully defined. Default is off.
- **Display arc centerpoints in part/assembly sketches.** Arc centerpoints can be useful to select for relations or dimensions, but they can also clutter your screen if there are too many of them. Default is on.
- **Display entity points in part/assembly sketches.** *Entity points* means endpoints for all sketch entities, as well as control points for splines, ellipses, and parabolas. Endpoints are

important to display because without them, you cannot know whether or not the length of the sketch entity has been defined. Figure C.16 shows a line segment with and without this setting. While the display looks cleaner without the big endpoints, the information conveyed by the color of the entity point is important. Default is on.

FIGURE C.16

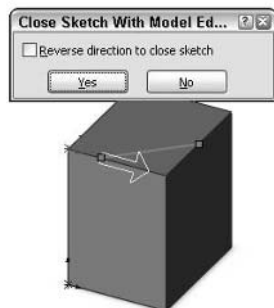
Showing sketch entity points



- **Prompt to close sketch.** If an open sketch profile can be closed using model edges, SolidWorks prompts you to allow it to automatically convert edges to sketch entities as shown in Figure C.17. The software closes the sketch in the direction of the arrow and allows you to reverse the direction in which to close the sketch. Default is off.

FIGURE C.17

A prompt appears, enabling you to automatically close the sketch.



Part V: Appendixes

- **Create sketch on new part.** If you create a lot of new parts, this may be a useful option. Default is off.

Caution

The Create Sketch on New Part option has been known to cause problems with third-party add-ins that create new parts automatically. The new sketch is placed on the Front (XY) plane. ■

- **Override dimensions on drag/move.** This is a great option for concept work, and works well with Instant3D. It allows you to drag fully defined sketches as if they were under-defined. It is also available through the menus at Tools ⇨ Sketch Settings ⇨ Override dims on Drag/Move. Default is off.

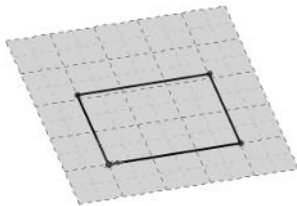
Tip

You can use the Override Dimensions on Drag/Move setting in conjunction with the Instant3D tool. This enables dragging sketches and blind extrusion depths without editing sketches or features. ■

- **Display plane when shaded.** When you are editing a sketch and the model is displayed in a shaded mode, the sketch plane is translucent. This is often an aid that is used when training new users to help them visualize when they are in Sketch mode and which sketch plane they are using. The setting is often used with grid display, as shown in Figure C.18. Default is off.

FIGURE C.18

A shaded sketch plane with grid display

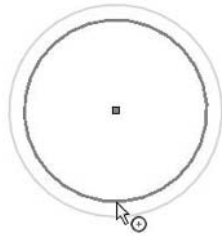


- **Display virtual sharps.** This setting is meant to control the display of virtual sharps in the sketch, but in fact it has done nothing since the 2007 version, as late as beta 3 of 2010. Default is on.
- **Line length measured between virtual sharps in 3D.** When you apply a dimension to a line in a 3D sketch with fillets on the ends of the line, the dimension goes to the virtual sharps rather than the endpoints of the line. Default is on.
- **Enable spline tangency and curvature handles.** This setting makes the handles visible. Default is on.

- **Show spline control polygon by default.** The spline control polygon is the set of gray straight lines and dots that surround a spline. This control polygon is helpful in getting a smoothly shaped spline. Default is off.
- **Ghost image on drag.** This setting enables a ghosted image of where your sketch was originally placed when dragging. The ghost disappears when the sketch entity is dropped. Figure C.19 shows a sketch ghost. This is useful on all kinds of sketches, particularly complex sketches and splines. Default is on.

FIGURE C.19

A sketch ghost



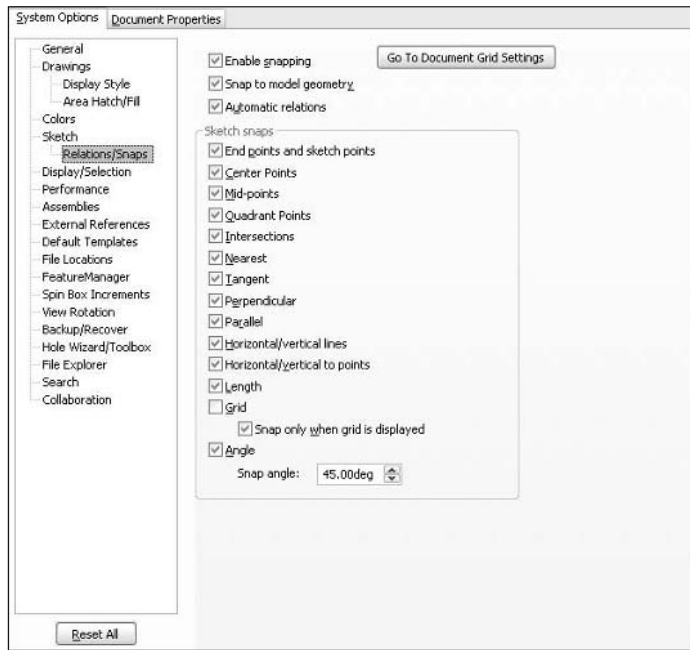
- **Show curvature comb bounding curve.** This option creates a curve that goes across the top spines of the curvature comb. The bounding curve was at one time the default, but there were problems found with its display in areas where curvature changes drastically that could not be fixed easily, so SolidWorks removed it. Again, users cried out for it to be replaced regardless of the imperfection, so now we have it as an option. Default is off.
- **Enable on screen numeric input on entity creation.** This option only works when you are using click-click sketching, and does not work for click-drag. This is the option that basically disables Input Dimension Value when used in conjunction with the Add Dimensions option in the sketch entity PropertyManager and click-drag sketching. This is confusing and unnecessarily hamstrung by the programmers.
- **Prompt to set driven state.** When adding a dimension that over-defines a sketch, SolidWorks prompts you to set the dimension to a driven or reference dimension state (gray). The alternative is to leave it over-defining (red or pink). Default is on.
- **Set driven by default.** This setting automatically makes a dimension driven if it over-defines the sketch. Default is on.

Relations/Snaps

The settings shown in the Relations/Snaps page control sketch snapping behavior. Most of the Sketch Snaps can also be accessed using the Quick Snaps toolbar. Other snapping and automatic relation settings can also be accessed in the menus at Tools ⇨ Sketch Settings or Tools ⇨ Relations/Snap (see Figure C.20). The settings include:

FIGURE C.20

The System Options ⇨ Relations/Snaps page



- **Enable snapping.** This option toggles all automatic relations, snapping, and inferencing. Items that are snapped to are controlled by the items selected in the Sketch snaps area. Default is on.
- **Snap to model geometry.** If this setting is off, then sketching does not interact with model geometry, such as edges and vertices, but it does interact with other sketch entities. Default is on.
- **Automatic relations.** If this setting is turned off, then no automatic relations are created, although the inference symbols and relations still appear. Default is on.

Tip

It is easy to tell which cursor symbols will result in an automatic sketch relation being applied. If the background of the symbol turns yellow, then the relation will be applied. If the background remains white, then no relation will be applied. For example, in Figure C.21, the horizontal relation will be applied, but the vertical relation is only implied. ■

- **Sketch snaps.** By default, all these options are selected except for Grid. You can easily change the snap behavior through the Quick Snaps icon on the Sketch toolbar, as shown in Figure C.22. A sketch tool must be active for this icon to be activated.

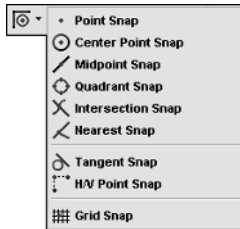
FIGURE C.21

The horizontal relation is applied, but the vertical relation is only implied.



FIGURE C.22

Quick Snap filters

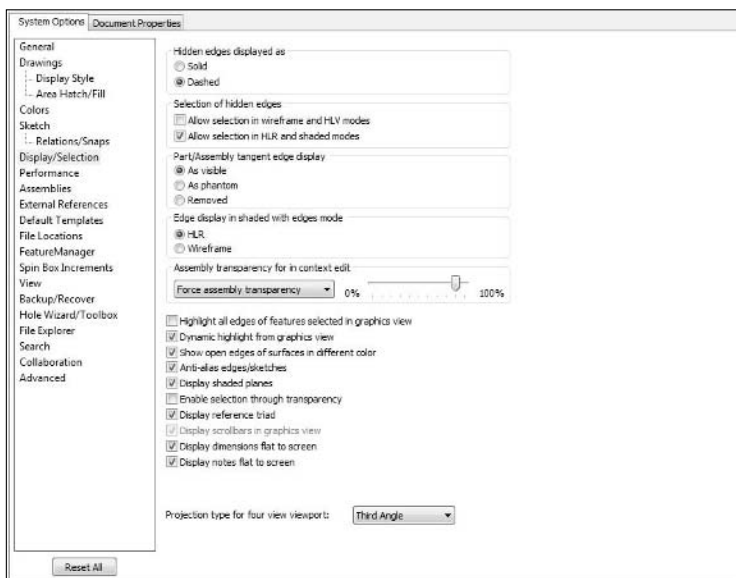


Display/Selection

The Display/Selection page enables you to set up your edge and in-context transparency settings, as shown in Figure C.23. Settings include:

FIGURE C.23

The System Options ⇄ Display/Selection page



- **Hidden edges displayed as.** This setting refers to edges that are not visible through the part. Default is Dashed.
- **Allow selection in wireframe and HLV modes.** This option enables you to select hidden edges in display modes where they can be seen. HLV is short for Hidden Lines Visible. Default is on.
- **Allow selection in HLR and shaded modes.** This option enables you to select hidden edges in display modes where they *cannot* be seen. On simple models where you want to be able to select edges without rotating the model or changing to Wireframe mode, this option can be useful. However, on complex models and assemblies, you may want to turn this option off. Default is off.
- **Part/Assembly tangent edge display.** Edges created by fillets and other tangent features follow this display setting. There is a similar setting on the Drawings ⇨ Display Style page for new drawing views. Default is As Visible.

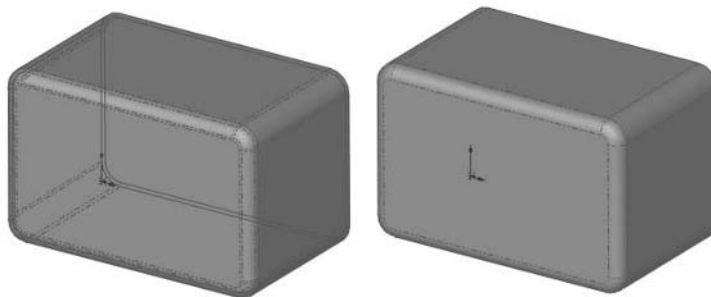
Tip

This is a simple evaluation technique that is often employed to see which faces on a model are really tangent. Deviation Analysis is a more detailed method, but simply showing tangent edges in a font shows most of what you need to know at a glance. Edges that are not tangent can also be analyzed separately in more detail. ■

- **Edge display in shaded with edges mode.** This setting hides edges that lie behind the solid or displays all edges in Shaded With Edges display mode, as shown in Figure C.24. HLR stands for Hidden Lines Removed. Default is HLR.

FIGURE C.24

Edge display in Shaded With Edges mode, showing the Wireframe option (left) and the HLR option (right)



- **Assembly transparency for in context edit.** This setting forces either assembly transparency or opacity, or leaves transparency as-is when editing parts or assemblies in-context. In-context colors are controlled on the Colors page. Transparency often has a negative impact on graphics performance. Default is Force Assembly Transparency, 90%.

- **Highlight all edges of features selected in graphics view.** If this option is turned on, then all of the edges of a feature will be highlighted. Selected faces are highlighted according to the Use Shaded Face Highlighting setting on the General page. Default is off.
- **Dynamic highlight from graphics view.** When the cursor is over an entity such as a face, edge, or sketch element, the entire entity is highlighted and the cursor changes to show what type of entity would be selected if you clicked at that instant. Default is on.
- **Show open edges of surfaces in different color.** Open surface edges are edges that are not shared by two faces. The default color for this setting is medium blue by default and is driven by the Surfaces ⇨ Open Edges object type from the Colors page. Default is on.
- **Anti-alias edges/sketches.** Model edges, and most noticeably shaded model edges, can be anti-aliased to display more smoothly. Anti-aliasing means that angled lines, which typically display as jagged stair-steps, appear smoother. Sometimes this results in a slight blurring or even thickening of the edges. Default is on.
- **Display shaded planes.** When planes are displayed, they appear translucent — typically green from one side and red from the other. Colors are controlled in Document Properties ⇨ Plane Display. This setting may slow down some graphics cards. Default is on.
- **Enable selection through transparency.** Transparent parts in an assembly are invisible to the cursor if there is a shaded part under the cursor. If there is nothing under the cursor except for the transparent part, then it can be selected. You can override this setting by holding the Shift key while selecting the part. Default is on.
- **Display reference triad.** This refers to the reference triad in the lower-left area of the screen. Default is on. See Figure C.25. The Triad can be used to reorient the view by clicking axes.

FIGURE C.25

The reference triad



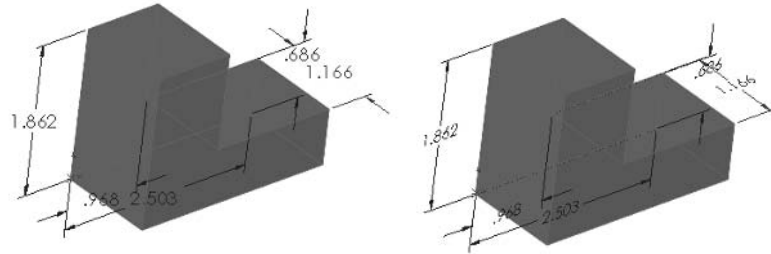
Note

You can reorient the view by clicking the axes of the triad. One click makes the axis you click point directly out of the screen. Alt+click rotates the view around that axis. Shift+click makes the view rotate 90 degrees around the selected axis. ■

- **Display scrollbars in graphics view.** If this option is grayed out, close all documents and try again. Default is off.
- **Display dimensions flat to screen** and **Display notes flat to screen.** The alternative is to show the dimensions or notes parallel to the sketch plane in which they were applied. Default is off (see Figure C.26).

FIGURE C.26

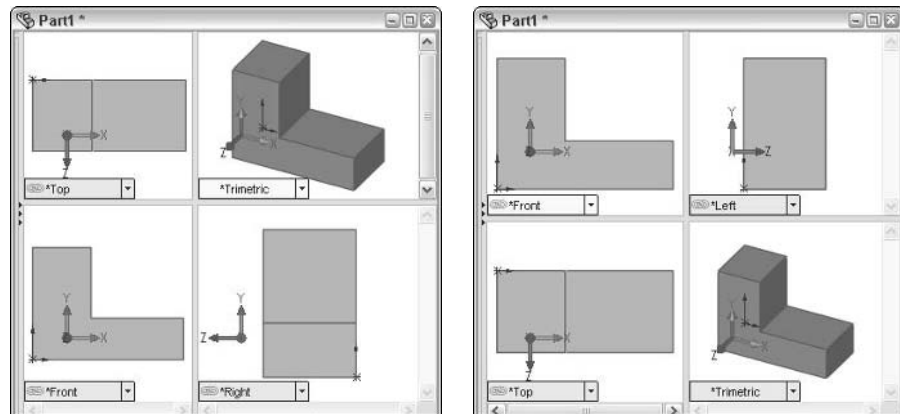
Dimensions flat to screen and flat to the sketch plane



- **Projection type for four view viewport.** The options are Third Angle or First Angle. This setting does not refer to drawings, only to viewports in the model or assembly window. In some cases SolidWorks will install with a standard you don't expect. Be sure to check this setting after installation, and make sure you understand the difference between First and Third Angle projections (see Figure C.27).

FIGURE C.27

Third and First Angle projection viewports



Caution

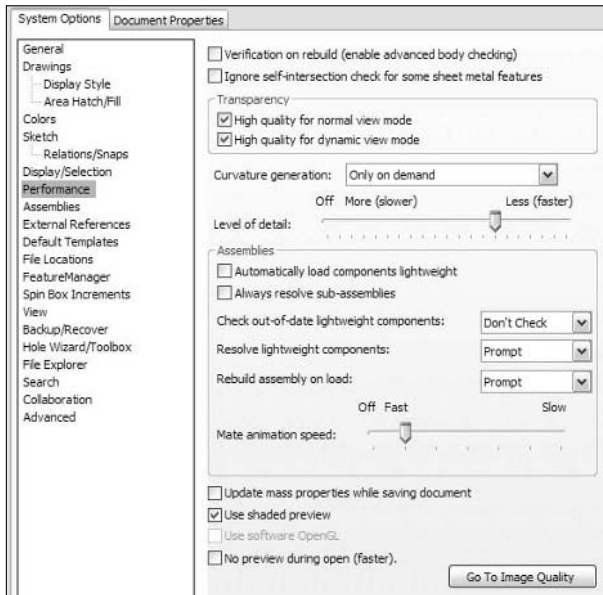
This setting is not linked to dimensioning standards or the country in which the software is installed. Third Angle projection is used in the United States and is part of the ANSI standard, while the First Angle projection is used in Europe and is from the ISO standard. Sending a Third Angle drawing to a manufacturer where the standard is First Angle may at least cause some confusion and could potentially cause incorrect parts to be manufactured. Suggested Best Practice: Include a variety of 3D views to reduce confusion or otherwise provide documentation with clear intent. ■

Performance

All of these settings, shown in Figure C.28, affect an item's rebuild time, file size, or load time.

FIGURE C.28

The System Options ⇨ Performance page



- Verification on rebuild (enable advanced body checking).** When SolidWorks rebuilds a model, it checks the geometry for errors. The errors it is checking for are faces within a given solid or surface body intersecting other faces of the same body anywhere except at an edge. To save on checking time (when the Verification on rebuild option is off), each face of a body is checked only against adjacent faces within that body (faces with which it shares an edge). This is adequate for about 90 percent of models. When the Verification on rebuild option is turned on, each face is checked against every other face in the body, which takes more time. The advantage of using this option is that you do not unknowingly create bad geometry. The rebuild time penalty varies, depending on the model, but adds between 5 percent and 60 percent to the rebuild time. The number of faces obviously affects the verification time (the number of faces is often greatly affected by the existence of small fillets), as does the number of bodies. If you are given two models with an equal number of faces, but one model is multi-body and the other is a single body, Verification on rebuild creates less of a rebuild time penalty on the multi-body part, because faces do not have to be checked between bodies. Aside from longer rebuild times, a result of having this option turned on will be failed features when SolidWorks finds face intersection errors. Default is off.

Best Practice

Best practice for the Verification on rebuild option is to turn it on until it becomes too much of a burden in terms of long rebuild times. You should then only turn it on to check the model after critical points and when the model is complete. To make full use of this tool, turn it on, press Ctrl+Q, and run a Tools⇨ Check analysis. ■

- **Ignore self-intersection check for some sheet metal features.** Sheet metal parts sometimes have an edge-on-edge condition that causes a self-intersection warning message to appear. This option suppresses the warning message when appropriate. Default is off.
- **High quality for normal view mode.** When a transparent model is not being rotated, zoomed, or panned, it appears in high-quality transparency. For reference, low-quality transparency causes a coarse “screen door” effect. This option is generally only turned off for low-end graphics cards and large assemblies. Default is on.
- **High quality for dynamic view mode.** When a transparent model is being rotated, zoomed, or panned, it is displayed in high-quality transparency. Default is on.

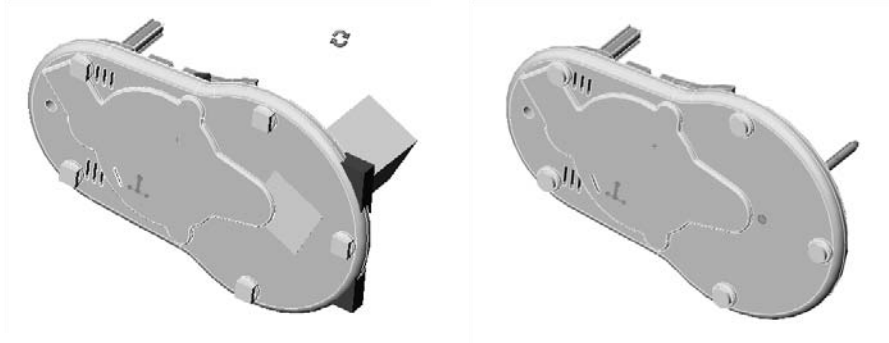
Tip

Some graphics cards may exhibit slowdowns if shaded (transparent) planes are in use (System Options⇨ Display/Selection⇨ Display Shaded Planes) with the high-quality transparency settings. ■

- **Curvature generation.** The options are Only on demand and Always. This should always be left at its default setting, which is Only on demand. Curvature can be displayed on a per-model or per-face basis. Because curvature display can slow down system performance, you should use this sparingly, only when it is needed. The Always option generates the curvature display information behind the scenes and makes it available more quickly, but at the expense of system memory and CPU time.
- **Level of detail.** Depending on the complexity of the assembly, and the performance of your graphics card, the Less (faster) end of the slider causes small parts to be simplified into blocks when you rotate the view. Moving the slider toward More (slower) prevents the simplification of parts but may result in more choppy rotation. The label Off in the interface for this setting is conspicuously misplaced. The Help information for it is also misleading. Off refers to the decimation of detail being turned off, so that all of the detail is displayed (Help says that none of the detail is displayed). Default is about 75 percent of the way to the right. Figure C.29 shows a view with detail settings of More and Less.
- **Automatically load components lightweight.** Assembly components are loaded lightweight. This includes parts and subassemblies, unless the Always Resolve Sub-assemblies option is turned on.
- **Always resolve sub-assemblies.** When a top-level assembly is opened lightweight, subassemblies are resolved, although subassembly components are still lightweight. Default is off.
- **Check out-of-date lightweight components.** The options are Don't Check, Indicate, and Always Resolve. Out-of-date lightweight components are parts or assemblies that have changed since the last time they were resolved in the upper-level assembly. When they are in this state, they appear like the last version of the part or subassembly that was resolved. Obviously the Don't Check option is the most vulnerable to this kind of error, and Always Resolve is the most conservative approach.

FIGURE C.29

A high level of detail compared to a low level of detail



The Indicate option will put a special symbol on the components in the tree, indicating that the component is out of date. The symbol is a part icon with a lightweight feature, where the feather is blue with red stripes. Default is Don't Check.

- **Resolve lightweight components.** The options are Prompt and Always. Some operations require data that, in turn, requires that lightweight components be resolved. In these cases, this option determines whether the user is prompted to approve the resolve or whether components are just resolved automatically. Default is Prompt.
- **Rebuild assembly on load.** The options are Prompt, Always, and Never. Sometimes when an assembly is opened, rebuild symbols will appear on some of the parts or features because referenced components have changed while the assembly was not open. In these cases, the assembly needs to be rebuilt to correctly represent the design. The best option for this setting will be determined by assembly size or complexity. Default is Prompt.
- **Mate animation speed.** When parts are mated in an assembly, it is a visualization aid to have them move into place so that you can see how the parts fit together. Pushing the slider all the way to the left turns off the animation. Default is Fast.
- **Update mass properties while saving document.** Solid mass properties take some time to compute and can slow down rebuilding and saving times, particularly for larger assemblies with many parts. However, if you have BOMs or notes on drawings linked to part or assembly mass properties, then you may want to use this option. Default is off.
- **Use shaded preview.** Most features allow a preview of the geometry. Although this can reduce performance on a marginal system, in most cases it is a good way to visualize the finished feature, and it is also useful for troubleshooting. Loft and Sweep features allow a mesh preview in addition to the shaded preview. Default is on.
- **Use Software OpenGL.** OpenGL, where GL stands for Graphics Language, is the standard that SolidWorks uses to drive the display of shaded 3D geometry. If your graphics card is not OpenGL compatible, then SolidWorks emulates the hardware acceleration with software. Software emulation is much slower than hardware acceleration. You can only

Part V: Appendixes

change this option when there are no documents open. If your graphics card is not OpenGL compatible, then SolidWorks will turn on this option, and you will not be able to turn it off. The Use Software OpenGL option is often used for troubleshooting. If you are having trouble with SolidWorks crashing, then selecting the Use Software OpenGL option will either eliminate or indicate the graphics card or driver as the cause.

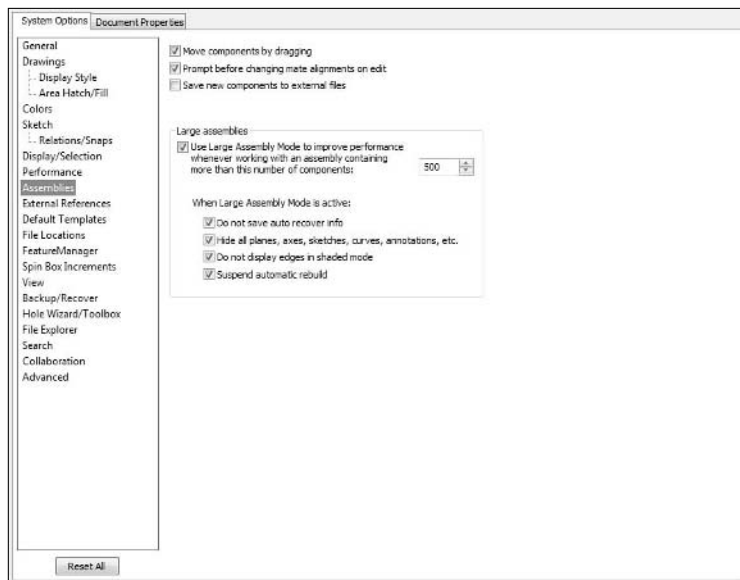
- **No preview during open (faster).** When opening documents, SolidWorks displays a preview that can be panned, rotated, and zoomed. Disallowing this preview speeds up opening the documents. Default is off.

Assemblies

The Assemblies page is mostly comprised of large-assembly settings, as shown in Figure C.30.

FIGURE C.30

The System Options ⇄ Assemblies page



- **Move components by dragging.** Without this option, you need to use the Move Component tool from the Assembly toolbar to move parts. The default setting (on) allows you to move under-defined components in an assembly by just using the Select cursor. Default is on.
- **Prompt before changing mate alignments on edit.** SolidWorks has been known to switch the mate alignment automatically, flipping parts in an assembly needlessly. Turning this option on makes sure the software prompts you to approve a mate flip before it does it.

- **Save new components to external files.** When you create parts within an assembly, and those parts are virtual parts, this setting forces you to save the virtual parts to external files when the assembly is saved.
- **Use Large Assembly Mode.** Large Assembly Mode, or LAM, automatically initiates when the number of components in the active assembly exceeds the number that you enter here. You can also initiate LAM manually by selecting Tools ⇄ Large Assembly Mode. LAM is saved with the document, and so the next time an assembly is opened, it is opened with the LAM settings. Default is on.
- **Do not save auto recover info.** This option overrides the Auto-recover setting on the Backup/Recover page. Default is on.
- **Hide all planes, axes, sketches, curves, annotations, etc.** This option is the most frequent cause of confusion with LAM because it turns off all of the items in the View menu. To get the items back, just deselect Hide All Types in the View menu. Default is on.
- **Do not display edges in shaded mode.** This setting disables Shaded With Edges mode, using simple Shaded mode instead. This is because displaying edges reduces graphics performance. Default is on.
- **Suspend automatic rebuild.** In assemblies with many mates, patterns, or in-context features, rebuilding the assembly can take a fair amount of time. Suspending automatic rebuilds can save that time, but it may cause other problems if you are not careful such as out of date references. You can manually rebuild the assembly or override the setting by using the Right Mouse Button menu at the top of the assembly tree. Default is off.

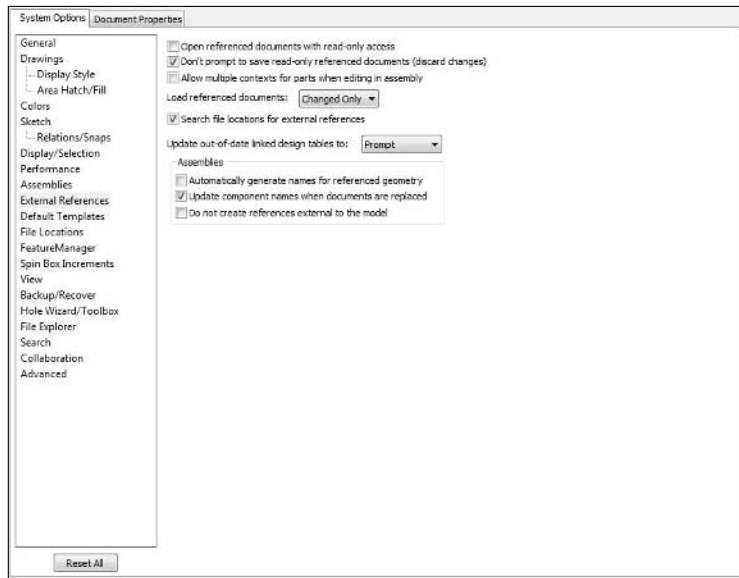
External References

The External References page, shown in Figure C.31, contains some settings that you should be familiar with, particularly if you deal with assemblies, in-context relations, or other more involved assembly reference functions.

- **Open referenced documents with read-only access.** If you open an assembly, then all of the parts and subassemblies are opened with read-only access. If you open a drawing, then whatever is referenced by the drawing is also opened read-only. This option is typically used when many users are sharing files from folders directly over a network, especially if one user is working mainly on drawings and another user is working on parts or assemblies. Using the Reload function allows you to change from read-only to read-write access, and vice versa. For more information, see the section of this appendix on Collaboration settings, as well as Appendix A. Default is off.
- **Don't prompt to save read-only referenced documents (discard changes).** If you open a large assembly as read-only, it is possible for changes to occur just in rebuilding the assembly, which, in turn, makes SolidWorks think that the parts have changed. In this case, when you exit the assembly, SolidWorks will prompt you to save the parts, even though they are read-only and cannot be saved. There is no way out of the loop, other than to just answer No to each prompt to save. This option can help you to avoid getting into that loop. Default is off.

FIGURE C.31

The System Options ⇨ External References page



- **Allow multiple contexts for parts when editing in assembly.** This option allows in-context references from one part to more than one assembly. Although it is usually considered very bad practice to use this option, there are times when you will need to. An example of an appropriate use for this option is if a plate has in-context references in both a subassembly and the top-level assembly. Remember that this is a System Option, and does not follow the document. In-context references are a major source of assembly performance problems. Default is off.

Best Practice

Best practice for in-context references calls for moderation and avoiding circular references. For more specific recommendations, please refer to the *SolidWorks 2009 Bible* (Wiley, 2009) or the most current edition. ■

- **Load referenced documents.** The options are Prompt, All, None, and Changed Only. In this case, referenced documents refer to base parts, mirror parts, and so on. For example, if you open a part called Right Shoe, and Right Shoe was made by mirroring Left Shoe, and this option is set to All, Left Shoe is automatically opened as well. Default is Changed Only.
- **Warn about saving referenced documents.** When saving an assembly, this option enables or disables the prompt to save referenced documents. If you turn this option off, then referenced documents are automatically saved without prompting the user. This eliminates a dialog box that is usually unnecessary. Default is on.

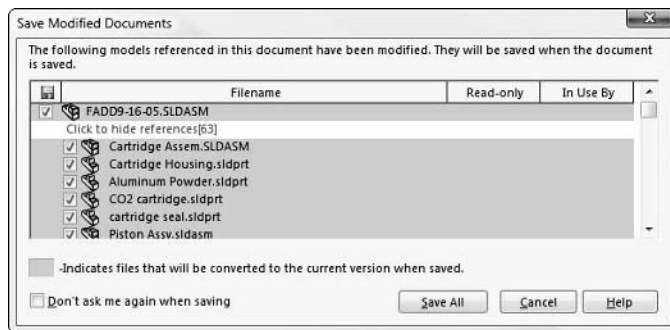
Note

This setting has been removed from SolidWorks 2010. The warning message is no longer issued because the Save Modified Documents dialog box has been added to let users know which assembly references are being saved. ■

Figure C.32 shows the Save Modified Documents dialog, which enables you to see and change the list of referenced components to be saved while saving the top level assembly.

FIGURE C.32

Using the Save Modified Documents dialog to view the list of referenced components to be saved with the top level assembly



- **Search file locations for external references.** The file locations referred to in this case are listed in System Options → File Locations → Referenced Documents. The Referenced Documents paths come first in the references search order. For more information on references search order, see Appendix A and the section of this appendix that discusses the System Options → File Locations page. Default is on.
- **Update out-of-date linked design tables to.** The options are Prompt, Model, and Excel File. This setting refers to linked design tables in two situations: design tables on drawings linked to model tables and external design tables linked to the model. Default is Prompt.
- **Automatically generate names for referenced geometry.** When you turn this option on, it creates names for geometry used in mates and thus requires write access to the parts being mated. The default setting is off, which is best unless you plan to replace parts using named entities. Naming is done through the Properties for each entity, such as face, edge, vertex, and so on. Default is off.
- **Update component names when documents are replaced.** This option should remain with the default setting (on) unless you use Component Properties to assign names to parts that are to be used in the assembly tree. Be aware that SolidWorks Routing disables this option. Default is on.

Part V: Appendixes

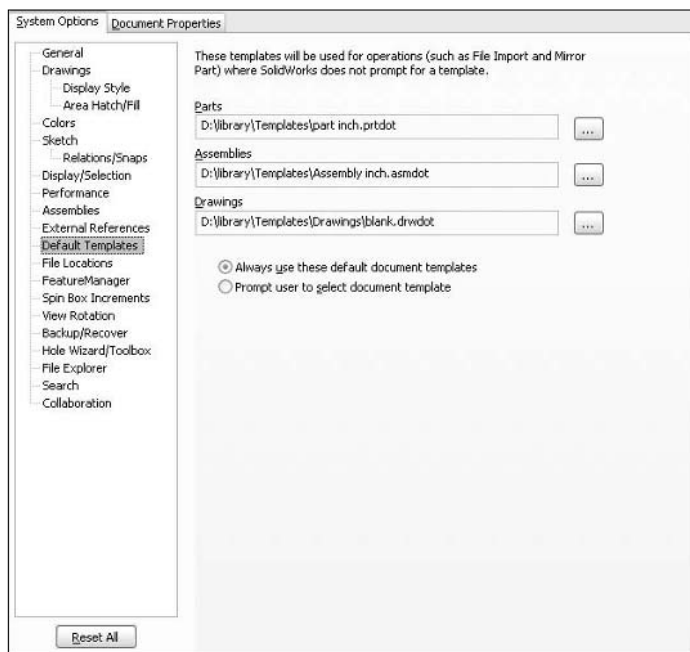
- **Do not create references external to the model.** This option applies to in-context references such as converted edges and feature end conditions. When you select this option, offset or converted sketch entities will be created without relations, and extrude “up to” features and offset surfaces will be created with broken references. This option is also available as a toolbar button on the Assembly toolbar, so it can be controlled on the fly. Default is off.

Default templates

The default templates settings, shown in Figure C.33, help to determine how templates are used for automatically created files.

FIGURE C.33

The System Options ⇄ Default Templates page



- **Default template paths.** You can specify the path to the template folder in the System Options, File Locations page.
- **Always use these default document templates** and **Prompt user to select document template.** These two options are actually a single toggle (one of them will always be on). When the system has to create a new document, this toggle determines how it will know which template to use. Situations where the system has to create a new document include

imported parts, split parts, and mirrored parts. The Always Use these Default Document Templates option means that the above listed templates will be used automatically. If they are not available, then automatically generated default templates will be used (these are the original templates that exist with a new installation of SolidWorks). The Prompt User to Select Document Template option means that in those situations, the user will be prompted to select a template for each part that is generated. In imported assemblies with a lot of parts, it may be wise to opt for the automatic setting unless you need to specify different templates for different parts. Default setting is Always Use These Default Document Templates.

Best Practice

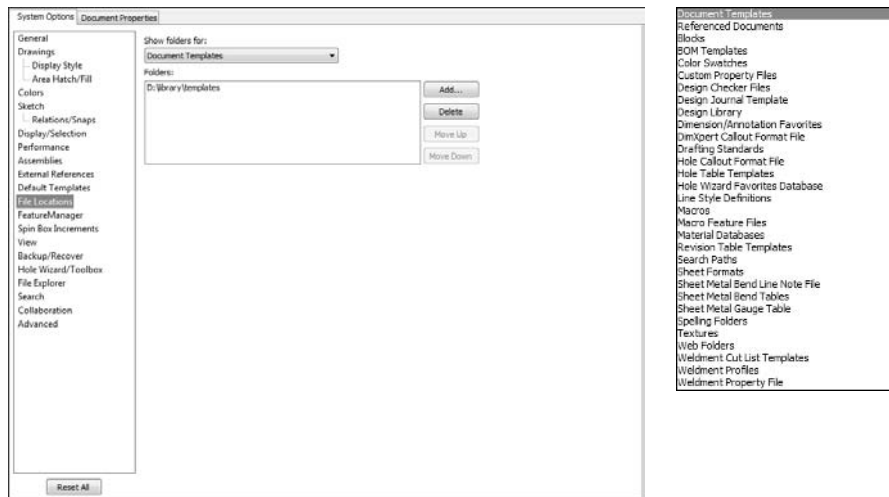
Create a Library folder that includes items such as templates, blocks, library parts, and features, forming tools, and so on. This folder should be separate from your installation directory. This allows you to access the library between computers or multiple SolidWorks installations. ■

File Locations

Use the File Locations settings shown in Figure C.34 to establish libraries of different types of files, stored from version to version.

FIGURE C.34

The System Options ⇨ File Locations page



- **File Locations.** The list of file types for which SolidWorks can specify a specific library location is fairly extensive.

Best Practice

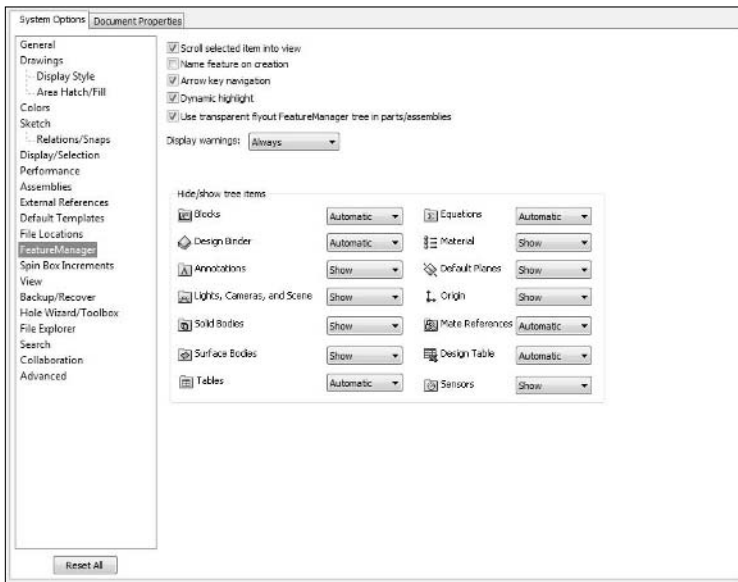
Best practice for File Locations is to create a library folder somewhere other than the default SolidWorks installation folder. When you upgrade SolidWorks, change computers, change jobs, or share data with co-workers, this data is portable. File types that you can store this way include document and table templates, formats, logos, blocks, Design Library parts, library features, forming tools, special fonts, gauge tables, customized symbol files, material libraries, macros, and basically anything else on the list that you have improved from its default, installed state. ■

FeatureManager

The FeatureManager settings, shown in Figure C.35, determine how the FeatureManager reacts to various actions.

FIGURE C.35

The System Options ⇄ FeatureManager page



- **Scroll selected item into view.** When something is selected in the graphics window, the FeatureManager scrolls to display the selected feature. Of course, this happens only if the number of features in the FeatureManager is greater than the available window space can accommodate. Default is on.
- **Name feature on creation.** Although you may seldom want to rename *every* new sketch and feature as it is created, when you want to do so, this is the option to select. Immediately

upon creation of a feature, you are put into Edit mode on the feature name. When I tried using this option, I found that I ended up with a lot of features named zzzz because I usually wanted to zoom out after creating a new feature. Default is off.

Best Practice

It is certainly best practice to name key features and sketches with names that will be easy to identify later. Renaming features is not just for your own convenience but is also useful if anyone downstream from you will need to edit your work. ■

Tip

In addition to renaming features, you can use the Go To function in SolidWorks. Available from the RightMouse Button menu, this function searches the FeatureManager for text that you enter. An even better functionality is the FeatureManager filter located at the top of the FeatureManager. ■

- **Arrow key navigation.** Under normal usage, the arrow keys rotate the model by an angle specified in System Options ⇄ View Rotation. However, when the Rollback bar in the FeatureManager has been selected and arrow key navigation is turned on, the up- and down-arrow keys control the Rollback bar. This allows you to more easily step through the features. One caveat with this is that if you are editing a part in-context, the down arrow will roll to the end in one keystroke. Default is off.

Tip

On the topic of rollback management, the right-mouse button menu has several rollback options, such as Rollback, Roll to End, and Roll to Previous. ■

- **Dynamic highlight.** This is not to be confused with the Dynamic Highlight from Graphics View option, found at System Options ⇄ Display/Selection. This option highlights faces, edges, vertices, sketch entities, and so on as you move the mouse over them in the graphics window. The other setting highlights items in the graphics window as you move the mouse over the features in the FeatureManager. Default is on.
- **Use transparent flyout FeatureManager tree in parts/assemblies.** There are many situations where the PropertyManager takes over the space where the FeatureManager should be, and you need to select something from the tree. When the flyout is activated, as shown in Figure C.36, it can be expanded by double-clicking the name of the feature at the top of the PropertyManager. Default is on.

Note

An alternative to displaying the flyout FeatureManager is splitting the FeatureManager, as shown in Figure C.37. An additional alternative to the flyout FeatureManager is to use the detachable PropertyManager, which is what makes the flyout necessary in the first place. This is my preferred solution. To detach the PropertyManager, just drag the PropertyManager tab out of the FeatureManager area. ■

FIGURE C.36

The flyout FeatureManager, activated through the PropertyManager

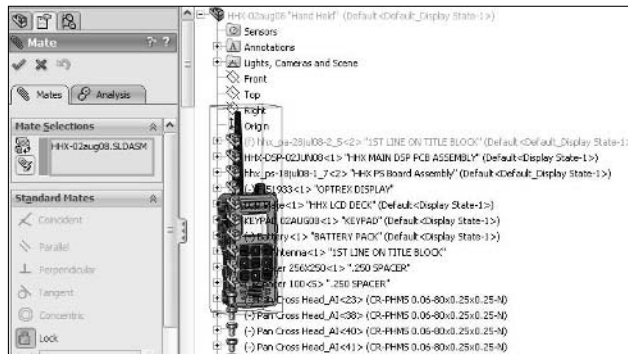
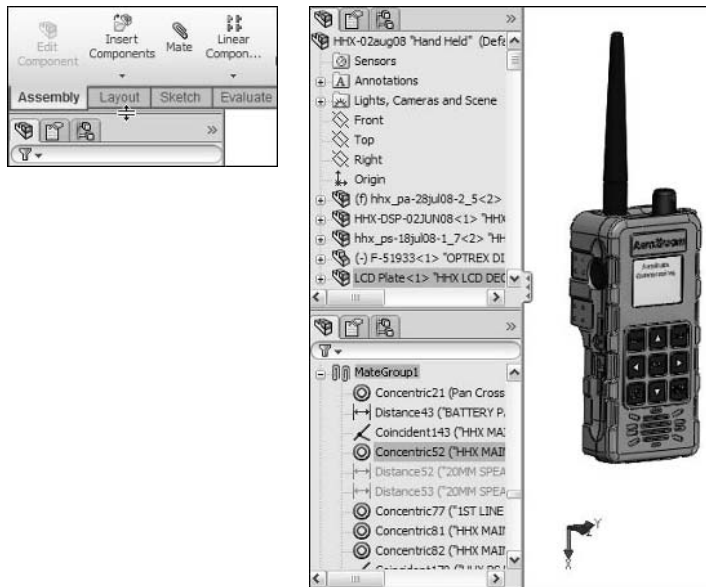


FIGURE C.37

Splitting the FeatureManager



- **Display warnings.** The options are Always, Never, and All But Top Level. Never and All But Top Level may be options that you want to use when your boss is looking over your shoulder, but for real-world work, you should select the Always option so that you know when there is a problem. Default is Always.

Best Practice

Best practice for displaying warnings is to set it to Always. Another best practice is to repair errors as soon as you can. Errors may cause more errors down the line, and they affect performance adversely because SolidWorks keeps trying to resolve the error. ■

- **Hide/show tree items.** These are folders and other items that can be shown at the top of the FeatureManager before the part features. Automatic means that the folder is not shown unless it has some content. Hide and Show are the other options.

Note

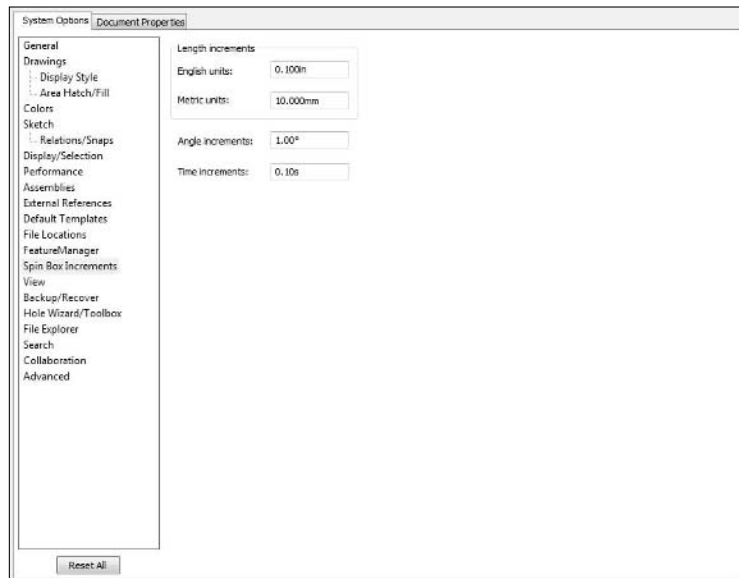
I have noticed that the Automatic setting does not always work. For example, a Surface Body might exist, but the folder is not shown. For this reason, I simply set everything that I want to use to Show, and everything else to Hide. This way there is no guessing if the software is making good decisions or not. ■

Spin Box Increments

Figure C.38 shows the settings for the spin box increments. Spin boxes enable you to increase values by clicking on the arrows. Some spin boxes, such as the Dimension Modify spin box, also have a button that enables you to select from a list of increment values. The +—? symbol shown in the Modify dialog box in Figure C.39 does this.

FIGURE C.38

The System Options ⇨ Spin Box Increments page



Part V: Appendixes

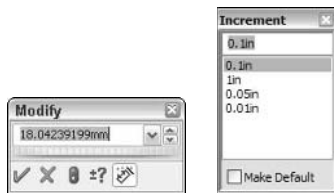
Whenever you click one of the spin arrows on a dimension or number setting, it increments by this value. The exceptions are features such as the tolerance for Fit Splines, where the increment values are determined by the scale of the geometry. The time increments shown in the Spin Box Increments options are used in simulation and motion functionality in SolidWorks.

Note

The Modify dialog box shown in Figure C.39 contains a spin wheel below the number field. By clicking this wheel and dragging right or left, you can increase or decrease the value in the number field. You can also use the scroll wheel on your mouse to drive it. Holding the Ctrl key while dragging or scrolling increases the increment by a factor of ten. Holding the Alt key while dragging or scrolling decreases the increment by a factor of ten. ■

FIGURE C.39

A Modify dialog box with a spin wheel below the number field



You can establish increment values in the Increment box to the right in Figure C.39. You can activate this box by choosing the question mark button in the Modify dialog box.

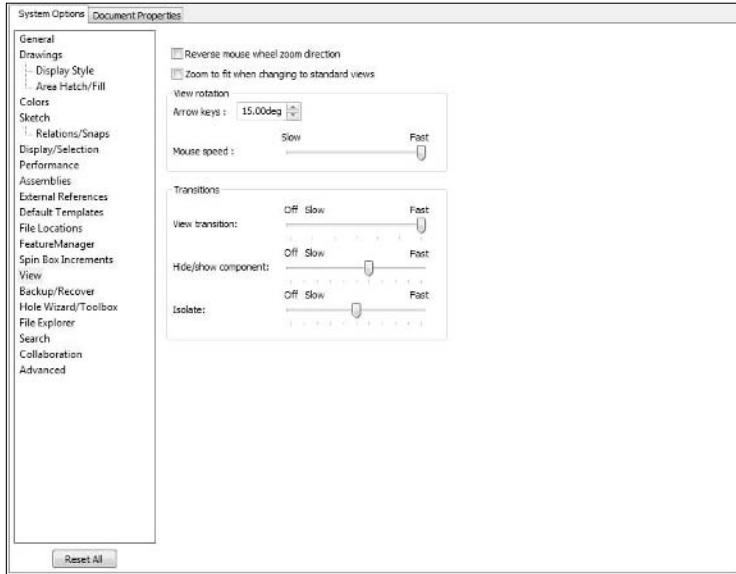
View

The View Rotation options, shown in Figure C.40, are discussed next. View rotation can be done with the mouse, with the keyboard, and automatically by certain functions. The speed of rotation or increment of rotation is controlled by the settings on this page.

- **Reverse mouse wheel zoom direction.** Some SolidWorks users have a substantial background in some other software. Different software packages function differently when it comes to the Zoom function. The default function in SolidWorks is to zoom out (get smaller) when you roll the top of the scroll wheel away from you, and larger when the scroll wheel comes towards you.
- **Zoom to fit when changing to standard views.** This setting controls the behavior when you change, for example, to a front view. When this setting is on, when it changes to a front view, it also zooms to fit. Otherwise, it maintains the same zoom state. Default is off.
- **Arrow keys.** This option determines how many degrees the view will rotate each time a user presses the arrow button. Default is 15.00deg.

FIGURE C.40

The System Options ⇄ View page



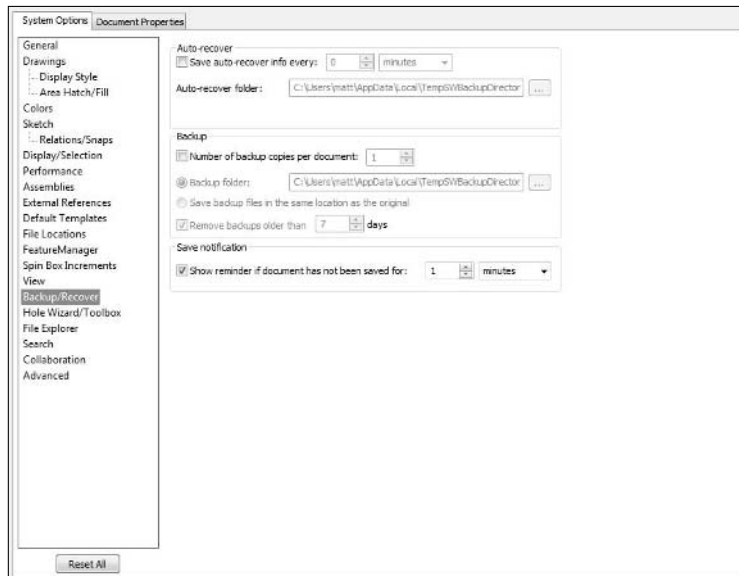
- **Mouse speed.** By default, the SolidWorks mouse settings are the same as the Windows mouse settings for cursor speed. Default is Fast.
- **View transition.** When changing between named views or using a Shift-arrow (90-degree rotation) key combination, the view change animates to help prevent the user from becoming disoriented by an immediate and significant change of view. Moving the slider completely to the left (past the Fast setting) disables the view animation. Default is Fast.
- **Hide/show component.** When components in an assembly are shown or hidden, they fade in and fade out. You can use this option to control the speed of the fade. Default is about 60 percent.
- **Isolate.** When you isolate parts in an assembly, this hides and shows parts. The hide/show fades in or out, and this controls the speed. Default setting is 50 percent.

Backup/Recover

Backup and recover settings control separate functions. They are not the same thing. Figure C.41 and the following list show the settings for these functions.

FIGURE C.41

The System Options ⇄ Backup/Recover page



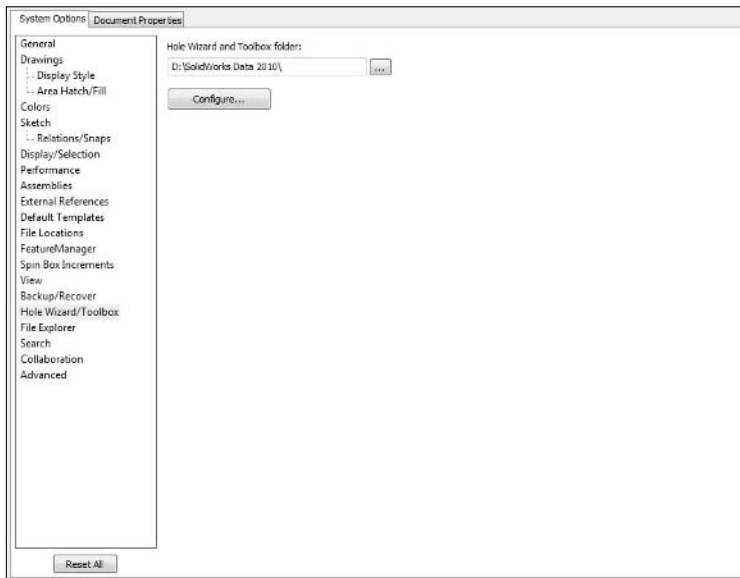
- **Save auto-recover info every.** This setting enables you to specify that SolidWorks save auto-recover information according to a number of changes or an amount of time. You can also specify the folder to be used for the auto-recover data. Auto-recover information is only used if SolidWorks crashes. The next time you launch SolidWorks after a crash, it will offer you the chance to open the recovered files. The auto-recovered files are shown in the Task pane. If SolidWorks exits normally, then the auto-recover information is deleted at the end of the session. This setting can degrade performance, especially of large parts or assemblies, because it periodically saves out data while you are working.
- **Number of backup copies per document.** SolidWorks has separated the Auto-recover settings from the Backup settings so that they are now two clearly different functions. Backup works differently from auto-recover in that backup occurs only during a save. The previously saved version is renamed (and moved to a new location if the Backup folder is specified). The backup functionality also includes the ability to purge backup files by age.
- **Save notification.** This is the kind of setting that you will probably use after you have crashed and lost a lot of work. It is a nice reminder that does not have to get in the way of work, but not everyone has enough patience to deal with persistent (if polite) nagging.

Hole Wizard/Toolbox

The Hole Wizard and Toolbox operate from the same database of hole/hardware size data. The path shown on the page shown in Figure C.42 establishes where the database for these functions is located on your hard drive or network.

FIGURE C.42

The Hole Wizard/Toolbox settings page



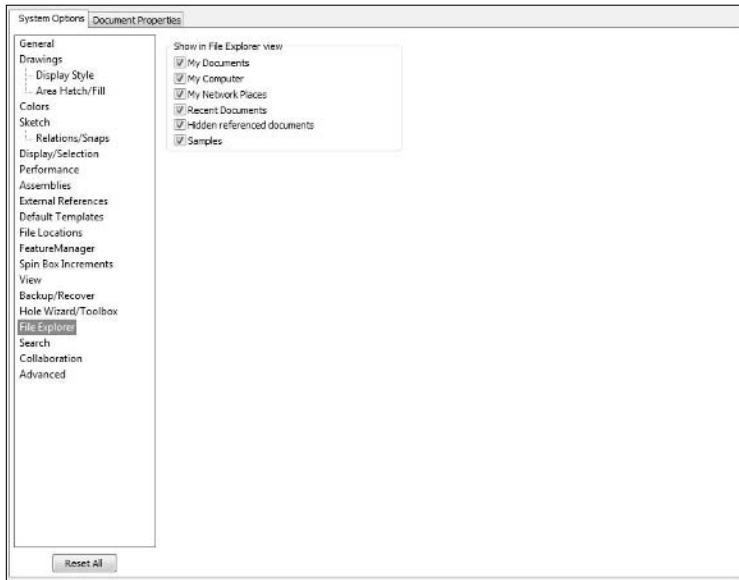
The Configure button shown here displays some of the settings for establishing how Toolbox functions. More details can be found about the Toolbox product in Chapter 10.

File Explorer

The File Explorer is a part of the Task Pane. It can be shown by clicking the tab with the icon of the folder on it. This icon may be replaced if you are using a PDM product. The options for what you can display in the File Explorer are shown in Figure C.43. The File Explorer can be used as a substitute for Windows Explorer right within SolidWorks, as well as showing data with special significance to SolidWorks such as samples that install by default with SolidWorks, files already open in SolidWorks, and recently opened SolidWorks documents.

FIGURE C.43

The System Options ⇄ File Explorer page



- **Show in File Explorer view.** File Explorer is the Windows Explorer–like part of the Task Pane, as shown in Figure C.44. Notice that it has a folder for documents that are currently open in SolidWorks, as well as a recent documents list. The panel will disappear after you use it once, although you can click the pushpin icon to keep it displayed. This is a very useful panel.

Tip

The Task pane can be detached from its default location on the right side of the screen and either allowed to float or dock on the left side. ■

Search

The Search options are shown in Figure C.45. Windows Desktop Search is integrated into the new SolidWorks Search function. In order for this function to work properly, Windows Desktop Search must be installed (it is located on the SolidWorks CD-ROM), and then your drive or drives must be indexed, which will take some time.

FIGURE C.44

The File Explorer panel

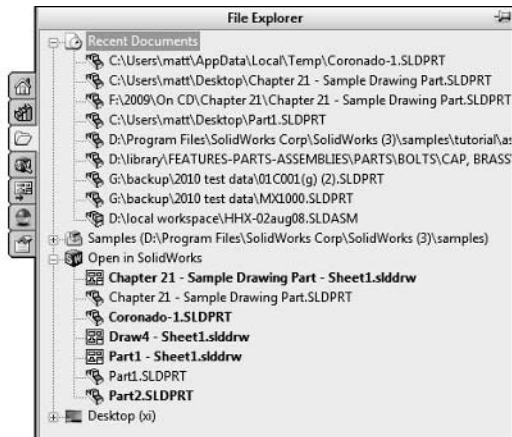
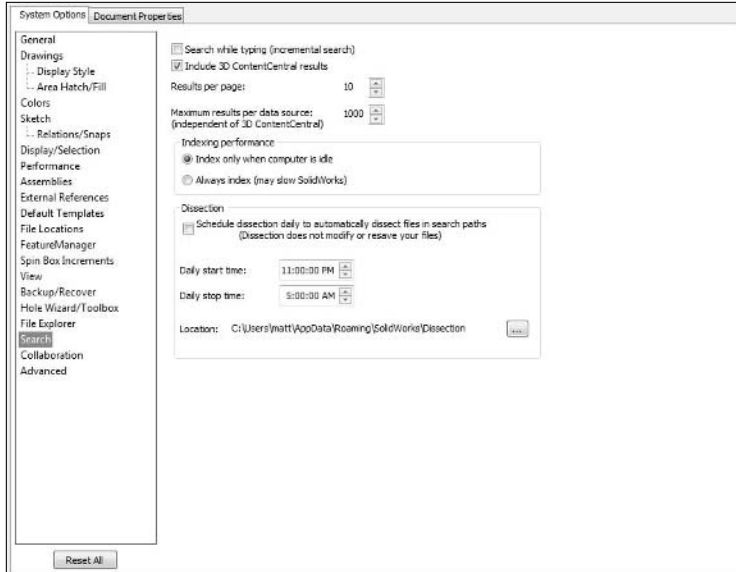


FIGURE C.45

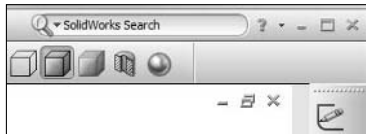
The System Options Search page



- **Search while typing (incremental search).** With this option enabled, the search takes place immediately as you type the string to be searched instead of waiting for you to press Enter. This enables you to see results on partial strings as well as the complete search name. The SolidWorks Search feature is shown in Figure C.46. Default is off.

FIGURE C.46

The SolidWorks Search function



- **Include 3D ContentCentral results.** This option enables the local search to also search 3D ContentCentral Web site for parts as well. Default is on.
- **Results per page.** The search returns results in the Search panel of the Task pane, with easy-to-see previews, filenames, and path information, as shown in Figure C.47. You can also search on custom property information, as well as keywords. Default is 10.

FIGURE C.47

SolidWorks Search results



- **Maximum results per data source.** This setting enables you to specify limits, so that you are not overwhelmed with all of the available data. Default is 1000.

Indexing performance

The Index only when computer is idle setting avoids slowing your computer down when it is in use. The default is on. The Always Index is off by default. I can't imagine a situation in which I would want this setting to be on.

Dissection

Dissection is a process by which SolidWorks searches through the parts in your hard drive, and makes the features and sketches available to you as an automatically created library. To be honest, I have yet to see anyone actually use this. The vast majority of sketches and features I create will never be of use to me again. If I do find one that I will need again, I manually make a library feature of it.

The main thing you need to know about Dissection is how to turn it off. Just make sure the box in front of this Tools Options option is cleared (see Figure C.48)

In early versions of SolidWorks 2008, the setting was turned on by default, and people noticed that their computers started grinding away and flashing SolidWorks screens at 11 p.m. It was very alarming even after you knew what it was and how to control it. In SolidWorks 2009, the setting is turned off by default.

FIGURE C.48

The Windows Desktop Search Indexing Status dialog box

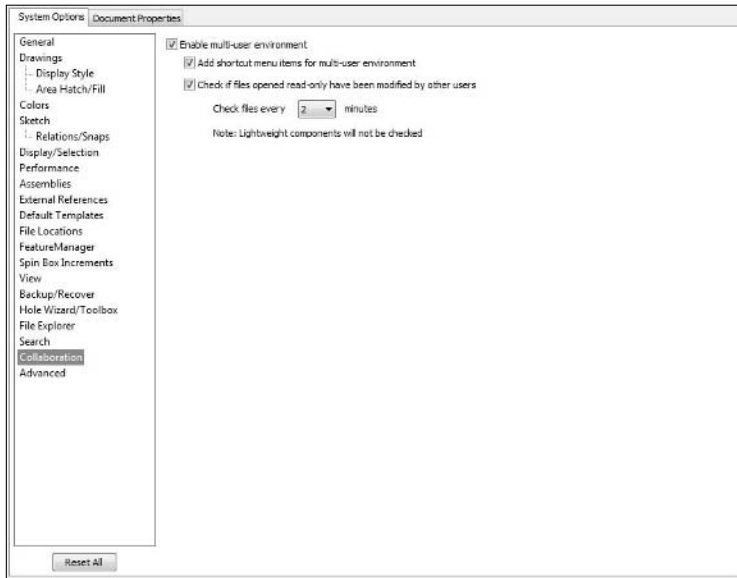


Collaboration

You should consider using the Collaboration settings when setting up a PDM application or a manual document management system between multiple users. The Collaboration settings are shown in Figure C.49.

FIGURE C.49

The System Options ⇄ Collaboration page



- **Enable multi-user environment.** This option enables the rest of the collaboration options.
- **Add shortcut menu items for multi-user environment.** When you select this option, the items Make Read-Only and Get Write Access become available on the Right Mouse Button (shortcut) menu and the File menu.

Tip

There is no indication on the FeatureManager whether individual parts in an assembly are read-only or read-write. The only indication appears at the top of the screen, after the filename, where it is indicated in brackets, for example, as [Read-Only]. Another way to check is to look at the File ⇄ Find References lists; a read-only indicator appears after the filename. ■

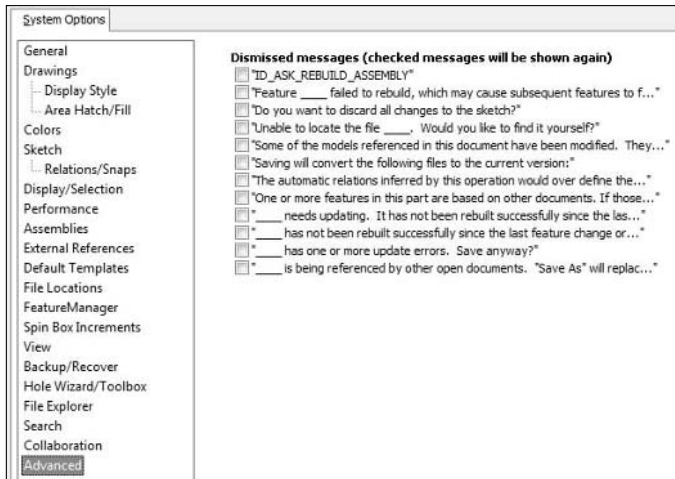
- **Check if files opened read-only have been modified by other users.** This option will check open read-only documents, at the indicated interval, for changes.

Advanced

The Advanced page of Tools ⇄ Options contains all of the messages that you have dismissed and asked SolidWorks not to remind you about again. These are the settings remembered between sessions. If you ever need to get one of these messages back, you can return to this page and click messages to show. Figure C.50 shows an Advanced page populated with several dismissed messages.

FIGURE C.50

The Tools ⇄ Options ⇄ Advanced page



Document Properties

In this case, the term Document Properties refers to the settings that are specific to each individual document. This does not have anything to do with File Properties or Custom Properties.

Document Properties settings are stored in templates and reused when you use the template to create a new document.

I often hear the question, “Can I change the template used by a document?” The answer is always “No.” However, you can change the settings for individual documents to match the template you would have used if you could. Some tools exist that can help you take a list of settings and write them to a group of documents. These tools are available on the Internet for download. One that I am familiar with is an Excel spreadsheet with some VBA programming that handles this function. This has forever been a SolidWorks shortcoming.

The following list details each of the Document Properties settings and gives some insight about when and why you should use each setting.

Drafting standard

In any company I have ever worked with (typically small entrepreneurial ventures rather than large regimented military or automotive/aerospace corporations), I have never seen anyone actually follow a standardized drafting standard to the letter, although I’m sure companies like that exist. Some companies take bits and pieces of different standards and personal preference and put them

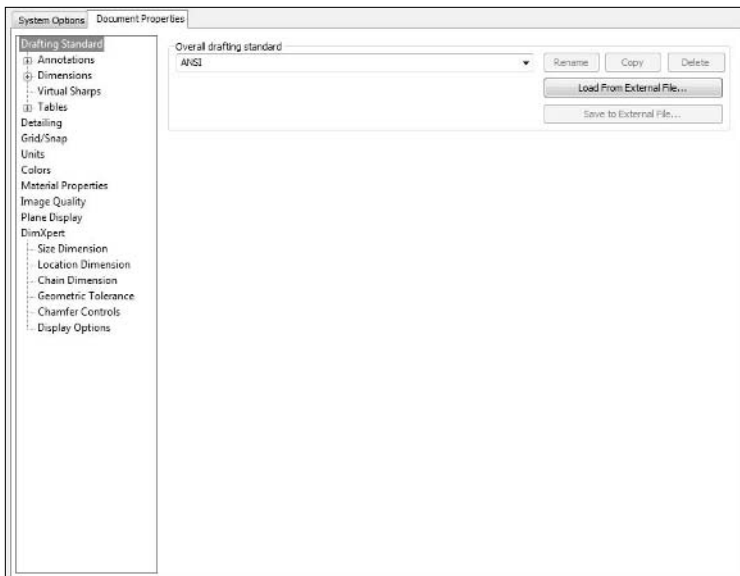
Part V: Appendixes

together into a custom standard that makes sense for their own use. SolidWorks has recognized this and allows you to create your own custom standard or select from one of the standardized standards.

The Drafting Standard page enables you to create your own standard, or load other custom standards from a file, as shown in Figure C.51.

FIGURE C.51

The Drafting Standard page



I think the move of reorganizing the Document Properties and adding prominence to standards capability, including custom standards, is very helpful. There are more settings, they are more easily accessed, and of course the custom standard enables users to take better advantage of the software through improved flexibility.

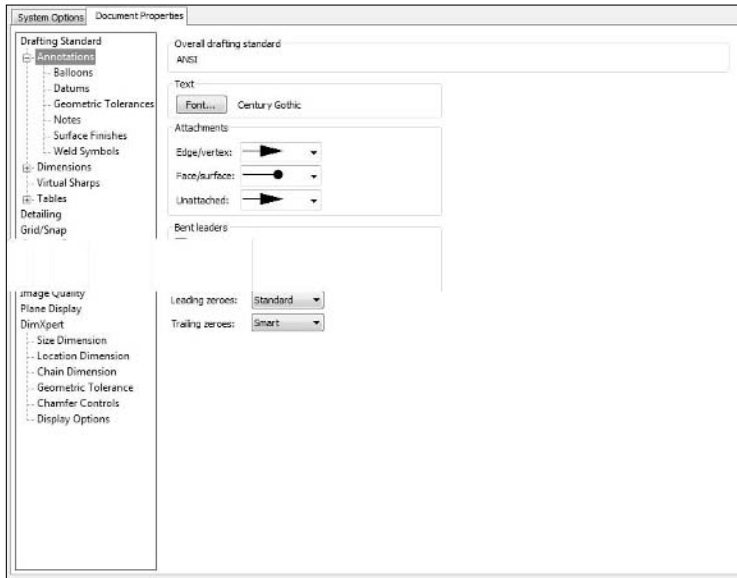
Annotations

The Annotations page enables you to set options that are common to all annotation types. Figure C.52 shows the Annotations page of the Document Properties. Many of these options are reused for other items under the Drafting Standard heading.

- **Text.** This enables you to set the font and size for all annotations text. Notice that dimensions text settings are separate from the annotations text settings.

FIGURE C.52

The Annotations page of the Document Properties window



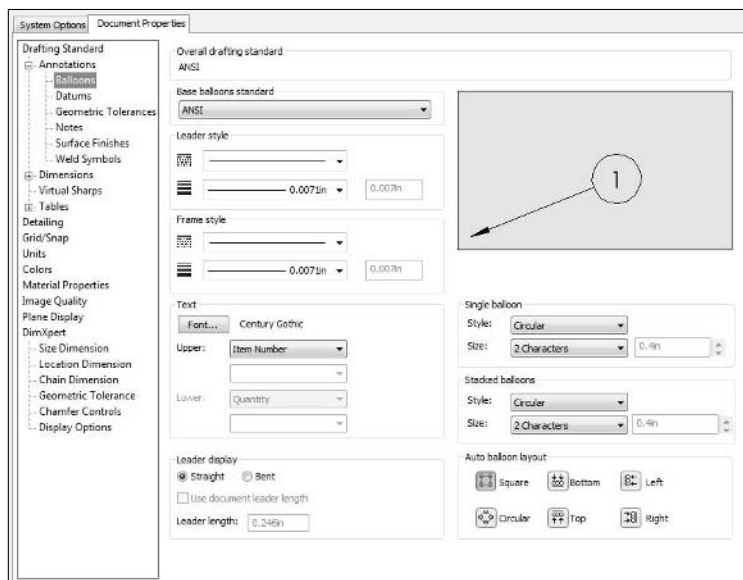
- **Attachments.** This enables you to set the type of arrowhead to be used when an annotation is attached to various types of entities.
- **Bent leaders.** Enables you to tell the software that all new annotations will or will not employ bent leaders by default, and what length those leaders will be.
- **Zeroes settings.** The “Standard” option means that the use of leading or trailing zeroes is to be determined by the drafting standard employed. The “Smart” option means that for metric dimensions, trailing zeroes are always removed for whole values (5.00 becomes 5), and shown for non-zero decimals (5.50 remains 5.50). Trailing zeroes affect implied tolerances and those tolerances based on decimal places of dimensions specified on most drawing borders.

The annotation types listed under the Annotations page reuse most of the same options, shown in the Balloons page in Figure C.53.

- **Leader style.** Establishes the line type and line thickness used for annotation leaders.
- **Frame style.** Establishes the line type and line thickness used for balloon border.
- **Text.** Allows you to override the overall annotations font for that particular type of annotation. With balloons, upper and lower refer to values in a split balloon.

FIGURE C.53

The Balloon options are representative of other annotation type options.



- **Leader display.** Allows you to override the overall annotations settings for bent or straight leaders and bent leader length.
- **Layer.** New annotations can be automatically placed onto layers as they are created.
- **Single and Stacked balloons.** Establish the default type of balloon used for either single or stacked balloons.
- **Auto balloon layout.** Establishes the default layout type when you invoke automatic ballooning.

Dimensions

Many of the options for dimensions are repeated from the other types above, so I will not duplicate definitions for them here. The options for dimensions are either self-explanatory or explained visually on the page, as Figure C.54 shows. The break lines and extension lines settings are only available in drawing documents, not for parts or assemblies.

You can control options for the individual dimension type defaults for angle, arc length, chamfer, diameter, hole callout, linear, ordinate, and radius dimensions as well.

Centerlines/Center marks

The options for centerlines and center marks are shown in Figure C.55.

System Options

Document Properties

Drafting Standard

Annotations

Dimensions

Angle

Arc Length

Chamfer

Diameter

Hole Callout

Linear

Ordinate

Radius

Centerlines/Center Marks

DimXpert

Tables

View Labels

Virtual Sharps

Detailing

Grid/Snap

Units

Line Font

Line Style

Line Thickness

Image Quality

Sheet Metal

Overall drafting standard

ANSI

Text

Font... Century Gothic

Dual dimensions

☐ Dual dimensions display
 ☐ Show units for dual display

☒ Top
 ☐ Bottom
 ☐ Right
 ☐ Left

Primary precision

118.123

Dual precision

118.12

Fractional display

Style:

 Stack size: 100%

Show double prime mark (")

Bent leaders

Leader length: 0.125in

Leading zeros: Standard

Trailing zeros: Smart

☐ Add parentheses by default
 ☐ Center between extension lines
 ☐ Include prefix inside basic tolerance box
 ☒ Show dimensions as broken in broken views

Tolerance...

Arrows

0.04in

0.12in

0.25in

Scale with dimension height

Style:

Offset distances

☐ Annotation view layout

0.236in

0.394in

Break dimension extension/leader lines

Gap: 0.06in

☐ Break only around dimension arrows

Extension lines

Gap: 0.05in

Beyond dimension line: 0.125in

Radius/Diameter leader snap angle: 15deg

FIGURE C.55

System Options

Document Properties

Drafting Standard

Annotations

- Balloons
- Details
- Geometric Tolerances
- Notes
- Surface Finishes
- Weld Symbols

Dimensions

- Centerlines/Center Marks**
- DimXpert
- Tables
- View Labels
- Virtual Sharps

Detailing

Grid/Snap

Units

Line Font

Line Style

Line Thickness

Image Quality

Sheet Metal

Overall drafting standard

ANSI

Centerlines

Centerline extension: 6.35mm

Center marks

Size: 2.54mm


☒ Extended lines


☐ Centerline font


Slot center marks


☒ Orient to slot

☐ Orient to sheet









Part V: Appendixes

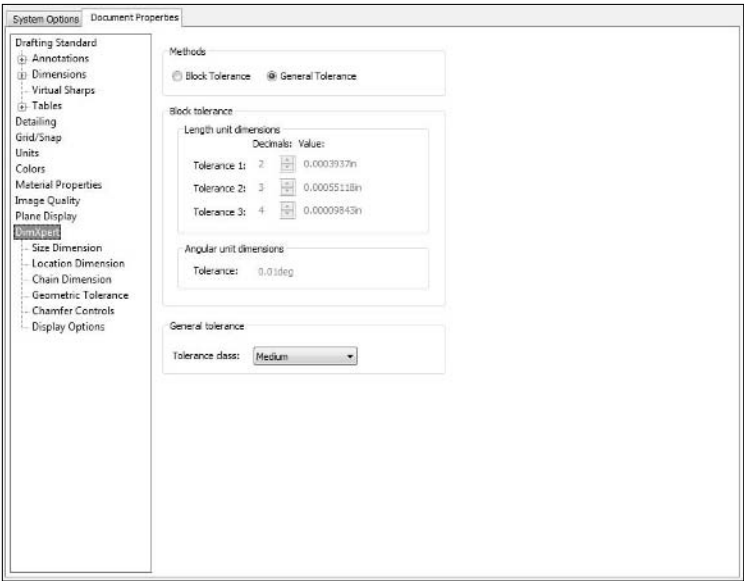
- **Centerline extension.** Refers to any lines extended beyond the “+” center mark. Extension distance refers to the distance past the arc or circular edge.
- **Center marks.** Size refers to the size of the “+”.
- **Extended lines.** Refers to lines extended beyond the “+” center mark.
- **Centerline font.** Applies the centerline font to extended lines.
- **Orient to slot and Orient to sheet.** Means that the “+” will be oriented such that it is square with the sheet or with the slot. If the slot is at an angle, and you choose to orient the center mark to the slot, the “+” will also be at an angle.

DimXpert

The DimXpert options are different for part and drawing documents. They are far more extensive for part documents, so I show the part options in Figure C.56 and Figure C.57. DimXpert options help you set up how things like instance counts or slot callouts are displayed, in addition to helping with options to make tolerancing faster and easier.

FIGURE C.56

DimXpert options for part documents



Tables

The options that you control through these pages are the defaults for layout and appearance for BOMs, General tables, Hole tables, and Revision tables. Here Figure C.58 shows the Bill of Materials options as a representation of the options available in the other table types.

FIGURE C.57

DimXpert Display options

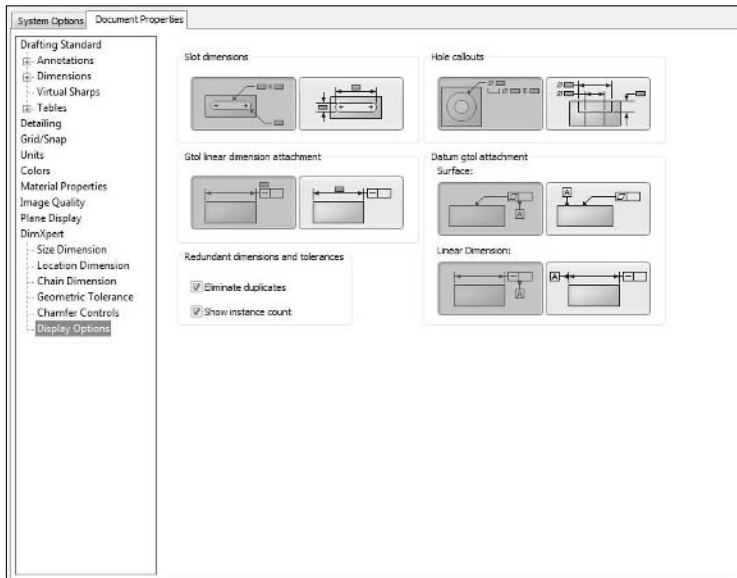
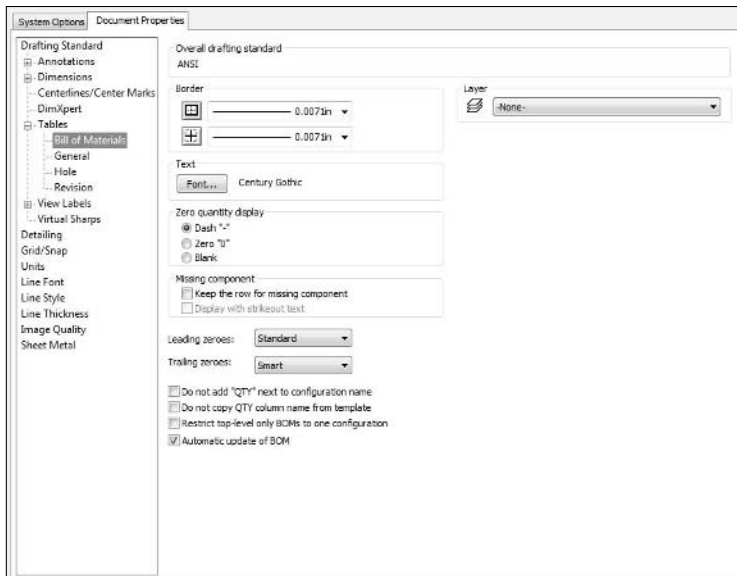


FIGURE C.58

Bill of Materials options

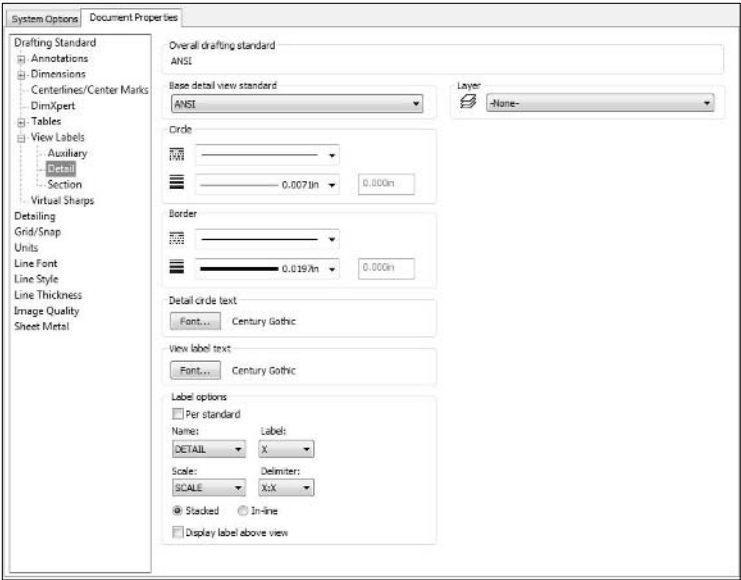


View Labels

Auxiliary, Detail, and Section views use arrows and labels to identify and show the alignment and cuts used in the views. The options under View Labels page control the various elements involved, which should be self-explanatory. Figure C.59 shows the Detail view label options, which are representative of the other types of views.

FIGURE C.59

View Labels Detail options



Virtual Sharps

Virtual Sharps display on a drawing (and in sketches) is controlled on this page. A virtual sharp is the corner where two edges would intersect if they were not interrupted by a fillet, chamfer, or other item breaking the edge. The options for Virtual Sharps display are shown in Figure C.60.

Detailing

Many of the Detailing options are also available by using the Right Mouse Button on the Annotations folder in the FeatureManager and selecting Details. The page also displays differently depending on if you are editing the properties of a part, assembly, or drawing. Figure C.61 shows the available options for a drawing document.

FIGURE C.60

Virtual Sharps display options

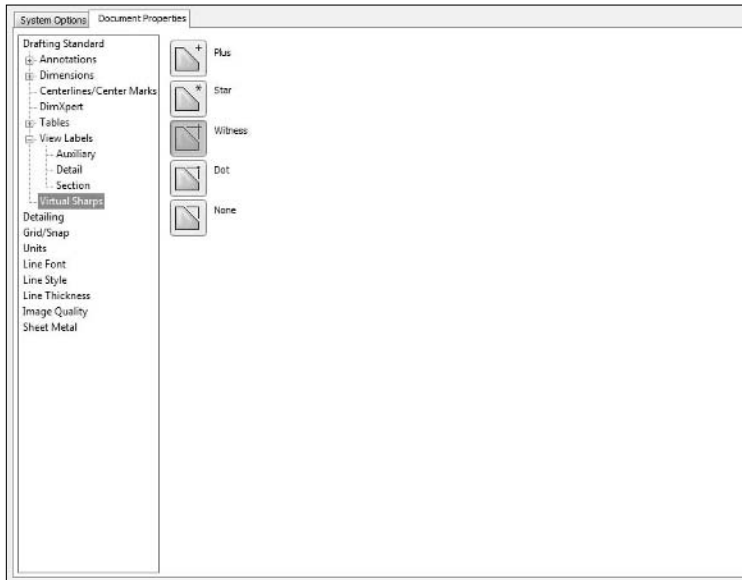
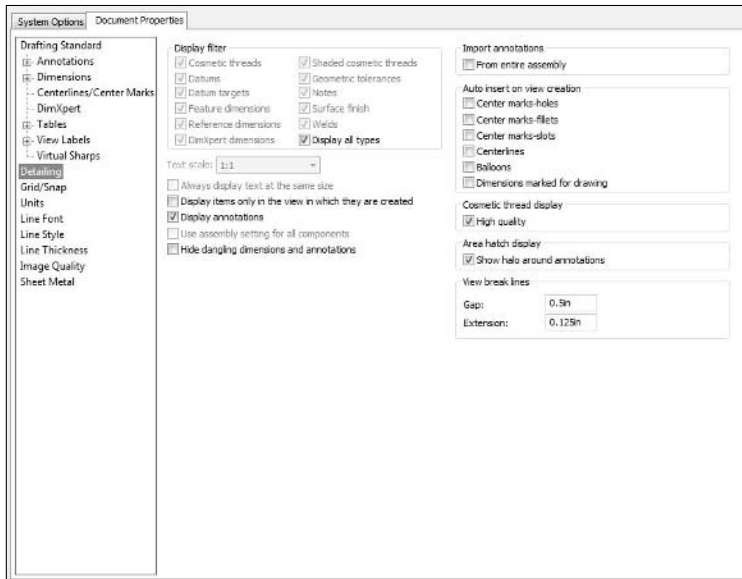


FIGURE C.61

Detailing display options



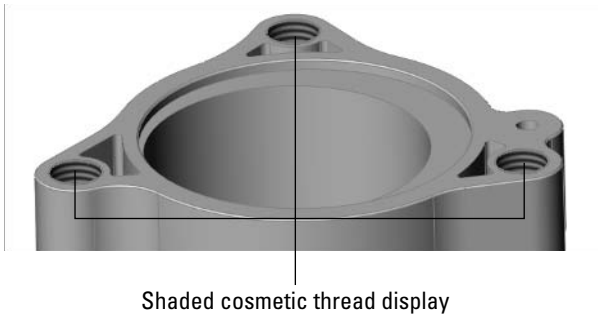
Part V: Appendixes

Display filter

Select the entity types to be displayed in this particular document or template. Display all types shows everything. Of special note here is the Shaded cosmetic threads. The difference between normal Cosmetic threads and Shaded cosmetic threads is shown in Figure C.62.

FIGURE C.62

Shaded cosmetic thread



Text scale

The Text scale option is only available for parts and assemblies, and only when Always display text at the same size is off.

Always display text at the same size

This option causes annotations and dimensions to appear the same size on the screen regardless of zoom state. It is not active in drawings because drawings always need to show text at 1:1 size.

Display items only in the view in which they are created

This setting is sometimes frustrating, hiding annotations unless the model is in exactly the view in which the annotation was created. It becomes very easy to lose annotations. On the other hand, if you are disciplined in how you create annotations (consider using Annotation Views), this can help reduce clutter in drawing and model views.

Use assembly setting for all components

This option makes it easier to see all the component annotations in the assembly using a single set of settings, regardless of differences between settings in individual component files. This is only available in assemblies.

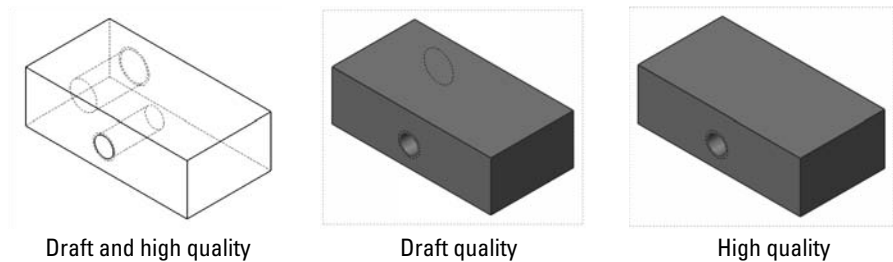
Hide dangling dimensions and annotations

SolidWorks has become increasingly “error-phobic” in recent releases, offering more and more options to make errors invisible. Best practice is definitely on the side of fixing problems as they come up, rather than masking them over.

- **Import annotations.** Selecting From entire assembly places dimensions from all the parts in an assembly on the assembly drawing when views are created. Default is off.
- **Auto insert on view creation.** Determines which entity types you want to be automatically inserted from the model into drawings when the view is created. Defaults for all of these are off.
 - Center marks-holes
 - Center marks-fillets
 - Center marks-slots
 - Centerlines
 - Balloons
 - Dimensions marked for drawing
- **Cosmetic thread display.** Cosmetic threads on the backside of a part will be hidden if the High quality option is on (see Figure C.63). For speed, turn this option off. Figure C.64 demonstrates that this setting only makes a difference for shaded model views on the drawing. For any wireframe mode, the hidden Cosmetic thread is not shown unless the hidden lines of the view are visible.

FIGURE C.63

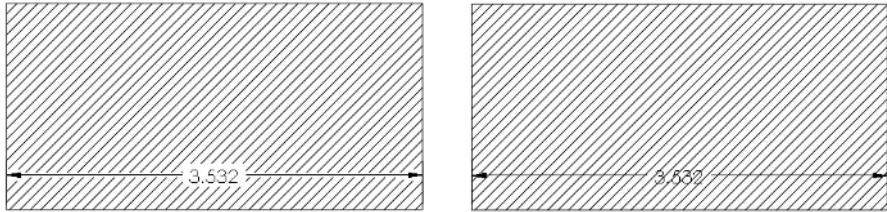
Cosmetic threads: High quality and Draft quality



- **Area hatch display.** When Show halo around annotations is on, a halo, which is simply a gap in the hatch pattern, appears, as shown in Figure C.64.
- **View break lines.** When you create a broken view, the break lines by default have a .5-inch gap between them, and extend past the view border by .125 inch.

FIGURE C.64

Show Halo around annotations turned on and off

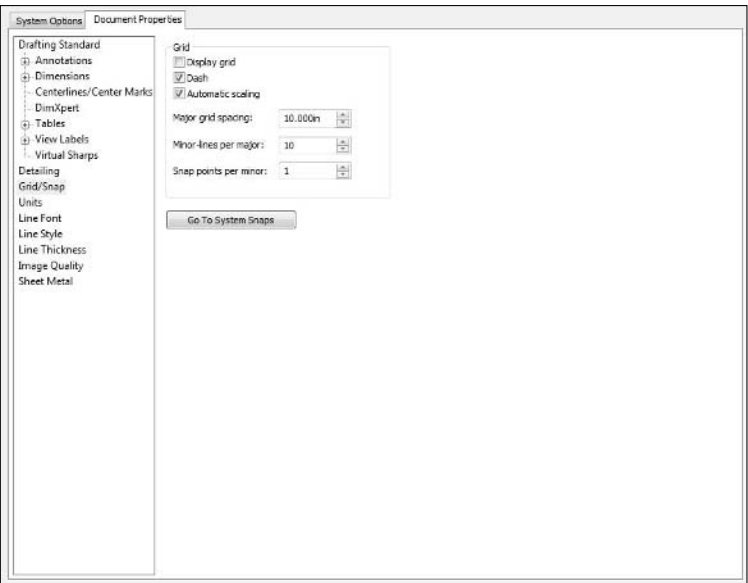


Grid/Snap

One of the first questions new users tend to ask is how to turn off the grid in sketches. This is a bit ironic, since the grid is most useful for new users, mainly to help them keep track of when they are in or out of sketch mode. The Grid/Snap options are shown in Figure C.65.

FIGURE C.65

The Grid/Snap options page



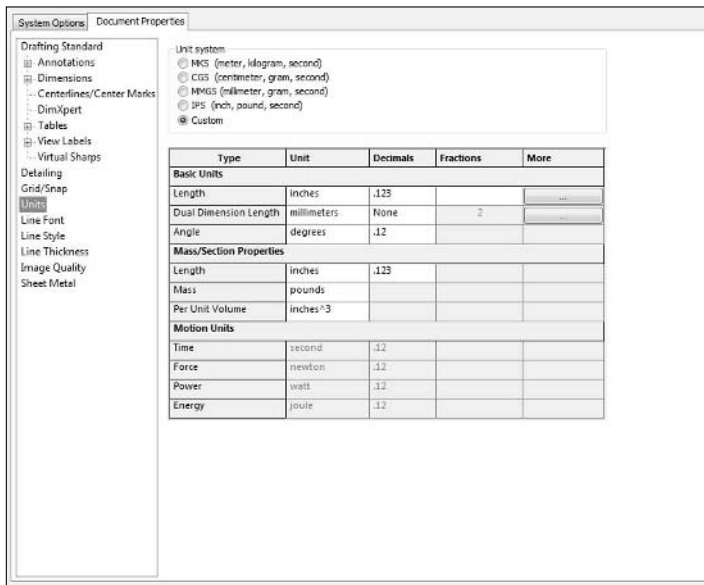
- **Display grid.** This option turns on/off the display of the grid in the sketch.
- **Dash.** Minor grid lines, specified below, are dashed. If this option is off, the minor grid lines are solid.
- **Automatic scaling.** As you zoom in and out, the minor grid goes from ten divisions to five to two, depending on your settings.

Units

The Units options page enables you to select either a predefined unit system or create custom settings including primary and dual length units, angular units, as well as special units for mass properties/ section properties. Units of force are included for the strength properties associated with COSMOSWorks Xpress. The Units options page is shown in Figure C.66.

FIGURE C.66

The Units options page



After the “How do I turn off the grid” question, the next most popular setting question is “How do I make SolidWorks remember that I want to use inches instead of mm?” Of course the answer is templates. Set the units the way you want them on a blank document (or delete all the features from an existing document) and save the file as a template. This applies to parts, assemblies, and drawings.

Fractions

SolidWorks can use fractional dimensions and convert decimals to fractions. To set up fractions, set the Denominator box to the smallest division that you will want to use, and check the Round To Nearest Fraction box. Be aware that when you do this, some values will be approximated.

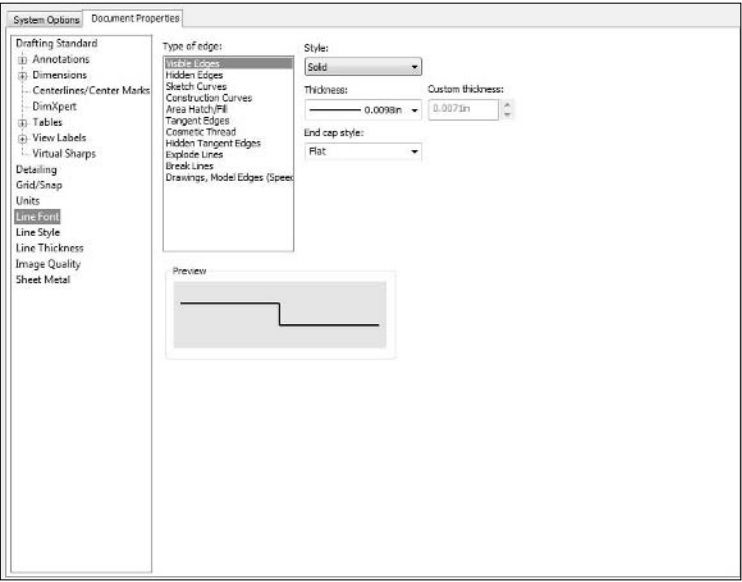
The only units that will accept fractional values are all of the English units except Feet. Interestingly, you can use fractional mils (.001, thousandths of an inch) and microinches (.000001, millionths of an inch). Fractions are most appropriate for inches and feet and inches.

Line font

Many types of edges, identified in Figure C.67, can have special line fonts assigned to them. You can even create custom line fonts, although the custom line styles are limited to being used in drawings, not in parts or assemblies. New line styles are created in the next section on line styles.

FIGURE C.67

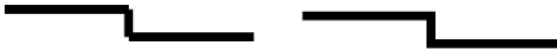
Line Font options page



The one setting that is not self-explanatory on this page is the End cap style. Using the Flat style, thicker line types have poorly jointed corners, as shown in Figure C.68 in the image to the left. The square end cap is shown to the right. If you use thick lines and see this happening, now you have a way to fix it.

FIGURE C.68

Using End cap style to make thicker lines look better

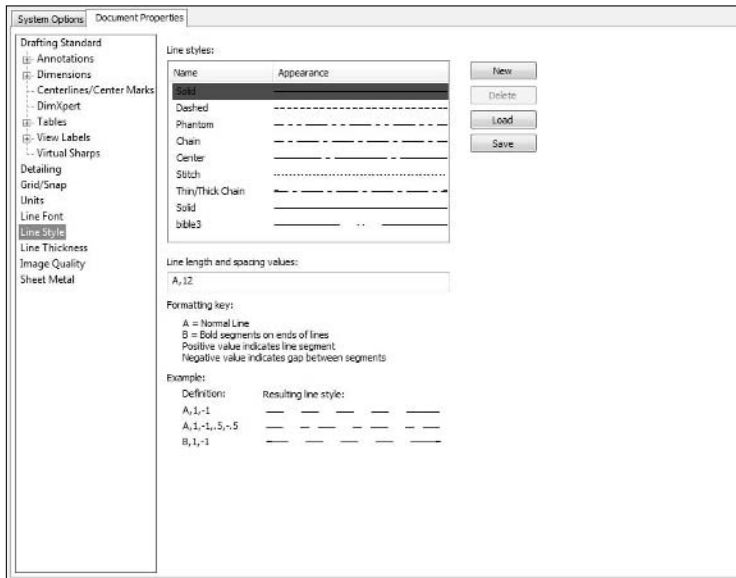


Line style

The Line Style options page is where you can modify existing styles and even create new line styles. The instructions are right there on the page, it is a simple code to create special line styles. This is shown in Figure C.69.

FIGURE C.69

The Line Style options page

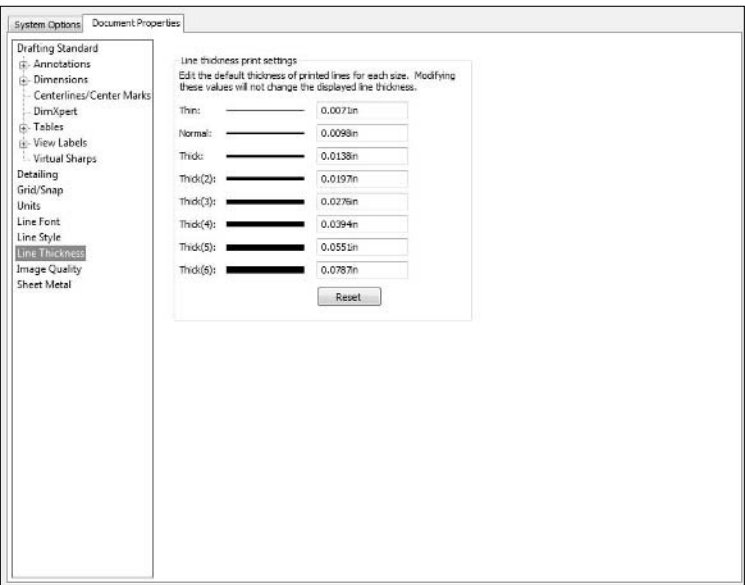


Line thickness

The thing to remember with line thickness is that the visual thickness has no relation to the actual thickness dimension that you assign to the line. You could assign the Thin line thickness to a value of 2 mm, and it would not change the way the Thin line displays. The options for this are shown in Figure C.70.

FIGURE C.70

The Line Thickness options page



Colors

The Document Properties Colors options page enables you to specify colors for certain feature types in part and assembly documents. System Options Colors options specify colors for portions of the interface rather than part geometry. The Advanced button enables you to specify transparency and other effects.

Curvature

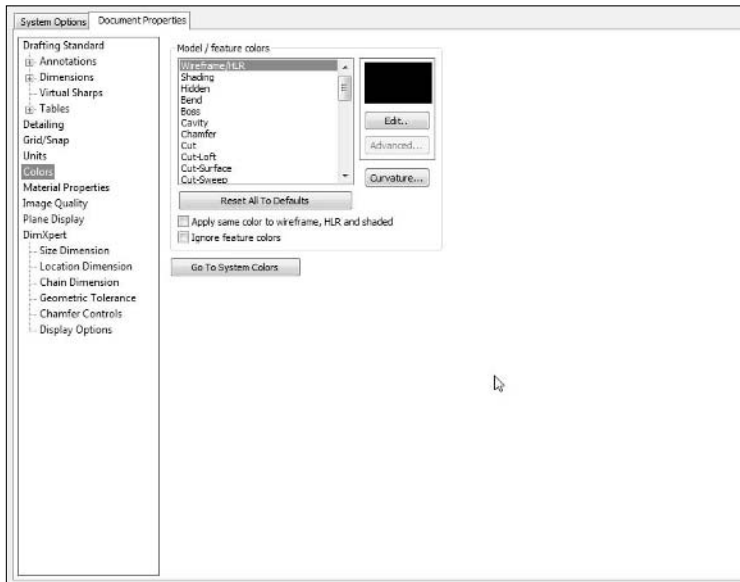
The curvature window is shown to the right in Figure C.71. This enables you to specify the color spread for various curvature values. Remember that curvature is the inverse of radius ($c=1/r$), so the smaller the radius, the bigger the curvature.

Apply same color to wireframe, HLR, and shaded

This setting is only valid in part documents. This is probably one of the least-appreciated options in SolidWorks. Many parts from SolidWorks Corp. do not use this setting, so when you open an assembly where the parts do not have this set, and turn the assembly to wireframe, all of the edges are black. This will make you feel like you are back 15 years ago modeling wireframes in AutoCAD. Do yourself a favor and save your part templates to use this setting.

FIGURE C.71

The Document Properties Colors options page



Ignore feature colors

The Ignore feature color option sets parts with colors applied to the feature back to a more homogeneous color scheme. Colors may still be applied to faces or bodies, and will not be affected by this setting.

Best Practice

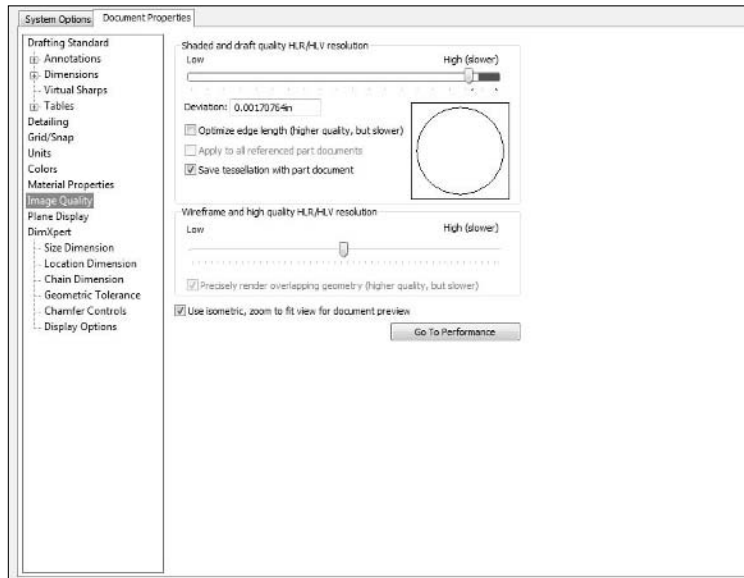
Use the Apply same color to wireframe, HLR, and shaded option to be able to differentiate parts in an assembly shown in wireframe by color. The alternative is a black-on-black wireframe display, which is guaranteed to make you grumpy quickly. ■

Image quality

Image quality can have a definite affect on the performance of parts and assemblies. This setting is highly dependent on a couple of things, primarily your graphics card, but also including part complexity and number of parts in an assembly. The Image Quality options page is shown in Figure C.72.

FIGURE C.72

The Image Quality options page



Best Practice

The best practice recommendation for image quality is to set it as low as you can stand it. This may be at different levels for different parts. ■

The way SolidWorks (and indeed most 3D graphics applications) displays curved 3D geometry is to create a set of smaller, flat triangular facets. This is called tessellation. If these triangles get too large, curved surfaces are not recognizable (they might look more like hexagons than circles), but if the triangular facets get too small, your graphics card may not be able to process them quickly enough. So image quality settings are definitely a compromise. The slider at the top of the window applies to shaded models.

Deviation

The Deviation setting measures how far from the real shape the displayed shape is. By making a curved edge into a series of straight lines, the number shown represents the maximum distance from the line to the curve. As the quality gets higher, the deviation gets lower.

Optimize edge length

SolidWorks is not explicit about this, but it is assumed that what is being “optimized” is the quality of the display, not the speed of the system.

Apply to all referenced part documents

This setting is valid for assembly documents only. In an assembly, individual part display quality settings are used unless this option is on. However, if you have your part quality settings high for good visualization, and the assembly setting low for good speed, you will need to change the part settings back to high after you have them open in the assembly. This becomes a bit of a frustration unless you can find a good middle ground or a macro that changes the settings for you quickly and easily.

Save tessellation with part document.

This setting is valid for part documents only. This option toggles the tessellated display data being saved in the part file. This can have an impact on save time and file size. Do not get over excited about turning this option off; however, you need to remember that it is faster to read existing data than it is to recalculate it.

Wireframe and high quality HLR/HLV resolution

This slider works for wireframe display and drawings. Again, set it as low as you can stand it if you are having performance problems.

Plane display

Planes in SolidWorks can be displayed either with just a rectangular outline or with a transparent shaded face. Each “side” of the plane can be viewed with different colors. Although immediately this might not seem useful, I find it very useful when working in detail on a larger part. It helps me keep my bearings. The slider shown in Figure C.73 enables you to change the transparency of the planes as well.

Plane intersections can also be shown by a dotted line of a specified color, but since you cannot select them or actually use them for anything other than visualization, I tend to turn them off.

Sheet metal

The extent of the sheet metal drawings options is to enable you to control flat pattern colors and bend notes on drawings, as shown here in Figure C.74.

FIGURE C.73

The Plane Display options page

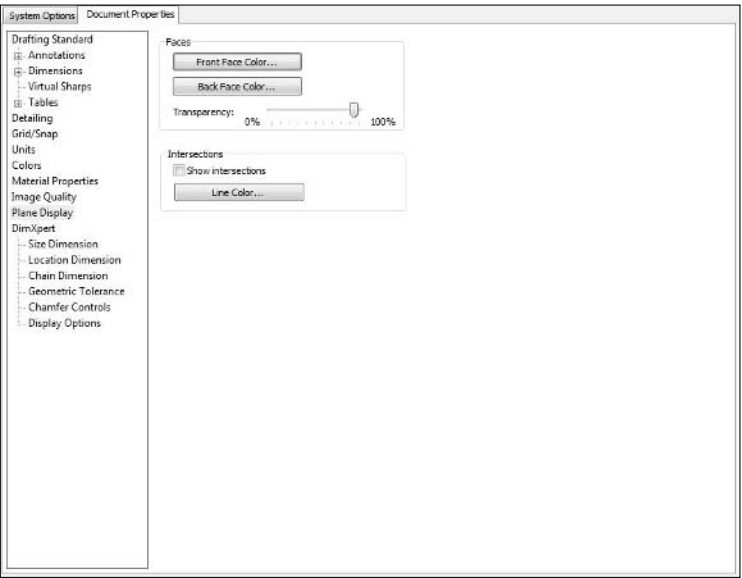
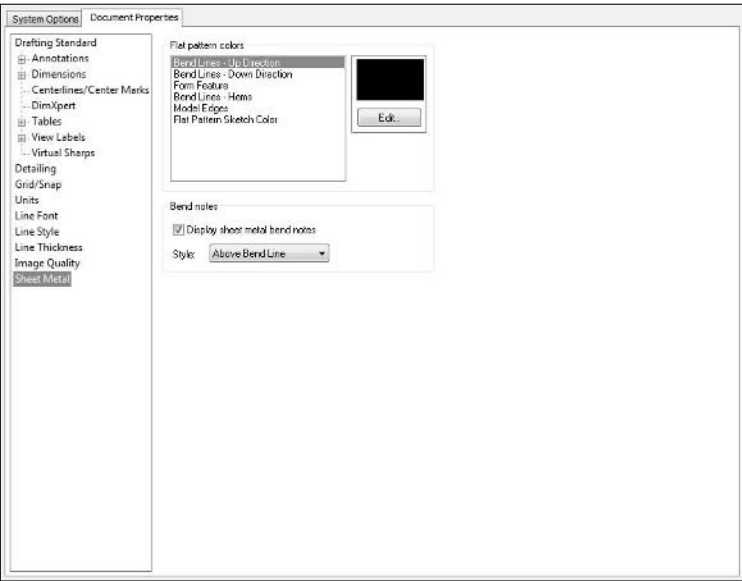


FIGURE C.74

The Sheet Metal options page



Index

Numerics

- 2D CAD, 280
- 2D drawing, 3, 7, 58
- 2D sketching, 291
- 2D standards, 158, 253–256
- 2-point rectangle, 283
- 3 Gig Switch tool, Windows, 93–94
- 3D CAD, 280
- 3D Content Central, Customer Portal, 248, 468
- 3D model, 7, 254
- 3D sketching, 291
- 3D standards, 158, 256–261
- 3-point rectangle, 283
- 24-character serial number, 124
- 30-day courtesy license, 152, 381
- 32-bit operating system, 21, 93, 116, 402
- 64-bit operating system, 21, 93, 116, 402

A

- ABCR (As Built Control Record), 61
- About Windows window, Windows, 89–90
- AC adapter brick, 27
- academic texts, 107
- activating licenses, 133–134, 150–153, 379–380
- Active Directory, 138–139
- Add New User interface, 429
- Add shortcut menu items for multi-user environment option, 339–342, 526
- add-in software, 334
- Add-Ins dialog box, 220
- Add-Ins menu, 221
- additive permissions, 429
- Adjust spacing when dimensions are deleted or text is removed option, System Options tab, 492
- Admin Guides area, Customer Portal, 469
- Administrative Image
 - installing service packs, 356–357, 363–366
 - installing SolidWorks from, 130–139
 - serial numbers and, 122
 - updating service pack on multiple machines, 360
- Administrative Image Option Editor, 131–136
- administrative install, service pack, 359
- Administrative Options tab, Option Editor, 134
- administrator access, installing software, 115
- Administrators panel, VaultAdmin tool, 414
- admin.pdmw item, 412
- Advanced mate, 296
- Advanced option, New Document interface, 179
- Advanced page, System Options tab, 526–527
- Advanced Search dialog box, Workgroup PDM, 457
- Advanced Server license type, 412
- Advanced Server software, 395
- aggregator, 473
- Alibre, 373
- Allow attachments to documents owned by other users setting, 413
- Allow auto-update when opening drawings option, 490
- Allow change document properties setting, 413
- Allow delete local copy on check in check box, 423
- Allow latest revision overwrite setting, 413
- Allow multiple contexts for parts when editing in assembly option, 510
- Allow Multiple Contexts setting, 167
- Allow revision bumping setting, 413
- Allow selection in HLR and shaded modes option, 502
- Allow selection in wireframe and HLV modes option, 502
- Allow users to create subprojects setting, 413
- Allow users to rename documents in the Vault setting, 414
- alphabetical revision scheme, 57
- Always display text at the same size option, 191, 536
- Always Resolve option, Check out-of-date lightweight components setting, 506
- Always resolve sub-assemblies setting, 506
- Always use these default document templates option, 512–513
- American National Standards Institute (ANSI), 191, 479
- American Society of Mechanical Engineers (ASME)
 - Web site, 478–479
 - Y14.1 standard, 478
 - Y14.1M standard, 478
 - Y14.2 standard, 478
 - Y14.3 standard, 478
 - Y14.4 standard, 478

Index

American Society of Mechanical Engineers (ASME) (*continued*)

- Y14.5 standard, 478
- Y14.5M, 255–256
- Y14.6 standard, 479
- Y14.7 standard, 479
- Y14.8 standard, 479
- Y14.13 standard, 479
- Y14.18 standard, 479
- Y14.24 standard, 478
- Y14.31 standard, 479
- Y14.32 standard, 479
- Y14.34 standard, 478
- Y14.35 standard, 478
- Y14.36 standard, 479
- Y14.37 standard, 479
- Y14.38 standard, 478
- Y14.40 standard, 479
- Y14.41 standard, 181–182, 191, 478
- Y14.42 standard, 478
- Y14.43 standard, 479
- Y14.44 standard, 479
- Y14.100 standard, 478

American Society of Testing and Materials (ASTM), 479

Anna's SolidMuse blog, 470

Annotations page, Document Properties tab, 194, 528–530

ANSI (American National Standards Institute), 191, 479

Anti-alias edges/sketches option, System Options tab, 503

antivirus applications, 33–35, 97, 115, 354

Appearances and Scenes icon, Design Library control interface, 235

Apple computers, 19

Apply same color to wireframe, HLR, and shaded option, 185, 542–543

Apply to all referenced part documents option, 192, 545

arc centerpoints, 496

Archive project button, VaultAdmin tool, 419

Area hatch display option, Document Properties tab, 537

Area Hatch/Fill page, System Options tab, 492–493

Arial Black font, 195

Arrow key navigation setting, 515

Arrow keys option, 518

As Built Control Record (ABCR), 61

*.asmdot extension, 178

ASME. *See* American Society of Mechanical Engineers

Assemblies page, System Options tab, 508–509

assembly documents, 29, 51, 184

assembly modeling, 176, 291–297

assembly templates, 191–192

Assembly transparency for in-context edit setting, 502

assembly units, 191

ASTM (American Society of Testing and Materials), 479

At the Intersection sketch relation, 290

ATI FireGL card, 23

Attachments option, Document Properties tab, 529

Audio Video Interleave (AVI), 108

Auto balloon layout option, 530

Auto insert on view creation option, 537

Auto insertion setting, Detailing page, 195

Auto-create Design Table functionality, 214

Automatic Relations option, 189, 500

Automatic scaling option, 539

automatic updates, 98

Automatically edit macro after recording setting, 488

Automatically Generate Names for Referenced Geometry setting, 169, 511

Automatically hide components on view creation option, 490

Automatically load components lightweight setting, 506

Automatically populate View Palette with views option, 491

Automatically scale new drawing views option, 490

automating tasks

- help resources, 313–314
- macros, 302–311
- overview, 301–302

automation

- Task Scheduler, 64
- Toolbox, 210

Auto-show PropertyManager option, 487

Auto-size PropertyManager when panels are split option, 487

AVI (Audio Video Interleave), 108

B

Back button, Create Basic Task Wizard, 66

Back face, plane, 186

Back up computer option, 76, 78

Back up files option, 76

Background appearance setting, 494

backing up, 76–83, 396

BackOffice, 382

Backup and Restore Center, 77

Backup/Recover page, System Options tab, 519–520

backwards compatibility, 139

Base Flange method, 257

basic input/output system (BIOS), 83–84

batch files, 65

Begin Recording button, 329

Bent leaders option, 529
 best practices. *See also* SolidWorks Workgroup PDM
 accessing user interface, 445–449
 in assemblies, 291–297
 concept of, 280–282
 defined, 158
 documentation classification, 48
 for features, 288–291
 fixing rebuild errors, 175
 for in-context modeling, 297–300
 for installation of SolidWorks Workgroup PDM, 394–395
 modeling detail, 175
 overview, 173, 279
 putting fillets at bottom of tree, 175
 relating first feature to part origin, 174
 relations to stable references, 174
 simplified configurations, 175
 for sketching, 282–286
 symmetry, 174
 Verification on Rebuild option, 175
 beta testing, 354, 374–375
 beta versions, installing, 117
 Bills of Materials (BOMs)
 custom properties, 187, 266–269, 277
 document numbers, 43
 filenames, 43
 metadata and, 240
 part numbers, 43, 225
 placing table anchors, 203
 Workgroup PDM, 392
 Bind ownership to SolidWorks read-write / read-only access
 setting, 414
 BIOS (basic input/output system), 83–84
 blocks, 379
 blogs, 323, 470–473
 bodies, 290
 BOMs. *See* Bills of Materials
 Boot Camp, 19
 Boot tab, Windows, 88
 bootable media, 83
 Boundary surface, 59
 Boxx brand, 19
 brevity, 283
 broken reference symbol, 170
 Browser folder, 211
 browsing, 49–50
 bugs, 326–327
 Bulk Check In button, VaultAdmin tool, 419

C

CAD administration
 blogs, 473
 versus design, 9
 expert users, 11
 Information Technology, 9–10
 mentoring, 11–15
 overview, 3–5
 software, 6
 SolidWorks versus AutoCAD, 7–8
 CAD Fanatic blog, 471
 Cam mate, 296
 capacity, hard drive, 23
 catalogs, 39
 Ccleaner utility, 95–97, 383
 CDs, backup, 77
 Center marks option, 532
 Centerline extension option, 532
 Centerline font option, 532
 Centerlines/Center marks page, 530–532
 centerpoints, arc, 496
 central processing unit (CPU), 22, 72
 central vault, 390
 centralization, 109
 certification program, Customer Portal, 470
 Certified SolidWorks Associate (CSWA), 469
 Certified SolidWorks Professional (CSWP), 470, 474
 change, management of, 347–352
 Change License button, 395, 412
 Change link, Products area, 125
 Check for Updates menu, 357–358
 Check if files opened read-only have been modified by other
 users option, 526
 Check In Document dialog box, SolidWorks Workgroup
 PDM, 458–459
 Check In/Attach Document dialog box, SolidWorks
 Workgroup PDM, 459–460
 Check out-of-date lightweight components setting, 506–507
 Check read-only files command, 340
 Choose 1 custom property to display option, Folders tab, 456
 circuit board design software, 6
 CircuitWorks parts, 245
 clarity, 281
 Clear button, vault log, 415
 click-click sketching, 499
 click-drag sketching, 499
 Client application, 392
 Client Install Options, 366

Index

- Client Options tab, Option Editor, 132–133
- cloud storage, 32, 33
- cmd utility, 92
- CNC. *See* Computer Numerical Control
- coincident mates, 236
- collaboration
 - Lightweight page, 344–346
 - overview, 337
 - process, 342–344
 - Selective Open option, 344
 - sharing data, 338–342
 - SolidWorks Workgroup PDM, 460
 - summary, 346
 - Workgroup PDM, 389
- Collaboration page, System Options tab, 172–173, 339–342, 525–526
- Color scheme settings option, 494
- Colors page
 - Document Properties tab, 542–543
 - System Options tab, 163–164, 494–496
- Command Defaults tab, SolidWorks Workgroup PDM, 452–454
- command line, installing Administrative Image from, 138
- Command Prompt, Windows, 92
- comma-separated value (CSV) file reader, 442
- company-specific training, 15
- Compare vault document option, Vault tab, 452
- comp.cad.solidworks forum, 314
- computer hygiene, 115
- Computer Management dialog box, Windows
 - basic input/output system, 83–84
 - Device Manager, 74
 - Disk Management section, 74–75
 - Event Viewer, 69
 - Local Users and Groups section, 70–71
 - Performance Monitor, 72
 - Reliability Monitor, 72–74
 - Services and Applications area, 75–76
 - Shared Folders section, 70
 - Task Scheduler, 64–68
 - Windows Backup, 76–83
- Computer Numerical Control (CNC), 254, 377
- computers
 - Apple, 19
 - backing up, 80
 - CAD requirements, 19
 - hard drives, 23–25
 - hygiene, 115
 - laptops, 25–29
 - memory, 20–21
 - monitors, 25
 - operating systems, 20–21
 - processors, 20
 - risks of building, 18
 - video cards, 21–23
- Concatenate command, Excel, 226
- concatenated revision scheme, 416
- concentric mates, 236–237
- concept designers/engineers, 9
- Concept lifecycle status, 392, 433–434
- concept model, 283
- confidentiality, 111, 123
- configuration properties, part templates, 190
- Configuration Properties PropertyManager, 190
- configurations
 - combining from multiple parts, 217
 - creating, 175, 226–227
 - handling in Workgroup PDM, 437
 - SolidWorks Workgroup PDM, 437
 - user settings, 227–228
 - using to store revisions, 61–62
 - using with custom properties, 277
- configurator, defined, 209
- Configure Toolbox utility, 221
- Confirm Attribute Changes dialog box, 215
- construction line, 165
- consultants, 105–106, 323–324
- contiguous space, 394
- Control Panel, 76, 142
- Conversion Wizard, 374
- Convert Entities tool, 170, 286
- Convert Files task, Task Scheduler, 100–101
- Coordinator computer, 99
- Copy Options Wizard, 133
- Copy Parts option, Toolbox, 216, 423
- Copy Project Permissions option, 435
- Copy Settings Wizard, 85, 141, 171, 366
- corner coordinate data, macro, 307
- corporate computer user rights, 71
- corrupted installation, 379
- cosmetic fillet, 285
- Cosmetic Threads, 261, 536–537
- courtesy rules, online forums, 110
- CPU (central processing unit), 22, 72
- crashes, 95
- Create Parts setting, Toolbox, 438
- Create PDF files in the Vault on normal check in of SolidWorks drawings setting, 414

- Create Sketch on New Part option, System Options tab, 498
 - Create Spline button, 311
 - credibility, standards, 481
 - CSV (comma-separated value) file reader, 442
 - CSWA (Certified SolidWorks Associate), 469
 - CSWP (Certified SolidWorks Professional), 470, 474
 - Current color scheme setting option, 494
 - Current_User section, SolidWorks registry, 381
 - Curvature generation setting, 506
 - curvature window, Document Properties tab, 542
 - cust_prop_defs.pdmw item, 412
 - custom properties
 - in Bills of Material, 266–269
 - forms, 276
 - migrating from one document to another, 180
 - overview, 265–266
 - part templates, 187
 - revisions and, 60
 - tabs, 274–275
 - in Title Block, 269–274
 - using configurations with, 277
 - Custom Properties icon, Design Library control interface, 235
 - Custom Property Definition dialog box, 223–224
 - Custom Property Tab Builder interface, 375
 - Custom Property tab, groupbox, 276
 - Custom property used as component description option, 488
 - Custom property used as Revision option, 492
 - Custom Setup screen, Workgroup PDM Server, 404
 - custom training, 15
 - Customer Experience Program option, 135, 363
 - Customer Portal
 - 3D Content Central, 248
 - accessing support database, 317–318
 - download installation files from, 118
 - getting information from, 318
 - managing assets, 149
 - online help resources, 467–470
 - Release Notes, 347, 353, 355
 - Customize interface, 309
 - Customize Macro Button dialog box, 310
 - Cutter number, 40–41
 - Cyganik, Ed, 258
 - cylindrical faces, 236–237
- ## D
- Daily options, Create Basic Task Wizard, 67
 - Dash option, Document Properties tab, 539
 - Dassault Systemmes, 387, 432
 - data
 - corruption, 326–327
 - entering in Title Block, 273
 - that can be linked to specific file locations, 172
 - types that can be automatically inserted into custom properties, 269
 - Data eXchange Format (DXF) format, 284
 - Data folder, 215
 - database, Toolbox, 210
 - DDS (Dewey Decimal System), 40–41
 - deactivating licenses, 150, 379
 - Debug button, macros, 308
 - default colors, 164
 - default libraries, 233
 - Default template paths setting, 512
 - Default Templates page, System Options tab, 512–513
 - defining standards, 254–255
 - defragmenting, 98
 - Delete, Rollback, Archive documents panel, VaultAdmin tool, 415
 - Delete Bodies option, 257
 - Delete local copies option, Command Defaults tab, 452
 - deleting, versus editing, 290
 - Dell brand, 19
 - Delphi Horizontal Modeling, 286
 - Describe Problem button, 329
 - description box, Create Basic Task Wizard, 66
 - Description custom property, System Options tab, 488
 - Description field, Toolbox Setup interface, 225
 - descriptive filenames, 45–47
 - design, 9, 280
 - Design Checker tool, 262
 - Design Clipart, 247
 - Design Library icon, Design Library control interface, 235
 - Design Library tab, Task Pane, 230
 - Design World, 475–476
 - Detach Segment On Drag setting, 327
 - detachable PropertyManager tool, 25, 515
 - detachable Task Pane, 25
 - detached drawings functionality, 51
 - detail drafters/modelers, 9
 - Detail view scaling option, System Options tab, 492
 - detailing options, 191, 195
 - Detailing page
 - Document Properties tab, 195, 534–538
 - System Options, 184
 - Deutsches Institut für Normung (DIN), 244, 479
 - Development lifecycle status, 392, 433
 - Deviation setting, Document Properties tab, 544

Index

- Device Manager, Windows, 74
- Devon Sowell blog, 471
- Dewey Decimal System (DDS), 40–41
- Dezignstuff blog, 471
- diagnostic logging, 134
- diagnostic startup, 87
- Diagnostics tab, SolidWorks Rx, 330–331
- Digital Visual Interface (DVI) connector, 28
- Dim statement, 307
- Dimension Modify spin box, 517
- dimensions, macro, 307
- Dimensions page, Document Properties tab, 194, 530
- DimXpert page, Document Properties tab, 532–533
- DIN (Deutsches Institut für Normung), 244, 479
- Direct 3D, 22
- Disable additional weldment configuration description strings option, 489
- Disable note/dimension inference option, 490
- Discussion Forums area, Customer Portal, 469
- disk checking, 98
- Disk Cleanup utility, Windows, 96–97
- Disk Management section, Windows, 74–75
- Disk Operating System (DOS) commands, 65
- disk space warnings, 128
- display
 - building computers, 25
 - laptops, 28
- Display arc centerpoints in part/assembly sketches setting, 496
- Display dimensions flat to screen option, 503
- Display entity points in part/assembly sketches setting, 496–497
- Display Filter area, 191
- Display filter section, Document Properties tab, 536
- Display grid option, Document Properties tab, 539
- Display items only in the view in which they are created option, Document Properties tab, 536
- Display new detail circles as circles option, 490
- Display notes flat to screen option, 503
- Display plane when shaded setting, 498
- Display quality for new views setting, 492
- Display reference triad setting, 503
- Display referenced files, 442
- Display scrollbars in graphics view option, 503
- Display Shaded Planes setting, 186, 503
- Display sketch arc centerpoints setting, 490
- Display sketch entity points setting, 491
- Display style for new views setting, 492
- Display Style page, System Options tab, 492–493
- Display virtual sharps setting, 498
- Display warnings setting, System Options tab, 516
- Display/Selection page, System Options tab, 501–504
- Dissection panel, System Options tab, 247–249, 525
- Dissolve Subassembly option, 293
- *.dll files, 211
- Do not allow a document to return to the selected or a prior status option, 434
- Do Not Create References External to the Model, 169–170
- Do not create references external to the model option, 512
- Do not display edges in shaded mode setting, 509
- Do not save auto recover info option, 509
- Document Information window, SolidWorks add-in client, 448–449
- document management
 - file organization, 47–55
 - naming conventions, 38–47
 - overview, 37
 - revisions, 55–62
- document permissions, 429
- Document Properties tab
 - Centerlines/Center marks page, 530–532
 - Colors page, 542–543
 - colors section, 163
 - Detailing page, 534–538
 - DimXpert page, 532–533
 - Drafting Standard page, 181–182, 191, 193–194, 527–530
 - Grid/Snap page, 182, 538–539
 - Image Quality page, 543–545
 - Line Font page, 196, 540–541
 - Line Style page, 196–197, 541
 - Line Thickness page, 197, 541–542
 - overview, 527
 - Plane Display page, 186–187, 545–546
 - Sheet Metal page, 198, 545–546
 - Tables page, 532–533
 - Units page, 183, 195, 539–540
 - View Labels page, 534
 - Virtual Sharps page, 534–535
- Document ranges in vault tree option, Vault tab, 452
- document relationships, 51–52
- Dome feature, 289
- dongles, 122, 145, 154
- Don't allow read-only users to open docs in the SolidWorks Workgroup PDM Viewer setting, 413
- Don't ask me again option, System Options tab, 486
- Don't Check option, Check out-of-date lightweight components setting, 506

Don't prompt to save read-only referenced documents
 setting, 167, 341, 509
 DOS (Disk Operating System) commands, 65
 Download Files option, 359
 Download link, SolidWorks, 119
 download monitor, service pack, 360–361
 Download Packages page, 121, 403
 Download section, Customer Portal, 358
 downloading SolidWorks
 manually installing, 119–121
 overview, 117–119
 SolidWorks Installation Manager, 121–128
 downloads, service pack, 357–363
 Downloads and Updates area, Customer Portal, 468
 Downloads page, Customer Portal, 402
 drafting department, traditional, 3
 Drafting Standard option, ISO, 179
 Drafting Standard page, Document Properties tab
 2D standards, 255
 Annotations page, 528–530
 assembly templates, 191
 Dimensions page, 530
 drawing templates, 193–194
 overview, 527–528
 part templates, 181–182
 drafting standards. *See* standards
 Drag and drop in tree option, Local tab, 449
 drawing
 2D, 3, 7, 58
 changing orientation of, 199
 standardizing format of, 162
 syntax for linking properties to, 271
 drawing ratio, 7–8
 drawing templates, 192–199
 Drawings page, System Options tab, 489–493
 driver maintenance, 95
 drop-down lists, Workgroup PDM, 421
 drop-in replacement, 55
 *.drwdot extension, 178
 dual dimension unit, 183
 dual quad-core processors, 19, 20
 dual units, 195
 dual-role, CAD Administrator, 8
 DVDs
 in-house training, 107
 installing service packs, 357–363
 installing SolidWorks from, 117, 128–129
 DVI (Digital Visual Interface) connector, 28
 DWG import, 200

DWGeditor, 201
 DXF (Data eXchange Format) format, 284
 DXF/DWG format, 58, 197
 dynamic assembly motion, 295
 Dynamic highlight from graphics view option, 503
 Dynamic highlight setting, 515
 dynamic libraries, 212, 438

E

Early Visibility (EV) testing, 356
 ease-of-use concept, 279
 ECO (engineering change order), 420
 Edge display in shaded with edges mode setting, 502
 Edit Macro tool, 311
 Edit Sheet Format option, 270
 Edit Sheet mode, 273
 Edit Sheet Properties box, 202–203
 eDrawing style viewer, 447
 efficiency, administration, 13–15
 electronic documentation, 392–393
 electronic file, 43
 electronic forms, 351–352
 electronic training material, 107
 Eliminate duplicate model dimensions on insert option, 489
 ellipses, 282
 e-mail, 67, 333
 Enable Confirmation Corner option, 487
 Enable FeatureXpert option, 488
 Enable multi-user environment option, 339, 526
 Enable on screen numeric input on entity creation option, 499
 Enable performance email setting, 414
 Enable performance feedback option, 489
 Enable selection through transparency setting, 503
 Enable snapping option, 500
 Enable spline tangency and curvature handles setting, 498
 End User License Agreement (EULA), 119, 150, 155, 359, 403
 endpoints, 496–497
 enforcing standards, 261–263
 engineering change order (ECO), 420
 Engineering Exchange, 474
 Engineering lifecycle status, 392, 433
 Engineering Manager, 4
 eng-tips forum, 314, 334
 enhancement request, 322
 Enterprise PDM, 374, 388, 432
 Enterprise Resource Planning (ERP) system, 5, 42, 227, 265
 entity points, 496–497
 ergonomics, workspace, 36

Index

ERP (Enterprise Resource Planning) system, 5, 42, 227, 265
“ERROR!:rev” note, 272
errors
 Smart Fasteners, 229
 Toolbox database, 218
EULA. *See* End User License Agreement
EV (Early Visibility) testing, 356
Event Log, 69
Event Viewer, Windows, 69
Every startup option, VaultAdmin tool, 414
Excel Concatenate command, 226
expanded interface, SolidWorks Explorer, 53
expert users, becoming, 11
Explain Access option, RMB menu, 462
Explain Document Access dialog box, Workgroup PDM, 460–461
Extended lines option, Document Properties tab, 532
extensions, format, 178
external hard drive, 31, 77
External References page
 accessing, 339
 accessing additional settings from, 341–342
 Allow Multiple Contexts setting, 167
 Automatically Generate Names for Referenced Geometry setting, 169
 Do Not Create References External to the Model, 169–170
 Don’t Prompt to Save Read-Only Referenced Documents setting, 167
 in-context modeling, 297–299
 Open Referenced Documents with Read-Only Access setting, 166
 overview, 509–512
 Search File Locations for Referenced Documents setting, 169
 Update Component Names when Documents are Replaced setting, 169
 Update Out-of-date Linked Design Tables option, 169
Extrude PropertyManager, 189

F

Face Property options, 169
Facebook, 106, 323, 474
faces, 236–237, 290
face-to-face mate, 294–295
FCS (First Customer Ship), 354
FEA (Finite Element Analysis), 4, 20
Feature Statistics tool, 291

FeatureManager
 checking parts for errors or warnings in, 356
 errors in, 175
 of Library feature file, 243
 Lightweight assembly, 345–346
 Show Update Holders option, 298
 T-bolt, 237
 tooltips on components in, 340
FeatureManager page, System Options tab, 514–517
features
 best practices for, 288–291
 versus sketches, 285
File Explorer functionality, SolidWorks window, 55
File Explorer icon, Design Library control interface, 235
File Explorer page, System Options tab, 521–523
file locations, specifying, 171–172
File Locations option, References/Where Used tab, 450
File Locations page, System Options tab, 513–514
File Locations settings, 171, 203
file management, 7–8, 32, 299–300
File menu, 339, 341
file organization
 document relationships, 51–52
 libraries, creating, 48–49
 overview, 47–48
 references, breaking, 50–51
 SolidWorks Explorer, 53–55
 subassemblies, 52
 Windows Search, 53–55
file permissions, 435
file settings, 172
file storage, networking, 31
file transfer protocol (FTP), 33, 333
filenames, 44–47, 393–394, 60
Files & Logs tab, SolidWorks Rx, 328, 332–333
fillets, 175, 285
Fillister Head Screw, 226–227
Find and include associated option, Workgroup PDM, 459
Finite Element Analysis (FEA), 4, 20
FireGL card, 22, 26
First Angle projection, 179, 504
First Customer Ship (FCS), 354
fixed plane, 294
Fixed SPR list, 320–321
Flash movies, 108
flat faces, 236
flat files, 47, 59, 389
flat pattern sketch color, 198

FLEXlm application, 121, 154
 FLEXlm End Users Guide, 154
 floating license, 121, 130, 140, 153–156
 flyout FeatureManager, 515–516
 folder names, 61–62
 folder permissions, 429
 Folders tab, SolidWorks Workgroup PDM, 454–455
 font manager, Windows, 195
 font settings, drawing templates, 194–195
 Force user login setting, 413
 Form New Subassembly Here option, 293
 formats. *See* sheet formats
 forms
 custom properties, 276
 electronic, 351–352
 forums, online, 110–111, 322–323
 Forums area, Customer Portal, 469
 fractions, 540
 Frame style, annotation, 529–530
 free plane, 294
 Freeform feature, 289
 Front face, plane, 186
 FTP (file transfer protocol), 33, 333
 future version error, 378

G

GD&T (geometric dimensioning and tolerancing), 256, 481
 Gear mate, 296
 GeForce card, 26
 General page, System Options tab, 484–489
 General Services Administration (GSA), 478
 General tab, System Configuration dialog box, 86–87
 Generate Report button, search dialog box, 457, 462
 geometric dimensioning and tolerancing (GD&T), 256, 481
 geometry pattern, 290
 Get License link, My Products page, 152
 Get Write Access option, 339–340
 Ghost image on drag setting, System Options tab, 499
 global serial number, 132
 Global settings panel, VaultAdmin tool, 412–414
 global variables, part templates, 188–189
 globals.pdmw item, 411
 Google Search, 109, 334
 Google Updater, 69
 gpedit.msc. command, 92
 GPU (graphics processing unit), 22
 graphics cards, 22–23, 506

graphics processing unit (GPU), 22
 Grid/Snap page, Document Properties tab, 182, 538–539
 gross file management, 212
 groupbox, 276
 groups, 428
 GSA (General Services Administration), 478

H

“hack and whack” type modeling, 283
 hard drives, 23–25, 28, 394
 hard-coded limitation, SolidWorks, 7
 hardware
 buying, 17
 conflicts, 333
 Device Manager listing, 74
 failures, 327
 troubleshooting, 74
 help resources, online
 about macros, 313–314
 blogs, 470–473
 overview, 467
 social media, 323, 473–476
 SolidWorks Customer Portal, 467–470
 Hemmingway, Ernest, 257
 Hidden edges displayed as setting, 502
 Hide all planes, axes, sketches, curves, annotations, etc.
 option, 509
 Hide dangling dimensions and annotations option,
 Document Properties tab, 184, 537–538
 Hide Projects with no access setting, 413
 Hide/show component option, 519
 Hide/show tree items, 517
 High quality for dynamic view mode option, System Options
 tab, 506
 High quality for normal view mode option, System Options
 tab, 506
 Highlight all edges of features selected in graphics view
 option, System Options tab, 503
 Hinge mate, 296
 HKEY_CURRENT_USER (HKCU) key, 85, 143
 HKEY_CURRENT_USER\Software\SolidWorks key, 143
 HKEY_LOCAL_MACHINE (HKLM) key, 85, 143
 Hole Tables, 203
 Hole Wizard, 62, 257, 261
 Hole Wizard/Toolbox page, System Options tab, 521
 Home tab, SolidWorks Rx, 329
 home use licenses, 155–156

Index

Hoops graphics layer, 23
Horizontal Modeling, 174, 286–288
horizontal relation, 500–501
hotkey, 309–310
HP brand, 19
hybrid intelligent-sequential systems, 43–44
hyperthreading, 72

I

“if-then” rules, 281
Ignore feature color option, Document Properties tab, 543
Ignore self-intersection check for some sheet metal features
 option, System Options tab, 506
IIS. *See* Internet Information Services
Image Quality page, Document Properties tab, 185, 543–545
image quality settings
 assembly templates, 192
 drawing templates, 198
 part templates, 185
imaging software, 33
Import annotations option, Document Properties tab, 537
Import wizard, 201
Include 3D ContentCentral results option, 524
Include COSMOS (Simulation) results option, Command
 Defaults tab, 453
Include drawings option, Command Defaults tab, 453–454
inclusion, feeling of, 15
in-context modeling, 176, 293, 297–300
in-context references, 167
Indexing Performance panel, System Options tab, 525
Indexing Service, 415
Indicate option, Check out-of-date lightweight components
 setting, 506
individual install, service pack, 359
individualism, CAD, 281
Information Technology. *See* IT
in-house training, 106–108
input, team, 15
Input dimension value option, System Options tab, 484
Insert Bends method, 257
Installation location options, 127
installation monitor, service pack, 362
installation phase, 129
installation Summary information, 368
installing
 floating licenses, 154–155
 service packs, 357–368
 SolidWorks, 115–143, 378
 SolidWorks Workgroup PDM, 394–395
 SolidWorks Workgroup PDM vault, 402–411
 SolidWorks Workgroup PDM VaultAdmin tool, 402–411
 Toolbox, 209–218
Intel (Graphics Media Accelerator) graphics, 26
intelligent part numbering, 38–41, 394
internal hard drive, 78
internal testing, 354
International Standardization Organization (ISO)
 Drafting Standard option, 179
 Web site, 480
Internet Information Services (IIS)
 turning on in Windows Features dialog box, 406
 vault, 390
 Workgroup PDM Viewer, 395, 405
Internet Options settings, Windows, 91
Internet Protocol (IP) Configuration tool, Windows, 91
Interrupt ReQuest (IRQ) conflicts, 18, 83
intersection lines, plane, 187
Inventor modeler, 22, 47
IP (Internet Protocol) Configuration tool, Windows, 91
IrfanView application, 495
IRQ (Interrupt ReQuest) conflicts, 18, 83
IsFastener property, 424, 439–440
ISO. *See* International Standardization Organization
Isolate setting, 519
IsToolbox property, 424, 439–440
IT (Information Technology)
 antivirus applications, 33–35
 backups, 31–33
 and CAD administration, 9–10
 computers, 18–29
 grouping CAD users with other technology users, 6
 networking, 29–31
 overview, 17
 professionals, 3
 workspace, 35–36

J

Japanese Industrial Standards (JIS), 480
Jefferson, Thomas, 257
JIS (Japanese Industrial Standards), 480
journal files, macros, 305–307

K

Kaspersky AV application, 97
Keep extra copy of latest files setting, 413

Keyboard movement increment setting, 492

keywords, SolidWorks Search, 248–249

Knowledge Base

- Customer Portal and, 468

- links on Troubleshoot tab, 330

- searching, 320–321

L

Labels tab, SolidWorks Workgroup PDM, 455–456

LAN (local area network), 30–31, 78

laptops

- display, 28

- hard drives, 28

- operating systems, 27

- power considerations, 26–27

- processors, 25–26

- RAM, 27

- solid-state drives, 23

- touch screens, 28–29

- video cards, 26

- wattage, 26

Large Assemblies mode, 159–160

Last revision type optional check box option, 416

Latest revision option, revisions, 417

Layer option, Document Properties tab, 530

layering, 7, 201, 477

layout sketches, 286–288, 291–292

ldadminoptioneditor.exe file, 131

le_mapping.pdmw item, 411

Leader display option, Document Properties tab, 530

leadership, 12–13

legacy macro, 301

Lenovo brand, 19

Level of detail setting, 506

libraries

- 3D Content Central, 244–245

- custom properties and tags, 240–241

- defined, 208

- Dissection feature, 247–249

- including metadata in, 240

- Library Features, 241–244

- overview, 48–49, 233–234

- parts, 234–240

- Routing, 245–246

- third-party, 232

- Weldments, 246–247

Library Components folder, 442

Library Features, 241–244

library parts, Toolbox, 210

license management software, 153–154

license server, 153

licenses

- activating, 133–134, 150–153, 380

- cost of, 14

- deactivating, 379

- floating, 153–155

- home use license policy, 155–156

- overview, 145

- serial numbers, 148–149

- versions of SolidWorks, selecting, 146–148

Lifecycle settings, SolidWorks Workgroup PDM, 433–435, 460–461

lifecycle status, Workgroup PDM, 392

Lifecycle tab, VaultAdmin tool, 419–420, 427

lightweight functionality, 51

Lightweight option, SolidWorks 2009 Open dialog box, 161

Lightweight page, 344–346

Line Font page, Document Properties tab, 196, 540–541

Line length measured between virtual sharps in 3D setting, 498

Line Style page, Document Properties tab, 196–197, 541

Line Thickness page, Document Properties tab, 197, 541–542

line weights, 7

Linear Coupler mate, 296

linetypes, 379

Link to thickness option, 188–189

link values, part templates, 188–189

Linked-In site, 334, 475

Linux operating system, 395

List External References dialog box, 170, 298

live user group, 111

Load referenced documents option, 510

local area network (LAN), 30–31, 78

local properties, 272

Local tab, SolidWorks Workgroup PDM, 449–450

Local Toolbox, 214

Local Users and Groups section, Windows, 70–71

Local View, SolidWorks add-in client, 447

Local_Machine section, SolidWorks registry, 381

Lock references, 170

LockVault button, VaultAdmin tool, 412

Lofted Bends functionality, 148

logging, diagnostic, 134

login screen, VaultAdmin tool, 410

longevity, 283

Look for option, References/Where Used tab, 450

“Lunch and Learn” session, 108

M

- Mac operating system, 395
- Macro Editor, 304
- Macro toolbar, 302
- macros
 - creating, 301, 310–311
 - help resources, 313–314
 - journal files, 305–307
 - recording, 302–304
 - rectangle sketching, 302–303
 - replaying, 307–310
- Make Read-Only menu option, 339–340
- malicious software, 32
- Manage Installs page, 119, 403
- management
 - getting support from, 350
 - versus leadership, 12
- manual data entry, 225
- manual file management, 57, 338
- manual file storage technique, 31
- Manual Install page, 119–120
- manual installation, service packs, 356–357
- manual revision method, 61
- manufacturer warranty, 18
- Manufacturing lifecycle status, 392, 433
- Manufacturing Resource Planning (MRP) system,
 - 5, 42, 227, 265
- mapped drives, 218
- Mark all part/assembly dimensions for import into drawings
 - by default option, 490
- Mark Kiker CADDManager blog, 473
- Mate animation speed setting, 507
- Mate Reference PropertyManager, 238
- Mate References, 211, 231
- mated mechanism, 296
- material settings, 191
- mating
 - analyzing degrees of freedom, 294–295
 - for motion, 295–296
 - for position, 294
 - standards suggestions, 296–297
- Maximize document on open option, 486
- Maximum results per data source setting, 525
- Mechanical mate, 296
- memory, 18, 20–21
- mentoring
 - efficiency, 13–15
 - leaders versus managers, 12
 - overview, 11
 - power users, 108–109
 - team building, 15
 - through leadership, 12–13
- metadata
 - defined, 390
 - including in libraries, 240
 - naming conventions, 45
 - putting revision levels in, 60
 - tags, 265
 - Workgroup PDM, 391
- Microsoft Active Directory, 138–139
- Microsoft Message Queueing (MSMQ) service, 424
- Microsoft Movie Maker, 107
- Microsoft Notepad, 305
- Microsoft Update, 98
- mirroring, 174, 290
- Missing Toolbox size warning message, 217
- mixed units, 39, 191
- modeling
 - assemblies and, 52
 - assembly, 176
 - best practices, 173–175
 - defined, 9
 - productivity, 173
 - syntax for linking properties to model, 271
- Modify dialog box
 - spin wheel, 518
 - System Options tab, 485
- Modify Sketch tool, 202
- Modify User Information dialog box, 428–429
- Moment of Inspiration, 373
- monitors, 25, 28
- moonlighting, 155
- More Properties button, 275
- mouse gesture wheel, 28–29
- Mouse speed setting, 519
- Move components by dragging option, 508
- Move Face feature, 255, 291
- Move option, 202
- Move Sketch tool, 202
- Movie Maker, Microsoft, 107
- Movie Picture Experts Group 4 (mp4), 108
- MRP (Manufacturing Resource Planning) system,
 - 5, 42, 227, 265
- msconfig command, 86–87, 92
- MSMQ (Microsoft Message Queueing) service, 424
- multi-core processors, 20
- multiple assemblies, 292

multiple contexts, 168
 multiple installs, automating, 129
 multiple monitors, 25
 multi-threaded document, 72
 multi-touch screen, laptops, 28–29
 multi-user environment, 29
 My Products link, Customer Portal, 149, 152
 My Products section, Customer Portal, 468
 My SolidWorks Workgroup PDM Documents option, Folders tab, 454–455
 My SPRs section, Customer Portal, 318–319, 468
 My SRs section, Customer Portal, 318–319, 468

N

Name feature on creation setting, 514
 naming conventions
 filenames, 44–47
 hybrid intelligent-sequential systems, 43–44
 intelligent part numbering, 38–41
 sequential part numbering, 42
 NAS (network-attached storage) device, 78
 National CAD Standard (NCS) organization, 256
 National Institute of Standards and Testing (NIST), 480
 NCS (National CAD Standard) organization, 256
 NDA (non-disclosure agreement), 374
 network drive, 77
 network folders, 30
 network license, 121–122, 130, 145, 153
 Network Monitor, SolidWorks, 98–101
 network-attached storage (NAS) device, 78
 networking, 29–31
 New Document interface, 178
 Next button, Create Basic Task Wizard, 66
 NIST (National Institute of Standards and Testing), 480
 No preview during open setting, 508
 Node computer, 98–100
 non-disclosure agreement (NDA), 374
 non-intelligent number, 42
 Normal license type, 412
 Notepad, 305
 Notes/Balloons selection filter, 202
 Novedge Pulse site, 473
 Novell operating system, 395
 Novice interface, New Document interface, 178
 NT File System (NTFS), 78
 Number of backup copies per document setting, 520
 numerical revision scheme, 57
 nVidia Quadro card, 21, 23

O

Obsolete lifecycle status, 392, 433
 office environment, quality of, 35
 off-the-shelf training, 15
 OnEdge sketch relation, 290
 one-to-one eligibility, stand-alone licenses, 155
 online forums, 110–111, 322–323
 online help resources
 blogs, 470–473
 overview, 467
 social media, 473–476
 SolidWorks Customer Portal, 467–470
 online training, 107
 Open dialog box, 161, 343, 345
 Open DWG/DXF documents option, Local tab, 450
 Open last used document at startup option, 484
 Open Read-only option, 343
 Open Referenced Documents with Read-Only Access setting, 166, 341, 509
 Open Selected button, 344
 OpenGL card, 21–22
 operating systems
 3 Gig Switch tool, 93–94
 building computers, 20–21
 Computer Management dialog box, 64–84
 laptops, 27
 overview, 63
 prerequisite to installing SolidWorks, 116
 SolidWorks Network Monitor, 98–101
 System Configuration dialog box, 86–93
 system maintenance, 95–98
 Windows Registry, 84–86
 operator error, 325
 Optimize edge length option, Document Properties tab, 545
 Option Editor Administrative Options tab, 134
 Option Editor interface, Administrative Image, 131
 Option Editor Software to Install tab, 135
 Option Editor Toolbox Options tab, 136
 Options dialog box, SolidWorks Workgroup PDM, 449–456
 Orient to sheet option, Document Properties tab, 532
 Orient to slot option, Document Properties tab, 532
 orientation of drawing, changing, 199
 O-Ring Grooves, 244
 Output file defaults option, Vault tab, 452
 outside trainers, 105–106
 Override Dimensions on Drag option, 285
 Override dimensions on drag/move option, 498
 Override Dims On Drag option, 189, 327

Index

Override quantity column name on Bill Of Materials
option, 492
ownership concept, PDM, 392

P

packaged license, 123
“Pack-and-Go” assembly, 231
page files, 21
Palette Features, 244
parabolas, 282
parallel installation, 116, 355, 378
Parallels option, 19
parametric modeling, 287
parent/child model, Horizontal Modeling, 287
part documents, dual dimensioning, 184
part files, 51
Part Number, Toolbox Setup interface, 225
part templates
color settings, 185
configuration properties, 190
custom properties, 187
detailing settings, 184
Drafting Standard, 181–182
global variables, 188–189
Grid/Snap page, selecting settings in, 182
image quality settings, 185
link values, 188–189
overview, 180
plane display parameters, 186–187
reference geometry, 189
sketch settings, 189–190
units settings, 183
Part/Assembly tangent edge display setting, 502
parts, mating, 257
parts library
adding to Design Library, 234–236
Mate References, 236–240
Toolbox, 211
Password option, Vault tab, 452
patterning, 174, 290
PDF (Portable Document File) documents, 58, 197
PDM (Product Data Management)
administrator role and, 5
goal of, 30
implementing, 350
libraries and, 234
metadata in, 46–47
multiple files, 44
revision levels, 60
sequential part numbers, 42
sharing data, 338
storing and retrieving revisions, 57–58
PDM/PLM application, 265
pdmrc folder, 411
*.pdmw file, 389
pdmwadmin user name, VaultAdmin tool, 410
PDMWorks Web Portal, 406. *See also* Workgroup PDM Viewer
PDMWorks Workgroup Server, 398
pdmwserver.exe file, 397
PEBCAK (problem exists between chair and keyboard) error,
91, 325
pen tablet, 25
Pending lifecycle status, 392, 433–434
people skills, 109
Performance Monitor, Windows, 72, 92
Performance page, System Options tab, 165–166, 505–508
per-license penalty, subscriptions, 372
permissions
adding to Workgroup PDM, 420
establishing, 427–436
Toolbox, 228–229
Permissions dialog box, Toolbox Setup Wizard, 228–229
personalizing workspace, 36
PhotoView 360, 20
PhotoWorks, 303
Pierce sketch relation, 290
Plane Display page, Document Properties tab, 186–187,
545–546
planes
mating to, 294
sketching on, 290
plane-to-plane mate, 294
PLM (Product Lifecycle Management), 5, 432
point release, 353–354
Portable Document File (PDF) documents, 58, 197
power users, 12–13, 108–109
PowerPoint presentation, 107, 334
PPM (Product Process Management), 432
Precisely render overlapping geometry setting, 198
predefined view, empty, 270
prefixes, part number, 39
Preview Window, SolidWorks add-in client, 448–449
primary dimension unit, 183
Primary Reference Entity, Mate Reference, 239
Print out-of-sync water mark option, System Options tab, 490
privacy, serial numbers, 123
Problem Capture function, SolidWorks Rx, 329–330, 332

Problem Devices category, 90
 problem exists between chair and keyboard (PEBCAK) error, 91, 325
 process-based modelling work, 8
 processors, 20, 25–26
 Product Activation Wizard interface, 150–151
 Product Data Management. *See* PDM
 Product Lifecycle Management (PLM), 5, 432
 Product options, 126
 Product Process Management (PPM), 432
 Product Selection window, 408
 Production lifecycle status, 392, 434
 production model, 283
 production revisions, 56
 productivity, 173
 Pro/ENGINEER modeler, 6, 47, 164, 279
 professionalism
 mixing personal conversations with professional conversations, 323
 standards, 481
 Profile corruption, 329
 Programs area, Windows, 91
 project folder scheme, 430–433
 Project labels option, Folders tab, 456
 Projection type for four view viewport setting, 504
 Projects tab, VaultAdmin tool, 418–419
 projects.pdmw item, 411
 Prompt before changing mate alignments on edit option, 508
 Prompt to close sketch setting, 497
 Prompt to set driven state setting, 499
 Prompt user to select document template option, 171, 512–513
 Properties tab, VaultAdmin tool, 421–422
 Properties Window, 311–312
 Property labels displayed in vault tree option, Folders tab, 455
 Property Tab Builder, 274–275
 PropertyManagers
 detachable, 25, 488
 dimensions in, 62
 libraries and, 235
 with Link to thickness option, 189
 \$PRP:"property", 270
 \$PRPSHEET:{property}, 270
 .prt/.asm/.drw extensions not associated with SolidWorks setting, 413
 *.prtdot extension, 178, 180
 pseudonym, using for online forums, 111
 public logins, SolidWorks Corp., 406
 purpose-specific mate, 296

Q

quad-core processors, 19
 Quadro card, 22, 26
 Query button, 320

R

Rack and Pinion mate, 296
 Radeon card, 26
 radio buttons, using to create forms, 276
 RAID (Redundant Array of Independent Disks), 24, 28, 70
 Random Access Memory (RAM), 18, 21, 27
 *.rar file, 33
 read-only access user, 338
 Read-only option, Data folder, 215–216
 Rebuild assembly on load setting, 507
 rebuild errors, fixing, 175
 rebuild speed, 285
 rectangle macro, 302–303, 310
 rectangle tool, 282–283
 Redundant Array of Independent Disks (RAID), 24, 28, 70
 reference geometry, 189, 290, 294
 reference triad, 503
 Referenced Documents setting, 171, 460–461
 references
 breaking, 50–51
 relations to stable, 174
 References button, 299
 References list, 422
 References option, 342
 References/Where Used tab, SolidWorks Workgroup PDM, 450–451
 *.reg file, 85
 regedit command, 84, 92
 registry, 139, 329, 381–383
 Registry Editor, 85, 142, 382
 registry keys, renaming, 86
 regression, 327
 reinstalling SolidWorks, 379–383
 Relations/Snaps page, System Options tab, 499–501
 Release Notes, Customer Portal, 347, 353, 355
 reliability, hard drive, 23–24
 Reliability Monitor, Windows, 72–74
 Reload command, 340, 342
 Reload dialog box, 341
 Reload option, 204
 Remember option, Local tab, 450

Index

Remote Assistance, Windows, 91
Rename Document function, RMB menu, 462
rendering software, 20
“replace first, ask questions later” mentality, 325
replaying macros
 connecting to custom toolbar button, 310
 connecting to hotkey, 309–310
 overview, 307–308
 storing, 308–309
Reporting dialog box, Workgroup PDM, 457–458
reproducibility, sketch, 284
resellers
 basic class, 105
 cost of, 103
 external support, 316–317
 in-house training, 106
 overview, 14–15
Reset All To Defaults setting, 495
Resolve lightweight components setting, 507
Resource Monitor, 72
restore points, 82
Results per page search option, 524
Retain ownership option, Command Defaults tab, 452
Retrieve references option, Command Defaults tab, 454
return on investment (ROI), 355
Reverse mouse wheel zoom direction setting, 518
Revision Charts, 203
“revision levels in filenames” method, 61
Revision Scheme tab, VaultAdmin tool, 416–417
Revision Table tab, VaultAdmin tool, 417–418
revision_def folder item, 411
revisions
 changing levels, 58–59
 overview, 55–56
 putting levels in filenames, 60
 putting levels in metadata, 60
 retrieving, 61
 schemes for, 56–57
 storing with configurations, 61–62
 tracking, 58
 updating, 59
revolutions per minutes (RPMs), 23
revolves, 288
Rhino, 22, 373
Rib feature, 289
Ricky Jordan blog, 471
right mouse button (RMB) menu, 203, 339, 445, 456
risk-taking, 349
Rob Rodriguez blog, 471–472

Robert Green CAD Manager blog, 473
ROI (return on investment), 355
rollout, 376–377
Root directory, vault, 412
Round To Nearest Fraction box, 539
round-trip test, 326
Routing library, 245–246
RPMs (revolutions per minutes), 23
ruled surface option, drafting, 291
Run dialog box, 86
Run interface commands, 92
Rx Diagnostics tab, SolidWorks Rx, 330

S

SAE (Society of Automotive Engineers), 480
SAE-AMS (Society of Automotive Engineers-Aerospace Material Specification), 480
SAE-MS (Society of Automotive Engineers-Military Specification), 480
SAP (Systeme, Anwendungen, Produkte), 432
sarcasm, online forums, 110
Save As functionality, 50
Save As Scheme setting, 495
Save auto-recover info every setting, 520
Save button, vault log, 415
Save Modified Documents dialog, 511
Save new components to external files option, 509
Save notification setting, 520
Save tessellated data for drawings with shaded and draft quality views setting, 491
Save tessellation setting, 185
Save tessellation with part document option, Document Properties tab, 545
scale, project, 7
scheduling backups, 80
screen shots, online forum, 110
Screw mate, 296
scripting, 65
Scroll selected item into view setting, 514
Search feature, 49–50, 248–249
Search file locations for external references setting, 511
Search File Locations for Referenced Documents setting, 169
Search for Document dialog box, 457
Search functionality, 234
Search Indexing Status dialog box, 525
Search page, System Options tab, 522–525
Search tab, VaultAdmin tool, 415–416
Search while typing (incremental search) option, 524

- Secondary Reference Entity, Mate Reference, 239
- security
 - online forums, 111
 - serial numbers, 123
- Select all references option, Command Defaults tab, 453
- Select hidden entities option, System Options tab, 490
- Selective Open option, 344
- Selective startup, 87
- semi-intelligent part numbering system, 49, 226, 394
- sequential part numbering system, 42, 394
- sequential revision scheme, 416
- serial numbers
 - activation and, 134
 - Administrative Image and, 122
 - Customer Portal, managing assets with, 149
 - identifying types of, 149
 - tracking, 148
- Serial Numbers tab, Option Editor, 131–132
- Server Busy file access error, 35
- Server files, 396
- service packs
 - installing, 117, 128, 356–368
 - overview, 353–354
 - rolling back, 368
 - SolidWorks Workgroup PDM, 396–397
 - updating, 354–356
- service request (SR), 317
- Services and Applications area, Windows, 75–76
- Services section, Computer Management dialog box, 75–76
- Services tab, Windows, 87–88
- Set Document Property utility, 440
- Set driven by default setting, System Options tab, 499
- Set filesystem read-only if not owner setting, 414
- setting problems, 327
- Setup dialog box, Network Monitor, 99
- Setup Type screen, Workgroup PDM Server, 403–404
- Shaded cosmetic threads, Document Properties tab, 536
- shaded quality slider, Image Quality page, 185
- shaded view, 198
- shaft coupler, 238
- Shape feature, 289
- Shared Folders section, Windows, 70
- shared Toolbox installation, 212–213
- sharing data, 338–342
- Shark!, 373
- sheet formats
 - accessing, 271
 - changing, 203–204
 - creating, 200–203
 - defined, 269
 - versus drawing templates, 192–193
 - overview, 177–178
 - placing table anchors in, 203
 - using multiple, 203
- Sheet Metal page, Document Properties tab, 198, 545–546
- Show contents while dragging drawing view option, System Options tab, 490
- Show curvature comb bounding curve option, System Options tab, 499
- Show dimension names option, System Options tab, 485
- Show errors every rebuild option, System Options tab, 485
- Show halo around annotations setting, Detailing page, 195
- Show in File Explorer view panel, 520
- Show in tree option, Local tab, 449
- Show Intersections setting, Plane Display page, 186
- Show latest news feeds in task pane option, System Options tab, 489
- Show open edges of surfaces in different color option, System Options tab, 503
- Show reference geometry names in drawings option, System Options tab, 490
- Show references option, References/Where Used tab, 450
- Show sheet format dialog on add new sheet option, System Options tab, 492
- Show spline control polygon by default setting, System Options tab, 499
- Show thumbnail graphics in Windows Explorer setting, System Options tab, 486
- Show Update Holders option, FeatureManager, 298
- silent installation, 138
- simplicity, 175, 281, 388
- Simulation, SolidWorks, 20, 374
- Simulation (Include COSMOS) results option, Command Defaults tab, 453
- Single balloon option, Document Properties tab, 530
- Single command per pick option, System Options tab, 484
- single core processors, 20
- single quote, macro, 307
- single-user Toolbox installation, 212–213
- 64-bit operating system, 21, 93, 116, 402
- Size folder, Weldments library, 246–247
- sketch blocks, 245
- sketch contours, 283–284
- sketch error warning message, System Options tab, 485
- sketch fillets, 175
- sketch geometry, 200
- Sketch page, System Options tab, 164–165, 189–190, 496–501
- sketch relations, 282–285

Index

- Sketch snaps options, System Options tab, 500
- sketching, 282–286, 290
- skills, maintaining, 10
- Skip drawings validation during check in setting, 414
- *.sld* file, 35
- *.sldprt extension, 178
- sldimdist.exe file, 359
- *.sldlfp extension, 242
- *.SLDPRT extension, 51
- *.sldreg file, 85, 133
- sldsetdocprop.exe utility, 424, 437, 438, 440
- Smart Fasteners, 229–230
- SmarTeam application, 59, 258
- SmartMates, 240
- SMTP server, 68
- Snagit tool, 107
- Snap to model geometry setting, System Options tab, 500
- SNL (SolidNetWork License), 121, 130, 140–141, 153
- social media, 323, 334, 473–476
- social security number system, 42
- Society of Automotive Engineers (SAE), 480
- Society of Automotive Engineers-Aerospace Material Specification (SAE-AMS), 480
- Society of Automotive Engineers-Military Specification (SAE-MS), 480
- software
 - CAD, 6
 - circuit board design, 6
 - rendering, 20
 - uninstalling, 94
- software interface gene, 11
- Software OpenGL Mode, 22
- Software OpenGL mode, 26, 329
- software performance requests (SPRs), 317, 468
- Software to Install tab, Option Editor, 135
- Solid Edge modeler, 47
- SolidNetWork License (SNL), 121, 130, 140–141, 153
- SolidSmack blog, 473
- solid-state drives, 23, 28
- SolidWorks
 - versus AutoCAD, 7–8
 - installing, 115–143
 - updating, 354–368
 - upgrading, 371–383
 - versions of, 146–148, 219
- SolidWorks 2009 Bible, 244
- SolidWorks 2010 Bible, 106–107
- SolidWorks Back Office Engine (swBOEngine), 87, 382
- SolidWorks Customer Experience Program, 363
- SolidWorks Customer Portal, 467–470
- \SolidWorks Data\ folder, Toolbox, 211
- SolidWorks Drafting Standard dialog box, 256
- SolidWorks Explorer, 53–55, 446–447
- SolidWorks Forums, 314, 322–323
- SolidWorks Installation Manager (SWIM)
 - estimating download time and size, 360
 - EULA, 359
 - installation modification, 408
 - installing service packs, 356–357
 - installing SolidWorks, 121–128, 130
 - overview, 117
- SolidWorks Office Premium version, 147
- SolidWorks Office Professional version, 147
- SolidWorks Office version, 147
- SolidWorks PDM, 107
- SolidWorks Premium version, 146, 263, 373, 387
- SolidWorks Professional version, 146, 263, 373, 387
- SolidWorks Reference Guide, 258
- SolidWorks Resources icon, 235
- SolidWorks Rx
 - Files & Logs tab, 332–333
 - overview, 328–329
 - Problem Capture function, 329–330
 - Rx Diagnostics tab, 330
 - System Maintenance tab, 332
 - transferring data to SolidWorks, 333
 - Troubleshoot tab, 330–331
- SolidWorks Search, 55, 235, 240
- SolidWorks Simulation, 20, 374
- SolidWorks Standard version, 146, 373
- SolidWorks Surfacing and Complex Shape Modeling Bible, 107
- SolidWorks Task Scheduler, 59, 248
- SolidWorks Workgroup PDM
 - backup plan, 396
 - best practices, 445–462
 - how it works, 390
 - implementing, 392–395
 - libraries, establishing inside, 441–443
 - limitations of, 388–389
 - organizing folders with, 391–392
 - overview, 387, 401
 - permissions, 427–436
 - Toolbox, 436–440
 - vault, 390–391, 396–398
 - VaultAdmin tool, 402–426

- SOW (Statement of Work), 481
- Sowell, Devon, 107
- spaceballs, 334
- SpaceClaim, 373
- specialist consultant, 323–324
- speed, hard drive, 23
- Speed Pak, 51
- Spens, Mike, 107
- Spin Box Increments page, System Options tab, 517–518
- Spin Increment control, 311
- spin wheel, Modify dialog box, 518
- splines, 282, 284
- SPRs (software performance requests), 317, 468
- *.sqy files, 457
- SR (service request), 317
- stability
 - ROI calculation, 355
 - SolidWorks software, 330
- Stacked balloons option, Document Properties tab, 530
- stand-alone license, 122, 130, 140, 145, 154
- Standard folder, Weldments library, 246–247
- Standard Libraries tab, VaultAdmin tool, 422–423, 437–438
- standards
 - American National Standards Institute, 479
 - American Society of Mechanical Engineers, 478–479
 - American Society of Testing and Materials, 479
 - defining, 254–261
 - Deutsches Institut für Normung, 479
 - enforcing, 261–263
 - establishing need for, 481
 - General Services Administration, 478
 - International Organization for Standardization, 480
 - Japanese Industrial Standards, 480
 - National Institute of Standards and Testing, 480
 - overview, 253–254, 477–478
 - Society of Automotive Engineers, 480
- Standards Store section, ANSI Web site, 479
- StartOptionEditor.hta file, 131
- StartSWInstall.hta file, 136–137, 366
- Startup tab, Windows, 87
- Statement of Work (SOW), 481
- static library, 212, 247, 437
- static notes, 270
- status_defs.pdmw item, 412
- “steering committee,” 375
- stepped sketch, 288
- Stop VSTA debugger on macro exit option, System Options tab, 488
- storing
 - macros, 308–309
 - revisions, 61–62
- straight upgrades, 218
- streamlining tasks, 157
- subassemblies, 52, 176, 257
- subscriptions, 14, 316–317, 372
- suffixes, part number, 39
- Summary section, installation workflow, 125
- SupplierSource Web site, 324
- Surfacing feature, 255
- Suspend automatic rebuild setting, 509
- Sustaining lifecycle status, 433
- sw_mapping.pdmw item, 412
- *.swb file, 301
- swBOEngine (SolidWorks Back Office Engine), 87, 382
- SWBrowser.mdb, 211
- SWIM. *See* SolidWorks Installation Manager
- *.swp file, 301
- Symantec Ghost, 33
- symmetry, 174
- System Configuration Boot tab, 89
- System Configuration dialog box, Windows, 86–93
- System Information tool, Windows, 90
- System Maintenance tab, SolidWorks Rx, 332
- System Options tab
 - Advanced page, 526–527
 - Assemblies page, 508–509
 - Backup/Recover page, 519–520
 - Collaboration page, 172–173, 339–342, 525–526
 - Colors page, 163–164, 494–496
 - Default Templates page, 170–171, 512–513
 - Display/Selection page, 501–504
 - Dissection panel, 247–249
 - Drawings page, 489–493
 - External References page, 166–170, 342, 509–512
 - FeatureManager page, 514–517
 - File Explorer page, 521–523
 - File Locations page, 171–172, 513–514
 - General page, 484–489
 - Hole Wizard/Toolbox page, 521
 - overview, 157–162, 483
 - Performance page, 165–166, 505–508
 - Search page, 522–525
 - Sketch page, 164–165, 189–190, 496–501
 - Spin Box Increments page, 517–518
 - View page, 518–519
- System Properties area, Windows, 91
- System Protection, Windows, 81–83

Index

System Requirements area, Customer Portal, 469
System Restore, Windows, 81–83, 91
SystemCheck section, installation workflow, 124
Systeme, Anwendungen, Produkte (SAP), 432

T

table anchors, placing in sheet formats, 203
Tables page, Document Properties tab, 532–533
tabs, custom property, 274–275
Tagger application, 241
Tags function, 46–47, 240–241, 265
talent, encouraging, 13
tangent arcs, 285
Tangent edges in new views setting, System Options tab, 492
Task Manager, Windows, 72–73, 92
Task Pane, 25, 234–235
Task Scheduler, Windows, 59, 64–68, 98, 248, 375
Tasks/Network tab, Network Monitor, 99, 101
T-bolt, 237, 239
tboptions.pdmw item, 412
team building, 15
technical support
 in-house, 324–334, 373
 overview, 315
 subscription package, 14
 through external organization, 316–324
technology manager, 4
templates
 assembly, 191–192
 default, 170–171
 drawing, 192–199
 overview, 177–179
 part, 180–190
 sheet formats, 200–204, 270
 standards for, 158
Temporary directory, 330
Temporary Viewing files folder option, Folders tab, 455
Tertiary Reference Entity, Mate Reference, 239
testing
 beta, 354, 374–375
 internal, 354
 service packs, 356
Text annotation type, Document Properties tab, 529–530
Text option, Document Properties tab, 528
Text scale option, Document Properties tab, 536
text-based metadata, 187
textbox, using to create forms, 276
Thin feature, 289

Third Angle projection, 179, 504
third-party library, 232
30-day courtesy license, 152, 381
32-bit operating system, 21, 93, 116, 402
Thread callout box, 261
threads, 175
threats, security, 32
3 Gig Switch tool, Windows, 93–94
3D CAD, 280–281
3D Content Central, Customer Portal, 248, 468
3D model, 7, 254
3D sketching, 291–292
3D standards, 158, 256–261
3-point rectangle, 283
three-view functionality, 198
thumb drives, 77
TIFF format, 197
time-based trigger options, Create Basic Task Wizard, 66
tire numbering system, 39–40
Title Block, 269–274
Toolbox
 adding user parts to library, 230–231
 alternatives to, 232
 customizing hardware, 223
 data management, 212–214
 file management, 231–232
 installing, 209–218
 overview, 207–208
 parts versus configurations, 226–227
 permissions, 228–229
 selecting hardware, 222–223
 Smart Fasteners, 229–230
 SolidWorks Workgroup PDM, 436–440
 starting, 220–221
 upgrading, 218–219
 user settings, 227
tools, standards for, 158
Tools tab, Windows
 About Windows window, 89–90
 Command Prompt, 92
 Internet Options settings, 91
 Internet Protocol Configuration tool, 91
 overview, 88–89
 Performance Monitor, 92
 Programs area, 91
 Registry Editor, 93
 Remote Assistance, 91
 System Information tool, 90
 System Properties area, 91

- System Restore, 91
- Task Manager, 92
- User Account Control, 92
- Windows Security Center, 91
- top-level folders, 50
- touch screens, 28–29
- tracking revisions, 58
- “Train the Trainer” session, 105, 108
- training
 - company-specific, 15
 - criteria, 104–105
 - custom, 15
 - evaluation, 104–105, 348
 - in-house, 106–108
 - live user groups, 111
 - off-the-shelf, 15
 - online forums, 110–111
 - outside trainers, 105–106
 - overview, 103–104
 - power users, mentoring, 108–109
 - upgrade, 375
- Transfer Licenses option, Help Menu, 142
- transparency, graphics, 502
- Triggers tab, VaultAdmin tool, 424–425
- Troubleshoot tab, SolidWorks Rx, 330–331
- troubleshooting
 - add-in software, 334
 - Computer Management dialog box, 64
 - corruption, 379
 - crashes, 95
 - hardware, 18, 74, 325–327
 - Horizontal Modeling scheme, 287
 - imported data, 326
 - looking past obvious for solutions, 333–334
 - mated mechanisms, 296
 - software, 325–327
 - SolidWorks Rx, 328–333
 - System Information tool, 90
 - user accounts and, 70–71
- TrueType fonts, 195
- 24-character serial number, 124
- Twitter, 106, 323, 350, 473–474
- 2D CAD, 280–281
- 2D drawing, 3, 7, 58
- 2D sketching, 291–292
- 2D standards, 158, 253–254
- 2-point rectangle, 283
- two-sheet template, 203
- Type folder, Weldments library, 246–247

U

- UAC (User Account Control), Windows, 92
- UNC (Universal Naming Convention) path, 218, 233
- underdefined sketch, 284–285
- unexpected version error, 378
- Unigraphics, 174
- uninstalling SolidWorks, 94–95, 141–143, 368, 380–381
- unique filenames, importance of, 51–52
- Units page, Document Properties tab, 539–540
- Universal Joint mate, 296
- Universal Naming Convention (UNC) path, 218, 233
- Universal Serial Bus (USB) flash drive, 31
- Universal Serial Bus (USB) port, 154
- Up To Surface feature end condition, 298
- Update Component Names when Documents are Replaced
 - setting, 169, 511
- Update Custom Properties function, Task Scheduler, 265, 440
- Update Error warning, System Options tab, 486
- Update Holder marker, 298
- Update mass properties while saving document setting, 507
- Update Out-of-date Linked Design Tables option, 169, 511
- updating
 - evaluating need for, 348
 - initiating through SolidWorks Web site, 358
 - revisions, 59
 - selecting what and how to, 359
 - SolidWorks, 354–368
 - SolidWorks Workgroup PDM, 396–397
- upgrading
 - deciding when, 371–383
 - evaluating need for, 348
 - major version, 397
 - straight, 218
 - Toolbox, 218–219
 - training, 375
- Usage History area, CPU, 72
- USB (Universal Serial Bus) flash drive, 31
- USB (Universal Serial Bus) port, 154
- Use an existing Toolbox option, 214–215
- Use Assembly Setting For All Components option, 191
- Use assembly setting for all components option, Document
 - Properties tab, 536
- Use document scene background setting, 164
- Use English language feature and file names setting, 487
- Use English language menus setting, 487
- Use Fully Defined Sketches option, 164–165, 496
- Use Large Assembly Mode option, 509
- Use shaded face highlighting option, 486

Index

- Use shaded preview setting, 507
- Use Software OpenGL setting, 507–508
- Use specified color for drawings paper color (disable image in sheet background) setting, 495
- Use specified color for Shaded With Edges mode setting, 495
- Use Specified colors when editing parts in assemblies setting, 495
- Use system separator for dimensions setting, 486
- Use transparent flyout FeatureManager tree in parts/assemblies setting, 515
- User Account Control (UAC), Windows, 92
- user accounts, 70–71
- user blogs, 470–473
- user form, 310–311
- user group, 15
- User Interface folder, 86
- user rights, 71
- user settings, Toolbox, 227
- User Settings dialog box, 227–228
- Users and Groups tab, VaultAdmin dialog box, 428
- `users.pdmw` item, 412

V

- Validate Vault setting, 397, 414
- vault, SolidWorks Workgroup PDM
 - configuring, 411–426
 - maintenance of, 396–397
 - moving, 397–398
 - overview, 390–391
 - size of, 388
 - VaultAdmin Tool and, 402–410
- Vault information section, VaultAdmin tool, 412
- Vault Log panel, VaultAdmin tool, 415
- Vault server application, 392
- Vault Settings tab, VaultAdmin tool
 - Administrators panel, 414
 - default settings, 411
 - Delete, Rollback, Archive documents panel, 415
 - Global settings panel, 412–414
 - LockVault button, 412
 - Validate vault and Every startup options, 414
 - Vault information section, 412
 - Vault Log panel, 415
 - Viewing with eDrawings panel, 415
- Vault tab, SolidWorks Workgroup PDM, 451–452
- vault validation option, 432
- VaultAdmin tool
 - installing, 402–411
 - Lifecycle tab, 419–420
 - Projects tab, 418–419
 - Properties tab, 421–422
 - Revision Scheme tab, 416–417
 - Revision Table tab, 417–418
 - Search tab, 415–416
 - Standard Libraries tab, 422–424
 - Triggers tab, 424–426
 - Vault Settings tab, 412–415
- VaultData folder, 411
- VBA (Visual Basic for Applications), 301
- VBS (Visual Basic Scripting), 65
- Verification On Rebuild option, 165–166, 175, 325, 505
- Verification option, 166
- version compatibility error, 219
- versions, 56
- vertical relation, 500–501
- VGA (Video Graphics Array) connector, 28
- video cards, 19, 21–23, 25–26
- Video Graphics Array (VGA) connector, 28
- View break lines option, Document Properties tab, 537
- View Labels page, Document Properties tab, 534
- View page, System Options tab, 518–519
- View Palette icon, Design Library control interface, 235
- View Palette, System Options tab, 491
- View transition setting, 519
- Viewing with eDrawings panel, VaultAdmin tool, 415
- views, 198–199
- virtual components, 293
- Virtual Private Networks (VPNs), 29–30
- Virtual Sharps page, Document Properties tab, 534–535
- viruses, 32
- Vista operating system
 - 3 Gig Switch, 93
 - installing Workgroup PDM Vault, 402
 - memory, 21
 - multiple monitors, 25
 - uninstalling SolidWorks, 380
 - versus Windows 7, 63
- Visual Basic for Applications (VBA), 301
- Visual Basic programming, 313
- Visual Basic Scripting (VBS), 65
- VPNs (Virtual Private Networks), 29–30

W

WAN (wide area network), 30–31, 91
 Warn about saving referenced documents option, 510
 Warn before saving documents with update errors option, 486
 warranty, manufacturer, 18
 wattage, laptops, 26
 Web Portal, 395
 WEI (Windows Experience Index), 91
 Welcome installation screen, SWIM, 140
 Weldments library, 245–247
 What's New document
 maintaining record of, 347
 when considering upgrade, 352
 What's New session, in-house training, 106
 What's Wrong dialog box, 485
 When rebuild error occurs option, 488
 Where Used list, 392, 422
 wide area network (WAN), 30–31, 91
 Windows Backup
 backing up computer, 80
 backing up files, 76–80
 recommended options, 83
 recovery tools, 80–81
 System Protection, 81–83
 System Restore, 81–83
 Windows Desktop Search, 249
 Windows Desktop Search Indexing Status dialog box, 525
 Windows Disk Cleanup utility, 96
 Windows Experience Index (WEI), 91
 Windows Explorer, 46, 300, 431
 Windows Features dialog box, 406, 425
 Windows font manager, 195
 Windows Indexing Service, 415
 Windows Registry, 84–86, 163, 379
 Windows Registry Editor, 382
 Windows Search service, 53–55, 75
 Windows Security Center, 91

Windows 7 operating system, 21, 25, 27, 93, 402
 Windows Vista operating system
 3 Gig Switch, 93
 installing Workgroup PDM Vault, 402
 memory, 21
 multiple monitors, 25
 uninstalling SolidWorks, 380
 versus Windows 7, 63
 Windows XP operating system, 20–21, 25, 63, 380, 402
 wiping registry, 381–383
 Wireframe and high quality HLR/HLV resolution slider, 545
 Wireframe mode, 198, 495
 Word summary, documenting solutions, 334
 Workgroup PDM, 56, 231, 265, 277, 404
 Workgroup PDM Viewer, 395, 405
 Working copy default option, Command Defaults tab, 453
 Working Copy designator, 417, 432
 workspace, 35–36
 workstation class computer, 19
 write access user, 338
 wrong version error, 378

X

Xi brand, 19
 XP operating system, 20–21, 25, 63, 380, 402

Y

Yahoo! Groups area, 334
 Yammer, 350

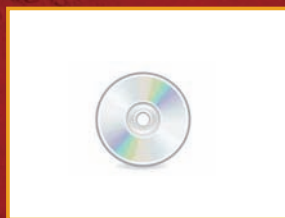
Z

Zeroes settings option, Document Properties tab, 529
 *.zip file, 33
 Zoom to fit when changing to standard views setting, 518

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Matt Lombard

is an independent mechanical design consultant. Because of the popularity of his SolidWorks blog DeZignstuff, and sessions at SolidWorks World, he has built a reputation as the go-to resource for everything SolidWorks.

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