

ENGINEERING GRAPHICS WITH **AutoCAD**[®] 2023



JAMES D. BETHUNE
DAVID BYRNES

FREE SAMPLE CHAPTER |



Engineering Graphics with AutoCAD® 2023

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Preface

This text teaches technical drawing and uses AutoCAD 2023 as its drawing instrument. Although it follows the general format of many technical drawing texts and presents much of the same material about drawing conventions and practices, the emphasis is on creating accurate, clear drawings. For example, the text shows how to locate dimensions on a drawing so that they completely define the object in accordance with ASME Y14.5-2009 national standards, but the presentation centers on the AutoCAD's **Dimensions** panel and its associated tools and options. The standards and conventions are presented and their applications are shown with the use of AutoCAD 2023. This integrated teaching concept is followed throughout the text.

Most chapters include drawing problems. The drawing problems are varied in scope and are open-ended, which means that there are several correct solutions. This is intended to encourage student creativity and increase their problem-solving abilities.

Chapters 1 through 3 cover tools on the **Draw** and **Modify** panels of the **Home** tab of AutoCAD's ribbon, and other commands needed to set up and start drawings. The text starts with simple **Line** commands and proceeds through geometric constructions. The final sections of Chapter 3 describe how to bisect a line and how to draw a hyperbola, a parabola, a helix, and an ogee curve. Redrawing many of the classic geometric shapes will help students learn how to use the **Draw** and **Modify** panels and other associated commands with accuracy and creativity.

Chapter 4 presents freehand sketching. Simply stated, there is still an important place for sketching in technical drawing. Many design ideas start as freehand sketches and are then developed on a computer. This chapter now includes extensive exercise problems associated with visual orientation.

Chapter 5 presents orthographic views. Students are shown how to draw three views of an object using AutoCAD 2023. The discussion includes projection theory, hidden lines, compound lines, oblique surfaces, rounded surfaces, holes, irregular surfaces, castings, and thin-walled objects. The chapter ends with several intersection problems. These problems serve as a good way to pull together orthographic views and projection theory. Several new, more difficult, exercise problems have been added to this edition. The chapter also includes an explanation of the differences between first- and third-angle projections as defined by ANSI and ISO conventions. Appropriate exercise problems help reinforce the understanding of the differences between the two standards.

Chapter 6 presents sectional views and introduces the **Hatch** and **Gradient** commands. The chapter includes multiple, broken-out, and partial sectional views and shows how to draw an S-break for a hollow cylinder.

Chapter 7 covers auxiliary views and shows how to use the **Snap**, **Rotate** command to create axes aligned with slanted surfaces. Secondary auxiliary views are also discussed. Solid modeling greatly simplifies the determination of the true shape of a line or plane, but a few examples of secondary auxiliary views help students refine their understanding of orthographic views and, eventually, the application of user coordinate systems (UCSs).

Chapter 8 shows how to dimension both two-dimensional shapes and orthographic views. The **Dimension** tools and their associated commands are demonstrated, and examples of how to use the **Dimension Styles** tool are included. The commands are presented as needed to create required dimensions. The conventions demonstrated are in compliance with ANSI Y14.5-2009.

Chapter 9 introduces tolerances. The chapter shows how to draw dimensions and tolerances using the **Dimension** and **Tolerance** commands, among others. The chapter ends with an explanation of fit types, and shows how to use the tables included in the Appendix to determine the maximum and minimum tolerances for matching holes and shafts.

Chapter 10 discusses the use of geometric tolerances and explains how AutoCAD 2023 can be used to create geometric tolerance symbols directly from dialog boxes. Both profile and positional tolerances are explained. The overall intent of the chapter is to teach students how to make parts fit together. Fixed and floating fastener applications are discussed, and design examples are given for both conditions.

Chapter 11 covers how to draw and design with the use of standard fasteners, including bolts, nuts, machine screws, washers, hexagon heads, square heads, set screws, rivets, and springs. Students are shown how to use the **Wblock** command to create drawings of the individual thread representations and how to use them for different size requirements.

Chapter 12 discusses assembly drawings, detail drawings, and parts lists. Instructions for drawing title blocks, tolerance blocks, release blocks, and revision blocks, and for inserting drawing notes are also included to give students better preparation for industrial practices.

Chapter 13 presents gears, cams, and bearings. The chapter teaches how to design by using gears selected from manufacturers' catalogs and websites. The chapter shows how to select bearings to support gear shafts and how to tolerance holes in support plates to maintain the desired center distances of meshing gears. It also explains how to create a displacement diagram and then draw the appropriate cam profile.

Chapter 14 introduces AutoCAD 3D capabilities. Both parallel (isometric) and perspective grids, as well as the world coordinate system (WCS) and user-defined coordinate systems (UCSs) are demonstrated so students learn the fundamentals of 3D drawings before drawing objects.

Chapter 15 shows how to create three-dimensional solid models. It includes examples of both parallel and perspective grids and using different **Visual Style** options. The chapter shows how to union, subtract, and intersect primitive shapes to create more complex models and orthographic views from those models.

Chapter 16, which is available online, presents two project problems: a milling vise and a tenon jig. These problems can be used for group or individual projects. These projects are intended to help students learn to work in groups and work on large, complex projects. This chapter can be found on the web as a supplement to the Instructor's Manual by registering your book at <https://www.pearson.com/us/higher-education/subject-catalog/download-instructor-resources.html>. Instructors may distribute this URL to students.

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James D. Bethune

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David Byrnes

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Brief Contents

Chapter 1	Getting Started	1
Chapter 2	Fundamentals of 2D Construction	29
Chapter 3	Advanced Commands	89
Chapter 4	Sketching	169
Chapter 5	Orthographic Views	199
Chapter 6	Sectional Views	285
Chapter 7	Auxiliary Views	327
Chapter 8	Dimensioning	367
Chapter 9	Tolerancing	441
Chapter 10	Geometric Tolerances	487
Chapter 11	Threads and Fasteners	535
Chapter 12	Working Drawings	579
Chapter 13	Gears, Bearings, and Cams	625
Chapter 14	Fundamentals of 3D Modeling	669
Chapter 15	Advanced Modeling	705
Index		781

You can find Chapter 16 and the Appendix at informit.com/title/9780137929993.
Click the *Downloads* tab to access the PDF file.

Chapter 16 Projects (Online Only)

Appendix (Online Only)

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Contents

Chapter 1 Getting Started	1		
1-1 Introduction	2		
Starting a New Drawing	2		
An Alternative Method to Starting a New Drawing	3		
1-2 Tabs and Panels	5		
Accessing Additional Commands Within a Panel	5		
Tooltips for Commands	6		
Accessing Other Help Information	6		
1-3 The Command Line Window	6		
Entering a Diameter Value	7		
1-4 Command Tools	7		
Determining the Command That a Tool Button Represents	7		
1-5 Starting a New Drawing	8		
1-6 Naming a Drawing	8		
Creating a New Drawing	8		
Saving a New Drawing File	9		
1-7 Drawing Units	11		
Specifying or Changing the Drawing Units	11		
Specifying or Changing the Precision of the Units System	13		
Specifying or Changing the Angle Units Value	13		
1-8 Drawing Limits	13		
Aligning the Drawing Limits with a Standard A3 (Metric) Paper Size	15		
1-9 Grid and Snap	17		
Setting the Grid and Snap Values	17		
1-10 Drawing Problem	18		
Specifying the Drawing Units	19		
Defining the Units Precision	19		
Setting the Sheet Size	20		
Setting Grid and Snap Values	20		
1-11 Save and Save As	21		
Using the Save and Save As Commands	21		
1-12 Open	23		
Using Open	23		
1-13 Close	24		
1-14 Exercise Problems	25		
Chapter 2 Fundamentals of 2D Construction	29		
2-1 Introduction	29		
2-2 Line—Random Points	29		
Randomly Selecting Points	29		
Exiting a Command Sequence	31		
Creating a Closed Area	32		
2-3 Erase	32		
Erasing Individual Lines	33		
Erasing a Group of Lines	34		
2-4 Line—Snap Points	34		
Drawing Problem	35		
2-5 Line—Dynamic Input	35		
Creating Lines Using Dynamic Input	37		
Accessing Dynamic Input Settings	38		
2-6 Construction Line	38		
Using Other Construction Line Commands: Hor, Ver, and Ang	40		
Using Another Construction Line Option: Offset	41		
2-7 Circle	42		
Drawing a Circle—Radius	42		
Drawing a Circle—Diameter	43		
Drawing a Circle—Two Points	43		
Drawing a Circle—Three Points	43		
Drawing a Circle—Tangent Tangent Radius Quadrant-Sensitive Applications	44		
2-8 Circle Centerline	45		
Creating Center Marks	45		
2-9 Polyline	46		
Drawing a Polyline	47		
Verifying That a Polyline Is a Single Entity	47		
Drawing a Polyline Arc	48		
Other Options with a Polyline Arc	48		
Drawing Different Line Thicknesses	50		
2-10 Spline	50		
2-11 Ellipse	51		
Drawing an Ellipse—Axis Endpoint	51		
Drawing an Ellipse—Center	52		
Drawing an Elliptical Arc	53		
2-12 Rectangle	54		
Drawing a Rectangle	54		
Exploding a Rectangle	54		
2-13 Polygon	55		
Drawing a Polygon—Center Point	55		
Drawing a Polygon—Edge Distance	56		
2-14 Point	56		
Changing the Shape of a Point	57		
Changing the Size of a Point	57		

2-15 Text	57	3-4 Osnap—Midpoint	93
Creating Multiline Text	57	Drawing a Circle About the Midpoint of a Line	93
The Multiline Text Editor	58	3-5 Osnap—Intersection	93
Accessing the Multiline Text Editor	59	Using Osnap Intersection Mode to Define an Ellipse	94
Changing Text Height	59	3-6 Osnap—Apparent Intersection	94
Changing the Text Font	59	Drawing a Circle Centered About an Apparent	
Justifying Text	60	Intersection	94
Justifying Text to the Right	60	3-7 Osnap—Center	95
Using the Symbol Options	61	Drawing a Line to the Center Point of a Circle	95
Changing Text Color	61	3-8 Osnap—Quadrant	96
2-16 Move	62	Drawing a Line to One of a Circle's Quadrant Points	96
Moving an Object	62	3-9 Osnap—Perpendicular	96
2-17 Copy	63	Drawing a Line Perpendicular to an Existing Line	97
Copying an Object	63	3-10 Osnap—Tangent	97
Making Multiple Copies	64	Drawing a Line Tangent to a Circle	97
2-18 Offset	64	3-11 Osnap—Nearest	97
2-19 Mirror	65	Drawing a Line from a Point to the Nearest	
2-20 Array	65	Selected Point on an Existing Line	98
Using the Rectangular Array Option	66	3-12 Drawing Problem	98
Using the Polar Array Option	67	3-13 Drawing Problem	100
Using the Path Array Option	67	3-14 Grips	101
2-21 Rotate	68	Turning Grips Off	102
Rotating an Object	69	Accessing the Grips Dialog Box	102
2-22 Trim	69	3-15 Grips—Extend	103
Using Trim	69	Extending the Length of a Line	103
2-23 Extend	70	3-16 Grips—Move	104
2-24 Break	71	Moving an Object Using Grips	104
Using the Break Command	71	3-17 Grips—Rotate	104
Using the First Point Option	72	Rotating an Object Using Grips	104
2-25 Chamfer	72	3-18 Grips—Scale	105
Creating a Chamfer	73	Changing the Scale of an Object	105
2-26 Fillet	73	3-19 Grips—Mirror	106
Creating a Fillet	73	Mirroring an Object	106
2-27 Table	74	3-20 Blocks	106
Creating a Table	74	Creating a Block	107
2-28 Exercise Problems	78	Inserting a Block	108
		Changing the Scale of a Block	109
		Exploding a Block	109
Chapter 3 Advanced Commands	89	3-21 Working with Blocks	110
3-1 Introduction	89	Inserting Blocks at Different Angles	110
3-2 Object Snap	89	Inserting Blocks with Different Scale Factors	112
Accessing the Object Snap Modes	89	Using the Blocks Palette to Change the Shape	
Enabling Object Snap Modes	91	of a Block	113
Changing the Size of the Osnap Marker	91	Combining Blocks	114
3-3 Osnap—Endpoint	92		
Snapping to the Endpoint of an Existing Line	92		

3-22 Wblock	115	4-7 Curves	173
Using Wblock to Create a New Drawing File	115	4-8 Drawing Problem	175
Verifying That Wblock Created a New Drawing	116	4-9 Isometric Sketches	176
3-23 Layers	117	4-10 Drawing Problem	179
Creating New Layers	118	4-11 Oblique Sketches	180
Changing the Color and Linetype of a Layer	119	4-12 Perspective Sketches	182
Drawing on Different Layers	122	4-13 Working in Different Orientations	184
Changing an Object's Layer	123	4-14 Exercise Problems	185
Changing the Scale of a Linetype	124		
Using the Match Layer Tool	125		
Turning Layers Off	126		
3-24 Attributes	126	Chapter 5 Orthographic Views	199
Adding an Attribute to a Block	126	5-1 Introduction	199
Creating a New Block That Includes Attributes	129	5-2 Three Views of an Object	200
Inserting an Existing Block with Attributes	130	5-3 Visualization	201
Editing an Existing Attribute	131	Drawing a Three-Dimensional Box	202
3-25 Title Blocks with Attributes	132	5-4 Hidden Lines	203
3-26 Edit Polyline	135	5-5 Hidden Line Conventions	204
Creating a Splined Polyline from a Given Polyline	135	5-6 Drawing Hidden Lines	205
3-27 Edit Spline	136	Adding Hidden Linetypes to a Drawing	205
Editing a Spline	136	Creating a Hidden Layer for General Use	207
3-28 Edit Text	137	Changing Layers	208
Changing Existing Text	137	5-7 Precedence of Lines	209
3-29 Constructing the Bisector of an Angle, Method I	138	5-8 Slanted Surfaces	210
3-30 Constructing the Bisector of an Angle, Method II	138	5-9 Projection Between Views	211
3-31 Constructing an Ogee Curve (S-Curve) with Equal Arcs	139	5-10 Drawing Problem	212
3-32 Constructing a Parabola	141	5-11 Compound Lines	213
3-33 Constructing a Hyperbola	141	5-12 Drawing Problem	214
3-34 Constructing a Spiral	143	5-13 Oblique Surfaces	215
3-35 Constructing a Helix	143	5-14 Drawing Problem	217
3-36 Designing by Using Shape Parameters	145	5-15 Rounded Surfaces	217
Drawing Problem	145	5-16 Drawing Problem	219
Drawing Problem	146	5-17 Holes	220
3-37 Exercise Problems	149	5-18 Holes in Slanted Surfaces	222
		Drawing an Ellipse Representing a Projected Hole	223
		Drawing Three Views of a Hole in a Slanted Surface	223
		Drawing Three Views of a Hole Through an Oblique Surface	224
Chapter 4 Sketching	169	5-19 Cylinders	226
4-1 Introduction	169	5-20 Drawing Problem	227
4-2 Establishing Your Own Style	169	5-21 Cylinders with Slanted and Rounded Surfaces	228
4-3 Graph Paper	170	5-22 Drawing Problem	228
4-4 Pencils	170	5-23 Drawing Conventions and Cylinders	229
4-5 Lines	171	5-24 Irregular Surfaces	230
4-6 Proportions	172		

5-25	Drawing Problem	231	Chapter 7 Auxiliary Views	327
5-26	Hole Callouts	232	7-1 Introduction	327
	Drawing a Countersunk Hole	234	7-2 Projection Between Normal and Auxiliary Views	328
5-27	Castings	234	Rotating the Drawing's Axis System	330
5-28	Drawing Problem	237	7-3 Drawing Problem	331
5-29	Thin-Walled Objects	238	7-4 Transferring Lines Between Views	332
5-30	Drawing Problem	239	Measuring the Length of a Line	332
5-31	Intersections	240	Using Grips to Move a Line	333
5-32	Drawing Problem	241	Rotating and Moving a Line	334
5-33	Drawing Problem	242	7-5 Drawing Problem	334
5-34	Drawing Problem	243	7-6 Projecting Rounded Surfaces	335
5-35	Designing by Modifying an Existing Part	245	7-7 Drawing Problem	335
5-36	Drawing Standards	247	7-8 Projecting Irregular Surfaces	337
5-37	Third- and First-Angle Projections	249	7-9 Drawing Problem	337
5-38	Exercise Problems	251	7-10 Drawing Problem	339
Chapter 6 Sectional Views		285	7-11 Partial Auxiliary Views	340
6-1	Introduction	285	7-12 Sectional Auxiliary Views	341
6-2	Cutting Plane Lines	287	7-13 Auxiliary Views of Oblique Surfaces	342
	Drawing a Cutting Plane Line—Method I	287	7-14 Secondary Auxiliary Views	343
	Drawing a Cutting Plane Line—Method II	290	Drawing the First Auxiliary View	345
6-3	Indicating Solid Cuts	290	Drawing the Secondary Auxiliary View	345
6-4	Hatch	292	7-15 Drawing Problem	346
	Hatching a Given Area	292	7-16 Secondary Auxiliary View of an Ellipse	348
	Changing Hatch Patterns	293	7-17 Exercise Problems	350
	Changing the Spacing and Angle of a Hatch Pattern	294	Chapter 8 Dimensioning	367
6-5	Drawing Problem	295	8-1 Introduction	367
6-6	Styles of Section Lines	296	8-2 Terminology and Conventions	368
6-7	Sectional View Location	296	Some Common Terms	368
6-8	Holes in Sections	297	Some Dimensioning Conventions	369
6-9	Gradients	298	Some Common Errors to Avoid	370
	Creating a Gradient	298	8-3 Linear Dimensions	370
6-10	Offset Sections	299	Creating a Linear Dimension	370
6-11	Multiple Sections	300	Creating a Vertical Dimension	371
6-12	Aligned Sections	300	Creating a Horizontal Dimension by	
6-13	Drawing Conventions in Sections	301	Selecting the Object to Be Dimensioned	371
6-14	Half, Partial, and Broken-Out Sectional Views	302	Changing the Default Dimension Text Using the Text Option	372
6-15	Removed Sectional Views	303	Changing the Default Dimension Text with the Mtext Option	373
6-16	Breaks	303	Editing an Existing Dimension	374
	Drawing an S-Break	304	8-4 Dimension Styles	376
6-17	Sectional Views of Castings	305	Changing the Scale of a Drawing	377
6-18	Exercise Problems	306	Using the Text Tab	378

8-5 Units	380	8-25 Knurling	413
Preventing a Zero from Appearing to the Left of the Decimal Point	382	8-26 Keys and Keyseats	414
Changing the Number of Decimal Places in a Dimension Value	383	8-27 Symbols and Abbreviations	414
8-6 Aligned Dimensions	384	8-28 Symmetry and Centerline	415
Creating an Aligned Dimension	384	8-29 Dimensioning to Points	415
Using Text Options	385	8-30 Coordinate Dimensions	416
8-7 Radius and Diameter Dimensions	385	8-31 Sectional Views	417
Creating a Radius Dimension	385	8-32 Orthographic Views	417
Altering the Default Dimension	386	8-33 Very Large Radii	419
Removing the Center Mark from a Radius Dimension	387	Creating a Radius for Large Radii	419
Creating a Diameter Dimension	388	8-34 Exercise Problems	420
Creating Center Marks and Lines	389		
Adding Linear Dimensions to Given Centerlines	389	Chapter 9 Tolerancing	441
8-8 Angular Dimensions	391	9-1 Introduction	441
Creating an Angular Dimension	392	9-2 Direct Tolerance Methods	441
Avoiding Overdimensioning	393	9-3 Tolerance Expressions	442
8-9 Ordinate Dimensions	393	9-4 Understanding Plus and Minus Tolerances	443
Moving the Origin and the Origin Icon	394	9-5 Creating Plus and Minus Tolerances with AutoCAD	443
Adding Ordinate Dimensions to an Object	395	Creating Plus and Minus Tolerances Using the Text Option	443
8-10 Baseline Dimensions	396	Creating Plus and Minus Tolerances with the Text Override Tool	444
Using the Baseline Dimension Tool	396	Using Dtext to Create a Plus and Minus Tolerance	444
8-11 Continued Dimensions	398	Using the Dimension Style Manager	444
Using the Continue Dimension Command	398	9-6 Limit Tolerances	446
8-12 Quick Dimension	399	9-7 Creating Limit Tolerances in AutoCAD	447
Using the Quick Dimension Tool	399	Creating a Limit Tolerance Using the Dimension Style Manager	447
8-13 Center Mark	400	Using the Properties Palette to Change an Existing Dimension to a Limit Tolerance Dimension	448
Adding Centerlines to a Given Circle	400	9-8 Angular Tolerances	448
8-14 Mleader and Qleader	401	Setting the Precision for Angular Dimensions and Tolerances	449
Creating a Quick Leader	401	Creating an Angular Dimension and Tolerance	449
Drawing a Curved Leader Line	402	9-9 Standard Tolerances	450
Drawing a Leader Line with a Dot at Its End	403	9-10 Double Dimensioning	450
8-15 Text Angle	404	9-11 Chain Dimensions and Baseline Dimensions	451
Changing the Angle of Dimension Text	404	Creating Baseline Dimensions	452
8-16 Tolerances	405	9-12 Tolerance Studies	453
8-17 Dimensioning Holes	406	Calculating A's Maximum Length	454
Dimensioning Individual Holes	406	Calculating A's Minimum Length	454
Dimensioning Hole Patterns	407	9-13 Rectangular Dimensions	454
8-18 Placing Dimensions	408	9-14 Hole Locations	455
8-19 Fillets and Rounds	409	9-15 Choosing a Shaft for a Toleranced Hole	456
8-20 Rounded Shapes—Internal	409	For Linear Dimensions and Tolerances	456
8-21 Rounded Shapes—External	410		
8-22 Irregular Surfaces	410		
8-23 Polar Dimensions	411		
8-24 Chamfers	412		

9-16 Drawing Problem	457	10-14 Profiles	505
9-17 Drawing Problem	458	10-15 Runouts	506
9-18 Standard Fits (Metric Values)	458	10-16 Positional Tolerances	508
Clearance Fits	460	10-17 Virtual Condition	509
Transition Fits	460	Calculating the Virtual Condition for a Shaft	509
Interference Fits	460	Calculating the Virtual Condition for a Hole	510
9-19 Nominal Sizes	461	10-18 Floating Fasteners	510
9-20 Hole and Shaft Basis	461	10-19 Drawing Problem	511
9-21 Drawing Problem	461	10-20 Drawing Problem	511
9-22 Standard Fits (Inch Values)	462	10-21 Fixed Fasteners	512
Fits Defined by Inch Values	462	10-22 Drawing Problem	513
9-23 Drawing Problem	463	10-23 Design Problems	514
9-24 Preferred and Standard Sizes	463	10-24 Exercise Problems	516
9-25 Surface Finishes	465		
9-26 Surface Control Symbols	467	Chapter 11 Threads and Fasteners	535
9-27 Drawing Problem	467	11-1 Introduction	535
Floating Condition	468	11-2 Thread Terminology	535
Fixed Condition	469	11-3 Thread Callouts—Metric Units	536
Designing a Hole Given a Fastener Size	471	11-4 Thread Callouts—English Units	536
9-28 Exercise Problems	472	11-5 Thread Representations	538
		Drawing a Detailed Thread Representation	538
Chapter 10 Geometric Tolerances	487	Creating an Internal Detailed Thread	
10-1 Introduction	487	Representation in a Sectional View	539
10-2 Tolerances of Form	488	Creating a Schematic Thread Representation	540
10-3 Flatness	488	Creating an Internal Schematic Thread	
10-4 Straightness	489	Representation	541
10-5 Straightness (RFS and MMC)	489	Creating a Simplified Thread Representation	542
10-6 Circularity	492	11-6 Orthographic Views of Internal Threads	542
10-7 Cylindricity	493	11-7 Sectional Views of Internal Thread	
10-8 Creating Geometric Tolerances in		Representations	544
AutoCAD	493	11-8 Types of Threads	544
Defining a Datum	494	11-9 How to Draw an External Square Thread	545
Defining a Straightness Value	494	11-10 How to Draw an Internal Square Thread	546
Creating a Positional Tolerance	495	11-11 How to Draw an External Acme Thread	546
Creating a Basic Dimension	495	11-12 Bolts and Nuts	548
Creating Basic Dimensions from Existing		11-13 Screws	549
Dimensions	496	11-14 Studs	550
Adding a Limit Feature Tolerance to a Hole	497	11-15 Head Shapes	550
Adding a Positional Tolerance to the Hole's		Drawing a Hexagon-Shaped (Hex) Head	550
Feature Tolerance	498	Drawing a Square-Shaped Head	552
Adding a Geometric Tolerance with a Leader Line	499	11-16 Nuts	554
10-9 Tolerances of Orientation	500	Drawing a Hexagon-Shaped Finished Nut	554
10-10 Datums	501	Drawing a Locking Nut	556
10-11 Perpendicularity	502	11-17 Drawing Problem	557
10-12 Parallelism	504		
10-13 Angularism	504		

11-18 Drawing Problem	558	Chapter 13 Gears, Bearings, and Cams	625
11-19 Standard Screws	559	13-1 Introduction	625
11-20 Set screws	560	13-2 Types of Gears	625
11-21 Washers	561	13-3 Gear Terminology—Spur	626
Drawing a Plain Washer	561	English and Metric Units for Spur Gears	627
11-22 Keys	562	13-4 Spur Gear Drawings	628
11-23 Rivets	563	13-5 Drawing Problem	629
11-24 Springs	564	13-6 Drawing Problem	631
Drawing a Detailed Representation of an		Drawing Meshing Spur Gears	632
Extension Spring	564	13-7 Drawing Problem	632
Drawing a Detailed Representation of a		13-8 Selecting Spur Gears	633
Compression Spring	566	13-9 Center Distance Between Gears	635
11-25 Tool Palettes	567	13-10 Drawing Problem	635
Creating a Shoulder Screw	567	13-11 Combining Spur Gears	636
Changing the Scale of a Tool Palette Block	567	13-12 Gear Terminology—Bevel	637
Modifying the Block	568	13-13 How to Draw Bevel Gears	639
11-26 Exercise Problems	569	Drawing a Matched Set of Beveled Gears	640
 		13-14 Worm Gears	641
Chapter 12 Working Drawings	579	13-15 Helical Gears	643
12-1 Introduction	579	13-16 Racks	644
12-2 Assembly Drawings	579	13-17 Ball Bearings	645
12-3 Drawing Formats (Templates)	582	13-18 Drawing Problem	646
Adding a Drawing Template	582	13-19 Bushings	647
12-4 Title Block	584	13-20 Drawing Problem	648
Drawing Titles (Names)	585	13-21 Cam Displacement Diagrams	650
Drawing Numbers	586	Setting Up a Displacement Diagram	651
Company Name	586	13-22 Cam Motions	652
Scale	586	Uniform Motion	652
Release Date	586	Modified Uniform Motion	652
Sheet	586	Harmonic Motion	653
12-5 Revision Block	586	Uniform Acceleration and Deceleration	654
12-6 Tolerance Block	588	13-23 Cam Followers	655
12-7 Release Block	588	13-24 Drawing Problem	655
12-8 Parts List (Bill of Materials [BOM])	589	13-25 Exercise Problems	658
12-9 Detail Drawings	591		
12-10 First-Angle Projection	591	 	
12-11 Drawing Notes	592	Chapter 14 Fundamentals of 3D Modeling	669
12-12 Design Layouts	593	14-1 Introduction	669
12-13 Drawing Problem	593	14-2 The World Coordinate System	669
Creating the Design Layout	594	Changing the Background Color to White	671
Creating a Drawing Using Layers	596	14-3 Viewpoints	672
Creating a Drawing from a Layout	597	Changing the Viewpoint by Using the View	
12-14 Drawing Problem	598	Command	672
Creating an Assembly Drawing	599	Returning to the Original WCS Orientation	673
12-15 Drawing Problem	601		
12-16 Exercise Problems	603		

14-4 Perspective and Parallel Grids	673	15-4 Cylinder	709
Creating a Drawing with a Perspective Grid	674	Drawing a Cylinder with a Circular Base	709
Returning to the 2D WCS	675	Drawing a Cylinder with an Elliptical Base	710
14-5 Setting the 3D Modeling Workspace	676	15-5 Cone	711
Accessing the 3D Modeling Mode	676	Drawing a Cone with an Elliptical Base	711
14-6 User Coordinate Systems	677	Drawing a Cone with a Circular Base	712
Drawing a Solid Box	677	15-6 Wedge	713
Creating a UCS on the Top Surface	679	Drawing a Wedge by Defining Its Corner Point	713
Saving a UCS	679	Drawing a Wedge by Defining Its Center Point	713
Returning to the WCS	680	Aligning a Wedge with an Existing Wedge	714
Restoring a Saved UCS	680	15-7 Torus	715
Defining a UCS by Selecting Three Points	681	Drawing a Torus	716
14-7 Editing a Solid Model	682	15-8 Extrude	716
Changing the Size of a Solid Model	683	Extruding a 2D Polyline	716
14-8 Visual Styles	683	Creating a Polyline from Line Segments	717
Changing Visual Styles	684	15-9 Revolve	718
14-9 Rotating a UCS Axis	685	Creating a Revolved Solid Object	718
14-10 Drawing Problem	686	15-10 Helix	719
14-11 Visual Errors	688	15-11 Sweep	720
14-12 Drawing Problem	688	15-12 Loft	721
Drawing a Circle on the Upper Front Surface	689	15-13 Union and Subtract	722
Adding a Rectangle on the Top Surface	689	Unioning Two Objects	722
Adding an Ellipse on the Left Vertical Surface	690	Subtracting an Object	723
14-13 Orthographic Views	690	15-14 Intersect	724
Creating Orthographic Views	692	15-15 Solid Modeling and UCSs	725
14-14 Line Thickness	693	15-16 Combining Solid Objects	727
Using the Thickness Variable	693	Setting Up the Drawing	727
Drawing a Curve with Thickness	695	Drawing the First Box	727
14-15 Using the Thickness Variable to Create Objects	696	Creating the Internal Open Volume	728
Drawing the Box	696	Creating the Top Cutout	729
Creating a New UCS	697	Creating a Box	730
Drawing the Right Cylinder	697	Creating a Wedge	730
Drawing the Top Cylinder	697	15-17 Intersecting Solids	732
Returning the Drawing to Its Original Settings	698	Setting Up the Drawing	732
14-16 Exercise Problems	699	Drawing the Cone	733
		Drawing the Cylinder	734
		Completing the 3D Model	735
		Creating Viewports for the Orthographic Views	736
Chapter 15 Advanced Modeling	705	15-18 Solid Models of Castings	737
15-1 Introduction	705	Drawing the Basic Shape	737
15-2 Box	706	Creating a Polyline from the Basic Shape	738
Drawing a Box (Corner Option)	706	Extruding the Shape	739
Changing the Visual Style	706	Adding the Holes	740
Drawing a Box from Given Dimensions	707	Creating the Rounded Edges	741
Drawing a Cube	708	15-19 Thread Representations in Solid Models	742
Using Dynamic Grips	708	15-20 List	743
15-3 Sphere	709	15-21 Massprop	744
Drawing a Sphere	709		

15-22 Face and Edge Editing	744
Extruding a Face	745
Extruding a Face Along a Path	745
Extruding Two Faces at the Same Time	746
Moving a Face	747
Offsetting Faces	747
Rotating a Face	748
Tapering a Face	749
Copying a Face	750
Copying Edges	750
Imprinting an Object	751
15-23 Exercise Problems	752

Index

You can find Chapter 16 and the Appendix at informit.com/title/9780137929993. Click the *Downloads* tab to access the PDF file.

Chapter 16 Projects (Online Only)	
16-1 Introduction	
16-2 Project 1: Milling Vise	
Creating the Milling Vise	
Creating the Base Subassembly	
Creating the Middle Subassembly	
Creating the Top Subassembly	
16-3 Project 2: Tenon Jig	
Drawing the Clamping Subassembly	
Drawing the Vertical Subassembly	
Drawing the Base Plate Subassembly	
Drawing the Guide Plate Subassembly	
Creating an Assembly Drawing of the Tenon Jig	
781	

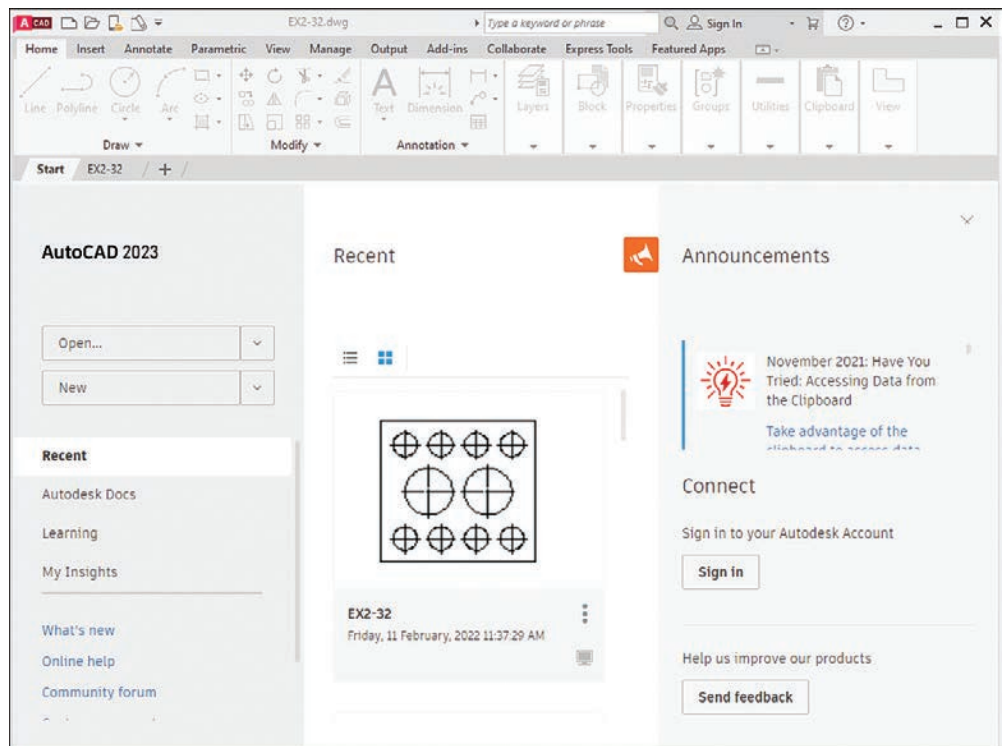
Appendix (Online Only)

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chapter one

Getting Started

Figure 1-1



1-1 Introduction

This chapter introduces you to AutoCAD 2023. It covers basics such as using the **Application** menu, starting new drawings, making settings and entering data in dialog boxes, and saving your work.

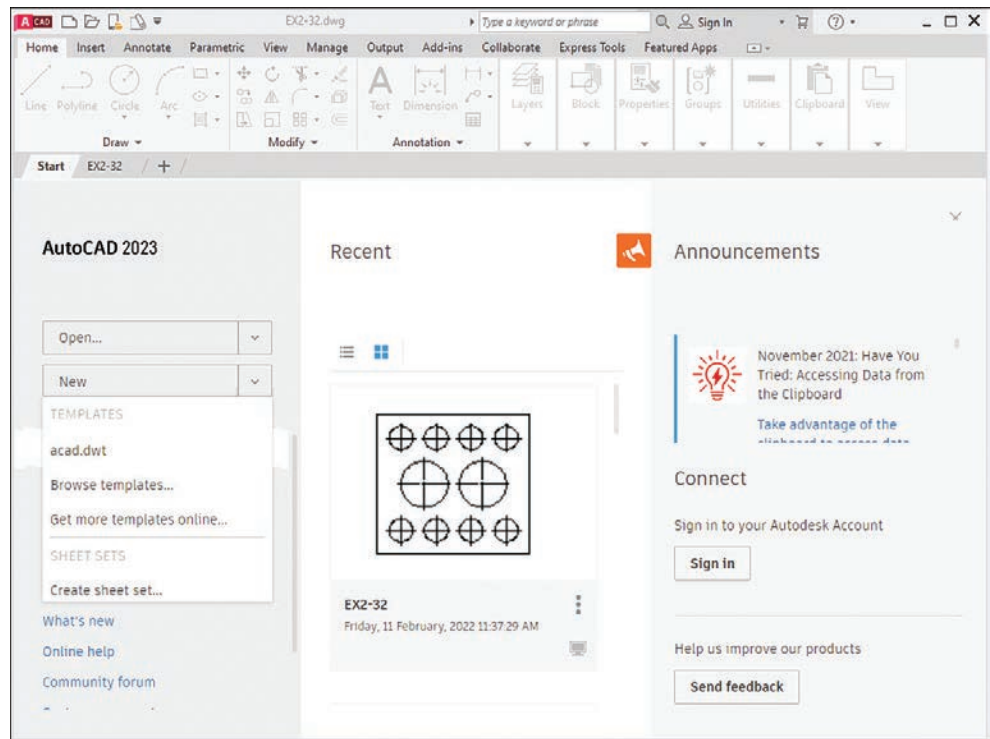
Figure 1-1 shows the initial AutoCAD drawing screen, which appears when the program is first started.

Starting a New Drawing

1 Click the down arrow beside the **New** button on AutoCAD's opening screen.

A list of recently used templates appears in the drop-down menu (Figure 1-2). You will use various templates throughout the text, but for a start, you will use the **acad.dwt** template. The **acad.dwt** template defines inches as its primary units. If the **acad.dwt** template does not appear in the drop-down, click **Browse templates...** and select it from there.

Figure 1-2



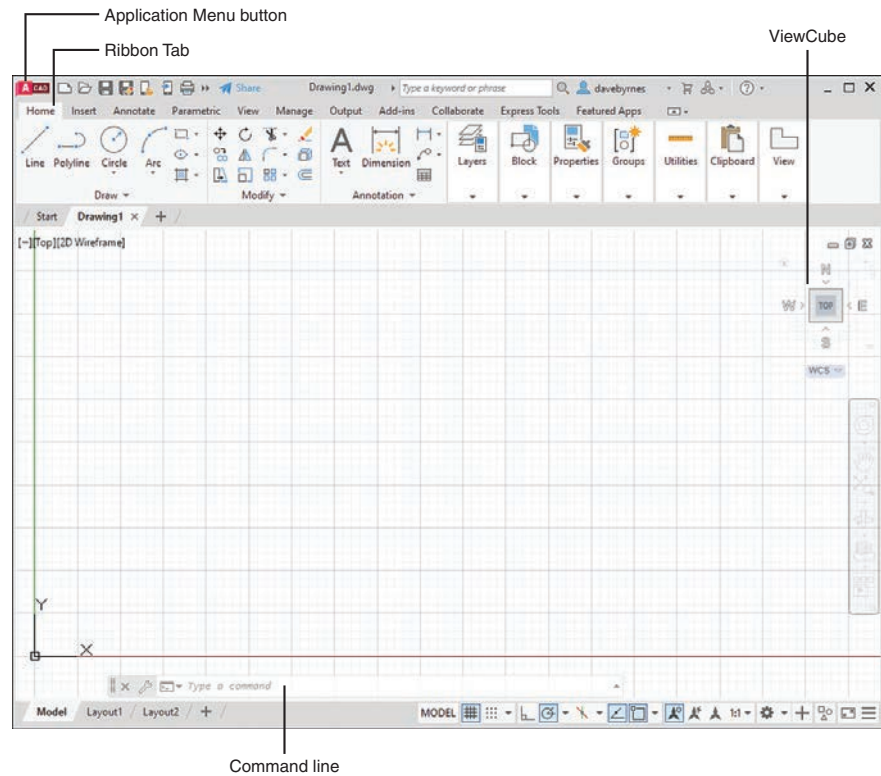
2 Click **acad.dwt** on the **Templates** list.

The drawing screen appears (Figure 1-3).

NOTE

The tool panels in the figure have a light-colored background—for printing clarity. Your background may be dark.

Figure 1-3



An Alternative Method to Starting a New Drawing

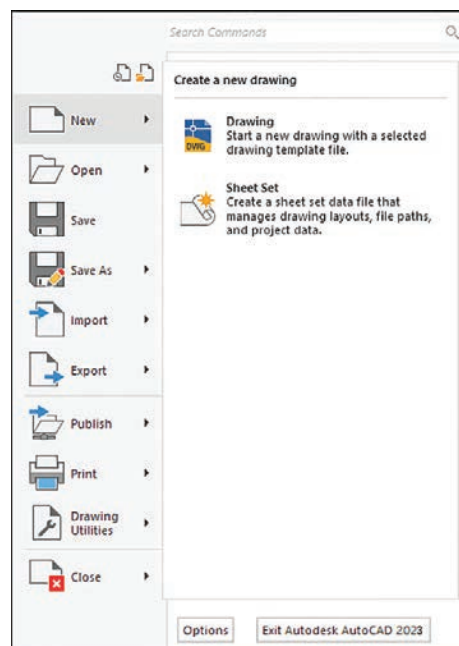
- 1 Click the **Application Menu** button in the upper-left corner of the drawing screen to display the **Application** menu.

A list of drawing commands and utilities appears (Figure 1-4).

- 2 Click **New**.

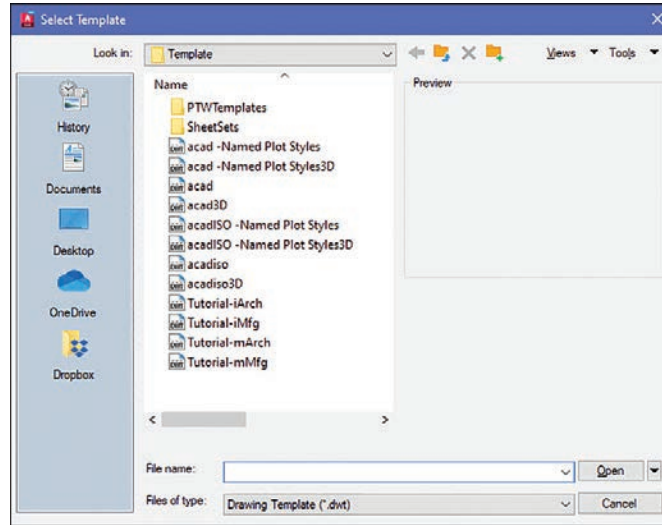
The **Select Template** dialog box appears (Figure 1-5).

Figure 1-4



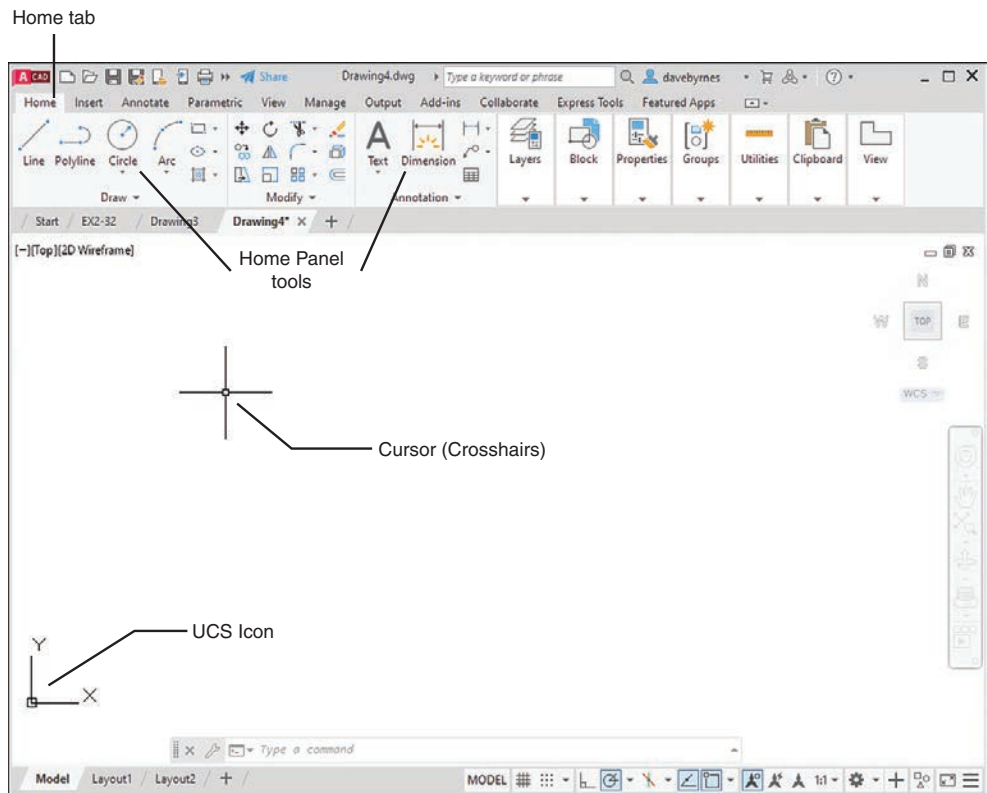
3 Select the **acad** template and click **Open**.

Figure 1-5



The AutoCAD drawing screen appears (Figure 1-6). The Ribbon appears at the top of the screen, showing a group of tabs and panels. Select different tabs to access other groups of panels. Panels contain commands.

Figure 1-6



The command line is located at the bottom of the screen, as are other tools (icons) for commands such as **Grid** and **Snap**. Use the command line to enter inputs for the commands, among other uses.

The drawing's name appears at the top of the screen. In Figure 1-4, for example, the drawing name is Drawing1.dwg. This is a default name created by AutoCAD. If a drawing name had been entered, it would appear where the Drawing1.dwg title currently appears.

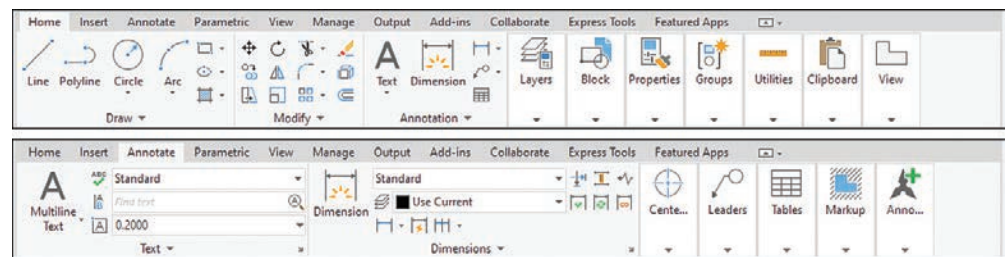
The large open area in the center of the screen is the *drawing area* or *drawing editor*. You create drawings in this area.

The symbol at the bottom-left corner of the drawing area is called the **User Coordinate System (UCS)** icon. It shows the direction of positive X and Y coordinates.

1-2 Tabs and Panels

The headings across the ribbon at the top of the screen (**Home**, **Insert**, etc.) are called *tabs*, and the groups of commands on the tabs are called *panels*. Figure 1-7 shows the **Home** panels and the **Annotate** panels.

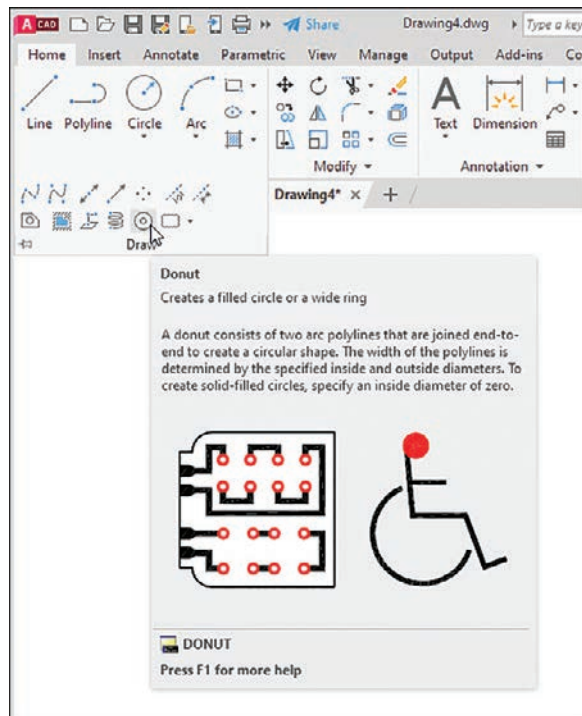
Figure 1-7



Accessing Additional Commands Within a Panel

Each panel shows a group of the most commonly used commands. Additional commands are available by clicking the arrow to the right of the panel's name. Figure 1-8 shows the additional **Draw** commands available.

Figure 1-8



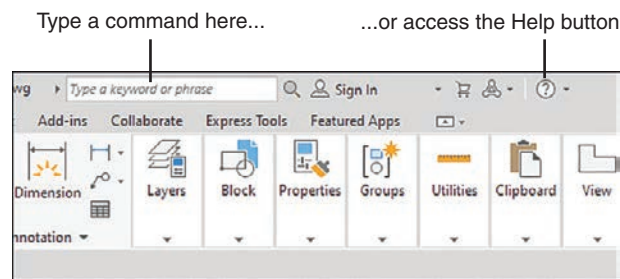
Tooltips for Commands

A *tooltip* is a pop-up help window that appears when the cursor is hovered over a command's icon (Figure 1-8). Initially, when you place the cursor over a command icon but don't click, a tooltip appears, identifying the command. After a few seconds the tooltip expands to further define the command.

Accessing Other Help Information

If you cannot find a command or if you need further instructions for operating a particular command, type a keyword into the text box in the program's title bar, and press **Enter** or click the **Access to Help** button located in the top-right section of the screen (Figure 1-9). The icon for the **Access to Help** tool is a question mark within a circle. The **Help** dialog box appears. Type in the name of what you are seeking and click the magnifying glass icon just to the right of the search box.

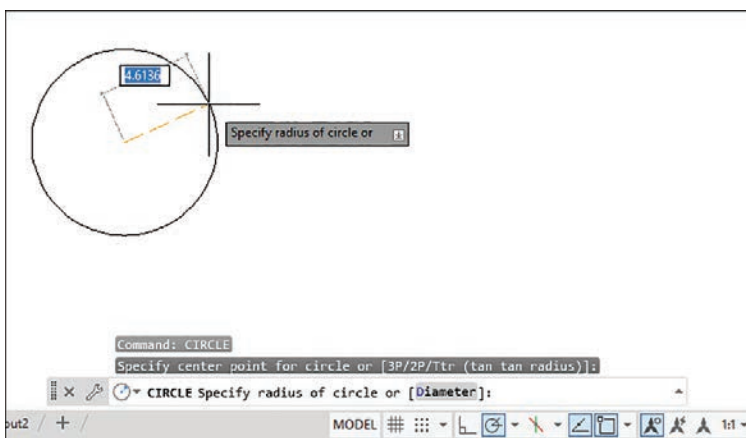
Figure 1-9



1-3 The Command Line Window

The command line window is located at the bottom of the drawing screen. Use it to access commands that do not have their own icons or to select options associated with the command. Figure 1-10 shows a circle. The word **CIRCLE** automatically appears in the command line when you click the **Circle** tool on the **Draw** panel. As presented, the circle will be defined by entering a radius value. Enter the radius value into the box with the blue background before clicking the left mouse button to complete the circle. If the radius value does not appear, press the F12 function key and ensure that the Dynamic Input is **ON**.

Figure 1-10



The command line shows the word Diameter in brackets: [Diameter]. Follow the next steps to use the **Circle** command's **Diameter** option.

Entering a Diameter Value

- 1 Click the **Circle** tool on the Home panel and draw a circle.
- 2 Click the command line box.
- 3 Type **d** and press **Enter**.

The system is now set for a diameter value for the circle.

- 4 Enter a value for the diameter of the circle and press **Enter**.

The options shown at the command line always include one uppercase letter. It may not always be the first letter. Type that letter and press **Enter** to access the option.

Diameter values may also be entered by first clicking the arrowhead next to the **Circle** tool and selecting the **Center, Diameter** option.

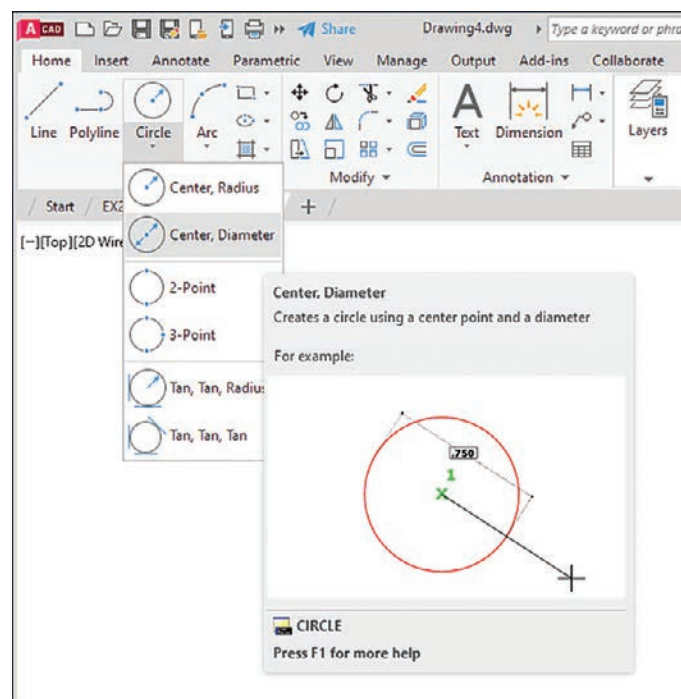
1-4 Command Tools

A *tool button* displays a picture (icon) that represents an AutoCAD command. Most commands have equivalent tool buttons.

Determining the Command That a Tool Button Represents

Figure 1-11 shows the steps to find the name and description of the command that the tool button executes.

Figure 1-11



1 Hover the cursor arrow over the selected tool button.
In the example shown, the **Circle** command tool button with the **Diameter** option is selected.

2 Hold the arrow still without pressing any mouse buttons.

The command name appears in a tooltip. If you continue to keep the cursor arrow on the tool button, an expanded tooltip that further describes the command appears.

1-5 Starting a New Drawing

When you start a new drawing, AutoCAD assigns a drawing name. The drawing units are specified, the drawing limits are modified, if needed, and **Grid** and **Snap** values are defined. The following four sections show you how to start a new drawing.

1-6 Naming a Drawing

You can use any combination of letters and numbers as a file name. Either uppercase or lowercase letters can be used, since AutoCAD file names are not case sensitive. The symbols \$, -, and _ (underscore) can also be used. Other symbols, such as % and *, cannot be used (Figure 1-12).

Figure 1-12

Correct drawing names:
FIRST EK-131-1 PA1-1a

Incorrect drawing names:
100% *.*

To locate a file on a the C: drive:
C:FIRST

All AutoCAD drawing files will automatically have the extension .dwg added to the given file name by default. If you name a drawing **FIRST**, it will appear in the files as **FIRST.dwg**. (A default setting is one that AutoCAD will use unless specifically told to use some other value.)

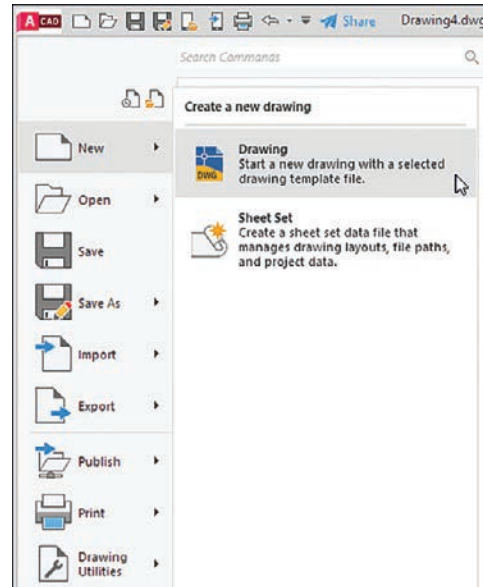
If you want to locate a file on another drive, specify the drive letter followed by a colon in front of the drawing name. For example, in Figure 1-12 **C:FIRST** will locate the drawing file **FIRST** on the C: drive.

Creating a New Drawing

There are three ways to access the **Create New Drawing** dialog box that is used to name a new drawing:

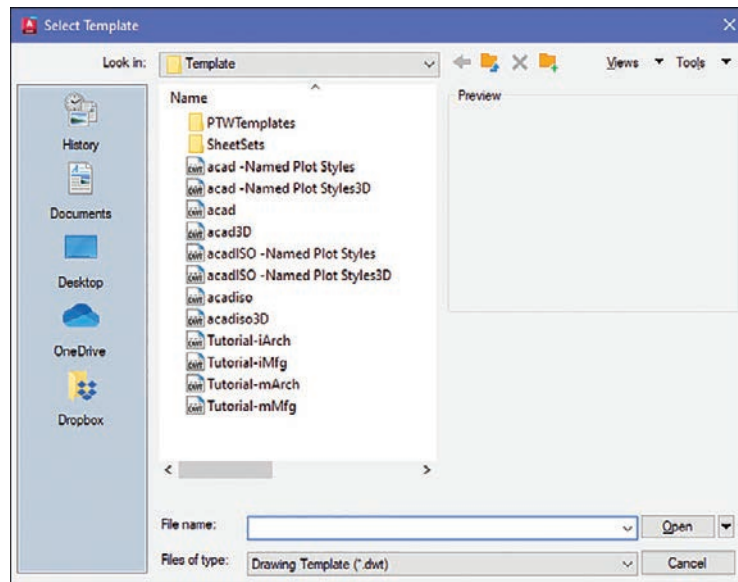
- Select **New** from the **Application** menu (Figure 1-13).
- Type the word **new** at a command prompt.
- Hold down the **Ctrl** key and press **N**.

Figure 1-13



Any of these methods will open the **Select Template** dialog box (Figure 1-14). The **acad** template will set up a drawing with inch values and ANSI style dimensions. The **acadiso** template will set up a drawing with millimeter values and ISO-style dimensions.

Figure 1-14



Saving a New Drawing File

The first time you use one of the **Save** tools to save your drawing, you must give your drawing a name (Figure 1-15). When you click **Save** for the first time in a new drawing, the **Save Drawing As** dialog box appears (Figure 1-16). Select a folder in which to save your work and enter a file name in the text box.

Figure 1-15

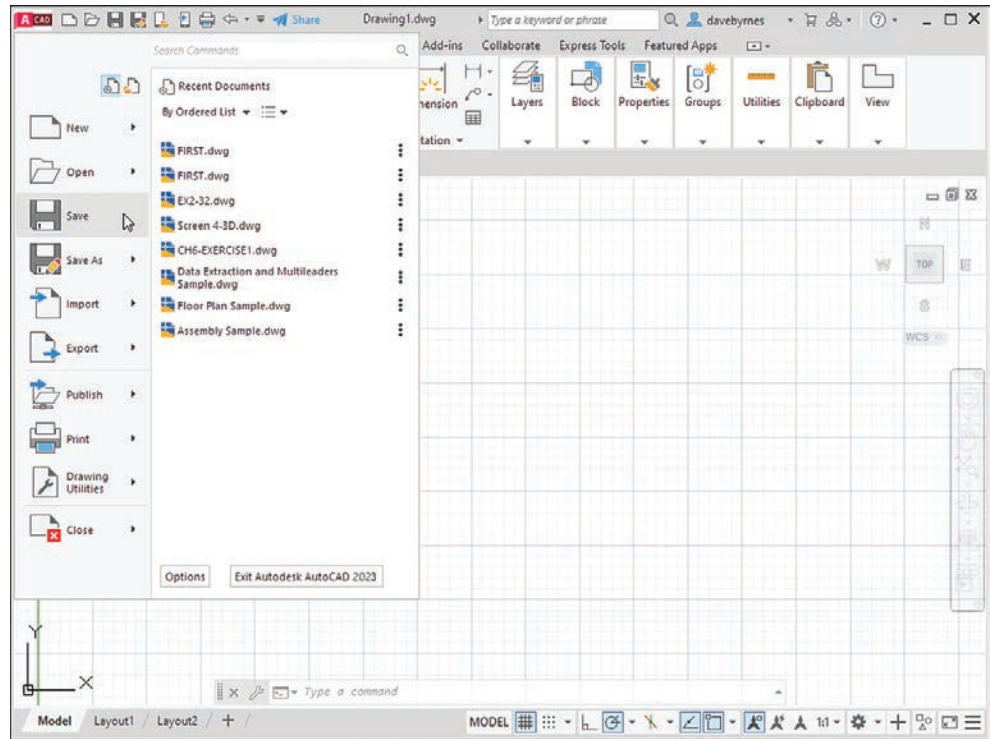
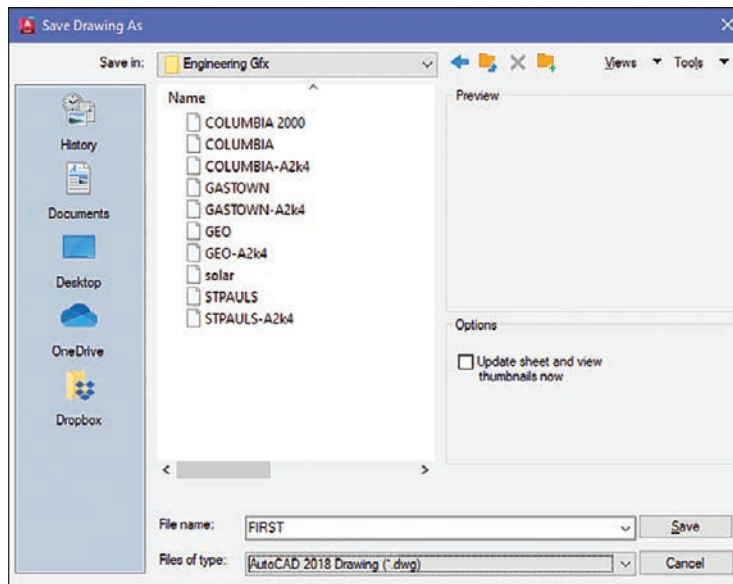


Figure 1-16



It's a good idea to save your work frequently. AutoCAD can be configured to save your drawings automatically, but it's a much better process to actively save your work. After you've created your drawing file in the **Save Drawing As** dialog box, using the **Save** command creates a backup version (*filename.bak*) and updates your saved file.

To save your work after you've given it a name, click the **Save** button in the **Quick Access Toolbar** at the top of the screen, or you can use the standard Windows shortcut: **Ctrl+S**.

- 1** Click the large **Application Menu** button in the upper-left corner of the screen.
- 2** Click **Save**.

Since you have not yet named and saved this drawing file, the **Save Drawing As** dialog box appears (Figure 1-16).

The **Save Drawing As** dialog box lists all existing drawings. Click on the thumbnail option to change the list to thumbnail drawings.

3 Enter the drawing name.

In this example, the drawing name **FIRST** was used.

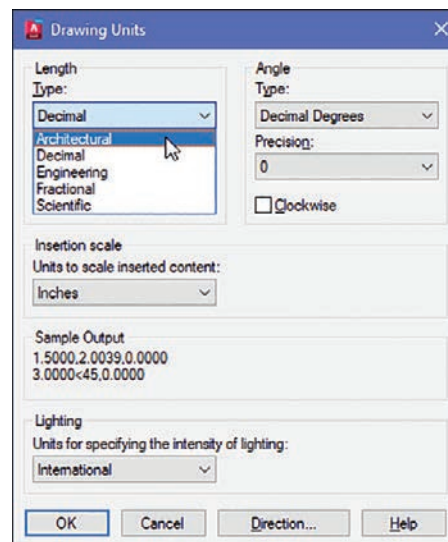
4 Click **Save**.

The name of the drawing appears at the top of the screen.

1-7 Drawing Units

AutoCAD 2023's **Drawing Units** dialog box allows for either English or metric units to be used as default values; however, AutoCAD can work in any of five different unit systems: scientific, decimal, engineering, architectural, or fractional. The default system is the decimal system, and it is used with either English values (inches) or metric values (millimeters). See Figure 1-17.

Figure 1-17



Access the Drawing Units dialog box by first opening the **Application** menu and then selecting **Drawing Utilities**.

Specifying or Changing the Drawing Units

1 Select **Drawing Utilities** in the **Application** menu.

2 Select **Units** (Figure 1-18).

The **Drawing Units** dialog box appears (Figure 1-19).

Figure 1-18

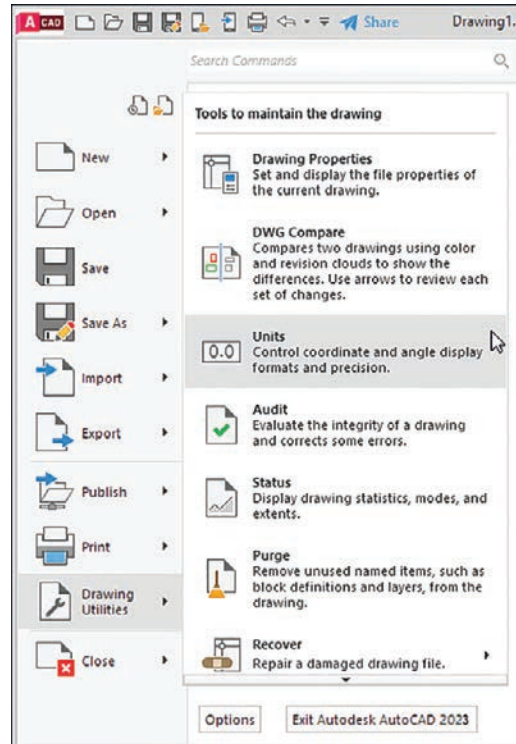
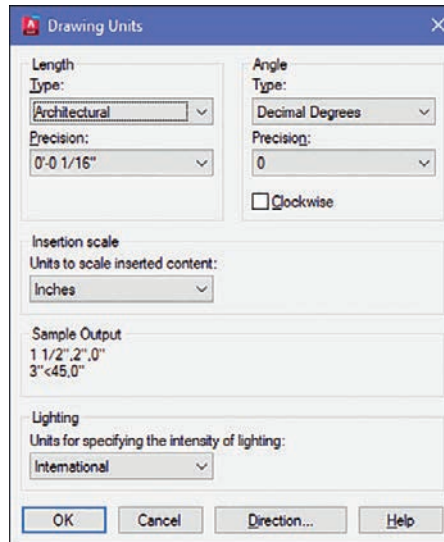


Figure 1-19



- 3 In the **Length** area, select architectural units by clicking the arrow to the right of the **Type** text box.

A list of the five unit options cascades down.

- 4 Select **Architectural**.

Note that the **Sample Output** section, located slightly below the center of the **Drawing Units** dialog box, shows fractional inches.

- 5 Repeat the procedure and set the drawing units back to **Decimal**.

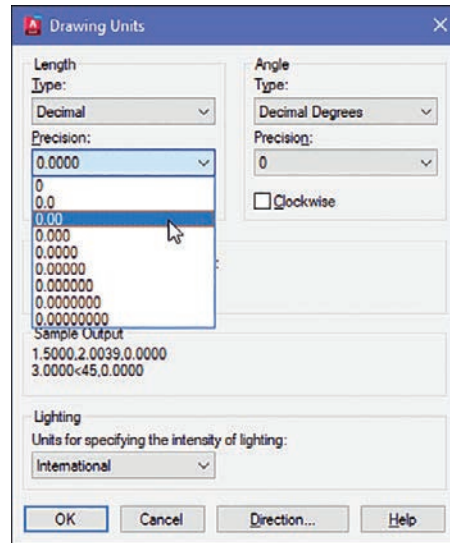
Specifying or Changing the Precision of the Units System

Unit values can be expressed with decimal places from zero to eight or in inches from 0 to 1/256 inch.

- 1 Access the **Drawing Units** dialog box as explained previously.
- 2 In the **Length** area, click the arrow to the right of the current precision value display box below the word **Precision**.

A drop-down list of the possible decimal precision values cascades from the box (Figure 1-20).

Figure 1-20



- 3 Select **0.00**.

The value 0.00 appears in the **Precision** box.

- 4 Click **OK**.

The original drawing screen appears.

Specifying or Changing the Angle Units Value

You can specify angles in one of five different units: **Decimal Degrees**, **Degrees/Minutes/Seconds**, **Gradians**, **Radians**, or **Surveyor** units. **Decimal Degrees** is the default value.

Change the angle units in the **Angle** area by selecting the desired units in the drop-down menu under **Type**. The precision of the angle units is changed in the same way as for linear units.

1-8 Drawing Limits

You can use **drawing limits** to set the boundaries of a drawing. The drawing boundaries are usually set to match the size of a sheet of drawing paper. This means that when the drawing is plotted and a hard copy is made, it will fit on the drawing paper.

Figure 1-21 shows a list of standard flat-size drawing sheets for engineering applications, Figure 1-22 shows standard metric sizes, and Figure 1-23 shows standard architectural sizes.

Figure 1-21

Standard Drawing Sheet Sizes—Inches	
A	8.5 × 11
B	11 × 17
C	17 × 22
D	22 × 34
E	34 × 44

Figure 1-22

Standard Drawing Sheet Sizes—Millimeters	
A4	210 × 297
A3	297 × 420
A2	420 × 594
A1	594 × 841
A0	841 × 1189

Figure 1-23

Standard Drawing Sheet Sizes—Architectural USA	
A	9 × 12
B	12 × 18
C	18 × 24
D	24 × 36
E	36 × 48

A standard 8.5" × 11" letter-size sheet of paper as used by most printers is referred to as an *A-size* sheet of drawing paper.

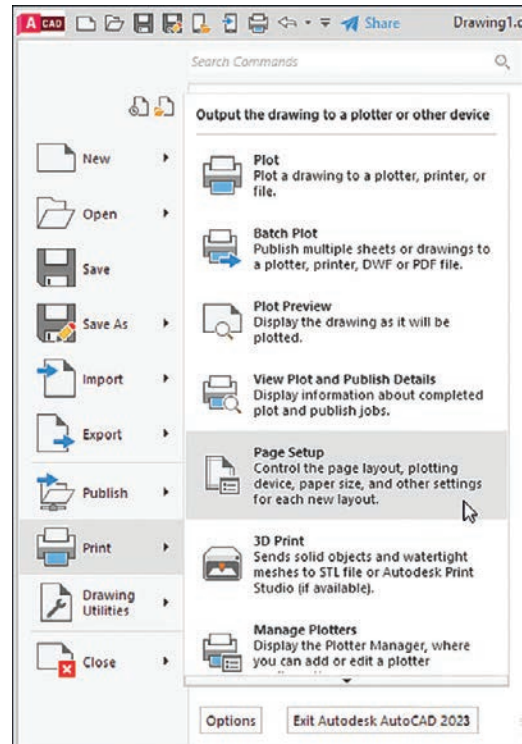
NOTE

A sheet of paper can be sized to match standard sheet sizes by the capabilities of the printer or plotter. Many printers and plotters have built-in scaling features, and some list standard sheet sizes that can be applied to a drawing.

Aligning the Drawing Limits with a Standard A3 (Metric) Paper Size

- 1 Click the **Application Menu** button in the upper-left corner of the screen.
- 2 Click **Print**, then click **Page Setup** (Figure 1-24).
The **Page Setup Manager** dialog box appears.

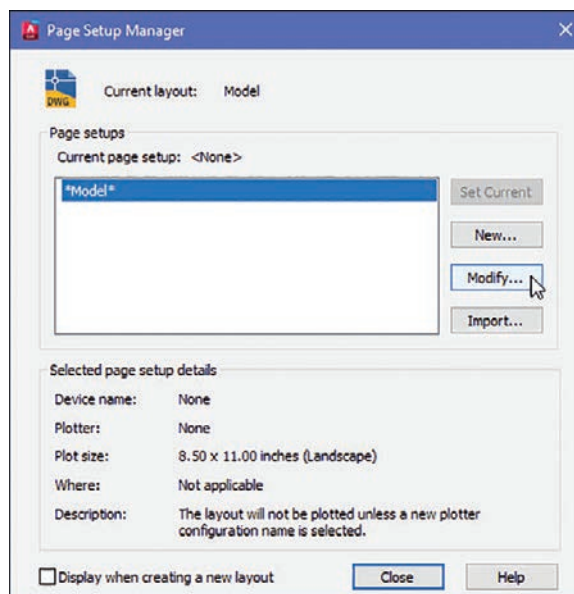
Figure 1-24



- 3 Click **Modify...** (Figure 1-25).

The **Page Setup - Model** dialog box appears.

Figure 1-25



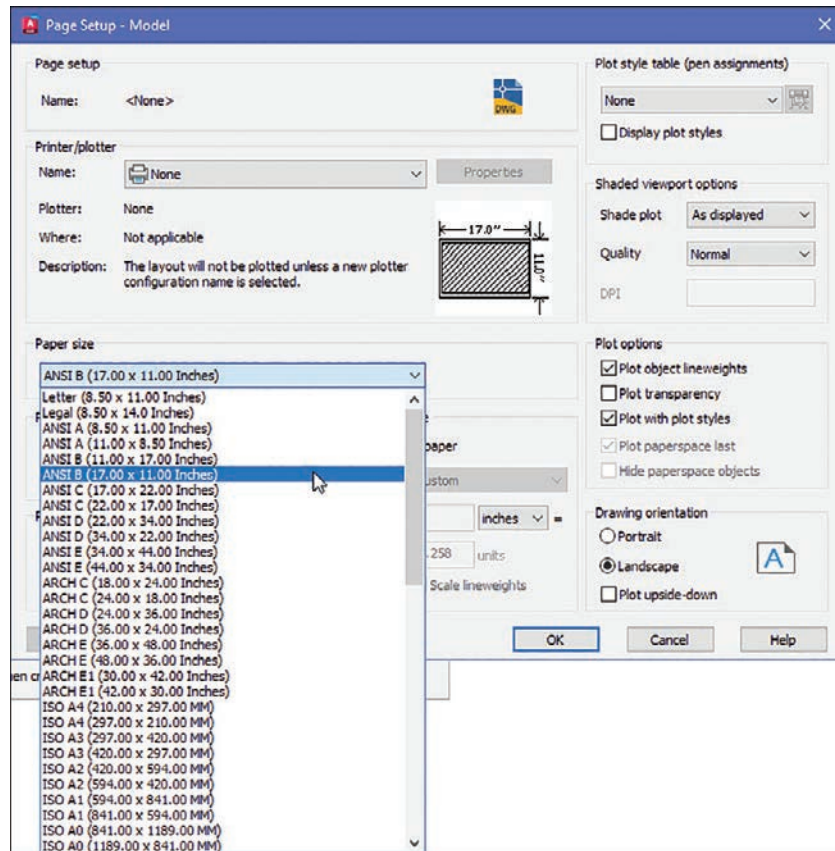
- 4 Click the arrow to the right of the **Paper size** box.

A drop-down list of available paper sizes appears (Figure 1-26).

- 5 Select the **ISO A3 (420.00 x 297.00)** size.

The dimensions in the preview box in the **Printer/plotter** area of the **Page Setup-Model** dialog box change to the selected values.

Figure 1-26



- 6 Click **OK**.

The drawing screen is now sized to the 420.00 × 297.00 ISO A3 dimensions.

NOTE

The sheet size may also be set with the **Limits** command. Type **Limits** at a command prompt and define the drawing limits by specifying the lower-left corner of the drawing as **0.00,0.00** (which is the default setting) and the upper-right corner as needed. If the new limits exceed the current screen limits, type **zoom** at a command prompt, then type **a** for **Zoom All**. The new drawing limits are matched to the screen size. The default sheet size for the **acad** template is 8.5 × 11 (ANSI A), and for the **acadiso** template the default is 210 × 297 (ISO A4).

1-9 Grid and Snap

The **Grid** command is used to place a grid background on the drawing screen. This background grid is helpful for establishing visual reference points for sizing and for locating points and lines. The grid may appear as lines or dots. You can specify the type of grid in the **Drafting Settings** dialog box.

NOTE

A graph paper-style grid background is used in most figures in this book.

The **Snap** command limits the movement of the cursor to predefined points on the screen. For example, if the **Snap** command values are set to match the **Grid** values, the cursor will snap from intersection to intersection (or dot to dot) on the grid.

The default **Grid** and **Snap** setting for the **acad** template is **.50** inch, and the default setting for **Grid** and **Snap** for the **acadiso** template is **10** millimeters.

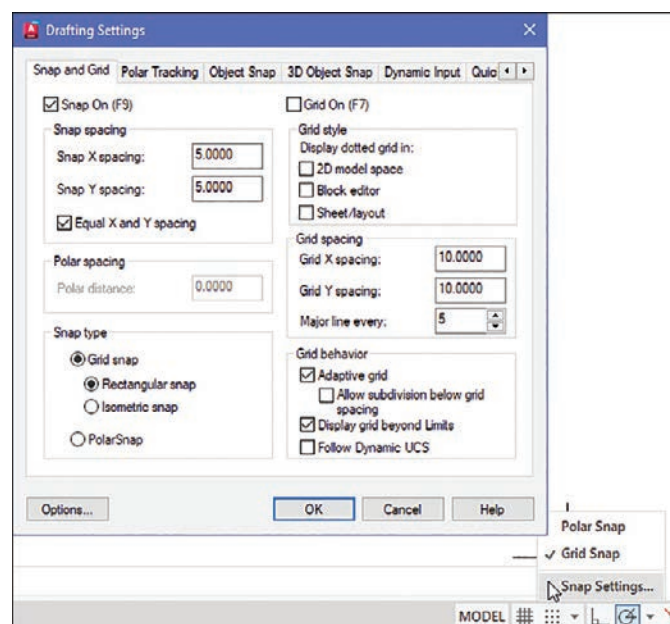
NOTE

The **Grid** function can be toggled off and on with the **F7** key, and the **Snap** function can be toggled with the **F9** key.

Setting the Grid and Snap Values

- 1 Start a new drawing and select the **acadiso** template (where values are in millimeters).
- 2 Right-click the **Snap** tool located at the bottom of the screen and click **Snap Settings** (Figure 1-27).

Figure 1-27



The **Drafting Settings** dialog box appears (Figure 1-27). If it is not already selected, click the **Snap and Grid** tab.

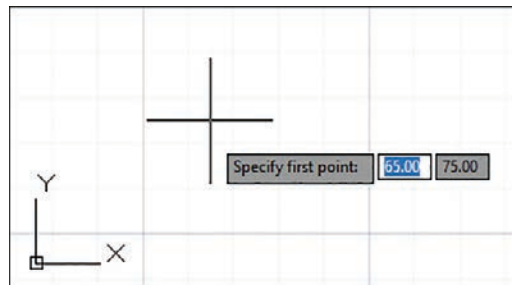
- 3** Click the **Grid On** and **Snap On** checkboxes. A check mark appears in each of the boxes.
- 4** Place the cursor in the **Snap X spacing** text box to the right of the given value under the **Snap On** heading. A vertical flashing cursor appears.
- 5** Backspace out the existing value and type in **5**.
- 6** Click the **Snap Y spacing** box.

The Y spacing automatically equals the X spacing value. You can create rectangular grid spacing by specifying different X and Y spacing values.

- 7** Select the **Grid X spacing** text box under the **Grid spacing** heading.
- 8** Backspace out the existing value and type in **10** if needed.
- 9** Click the **Grid Y spacing** box to make the X and Y values equal.
- 10** Click **OK**.

Figure 1-28 shows the result. Since the **Snap** values have been set to exactly half of the **Grid** values, the cursor can be located either directly on grid intersections or halfway between them.

Figure 1-28



You can turn the grid on and off either by double-clicking the **Grid** icon at the bottom of the screen or by pressing the **F7** key on the keyboard. Turn Snap on and off by double-clicking the **Snap** icon at the bottom of the screen or by pressing the **F9** key on the keyboard. You can also turn **Grid** and **Snap** off and on by clicking their respective buttons on the status bar located at the bottom of the screen.

1-10 Drawing Problem

Set up a drawing that uses millimeter dimensions and the following parameters:

Sheet size = **297,420 (A3)**

Grid = **10** spacing

Snap = **5** spacing

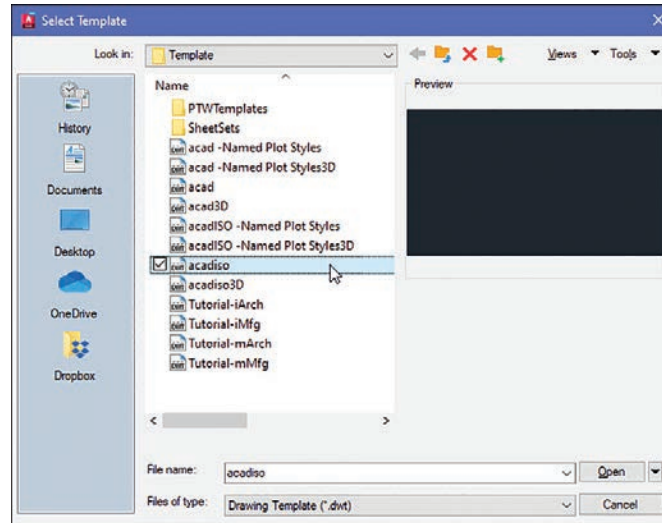
Whole-number precision

Specifying the Drawing Units

- 1 Click the **Application Menu** button in the upper-left corner of the drawing screen and then select **New**, then **Drawing**.

The **Select Template** dialog box appears (Figure 1-29).

Figure 1-29



- 2 Select the **acadiso** template and click **Open**.

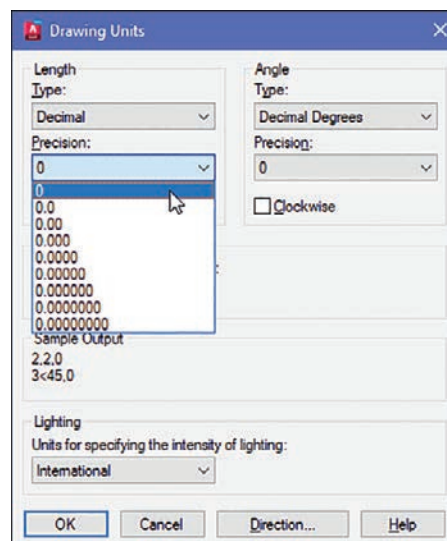
Defining the Units Precision

- 1 Click the **Application Menu** button in the upper-left corner of the drawing screen and then select **Drawing Utilities**, then **Units**.

The **Drawing Units** dialog box appears (Figure 1-30). In this example, only whole numbers will be used, so the **0** option is selected.

- 2 Select the **0** precision and click **OK**.

Figure 1-30



Setting the Sheet Size

The default values for an **acadiso** template are 210 × 297, but this drawing problem calls for 297 × 420, an A3 sheet size.

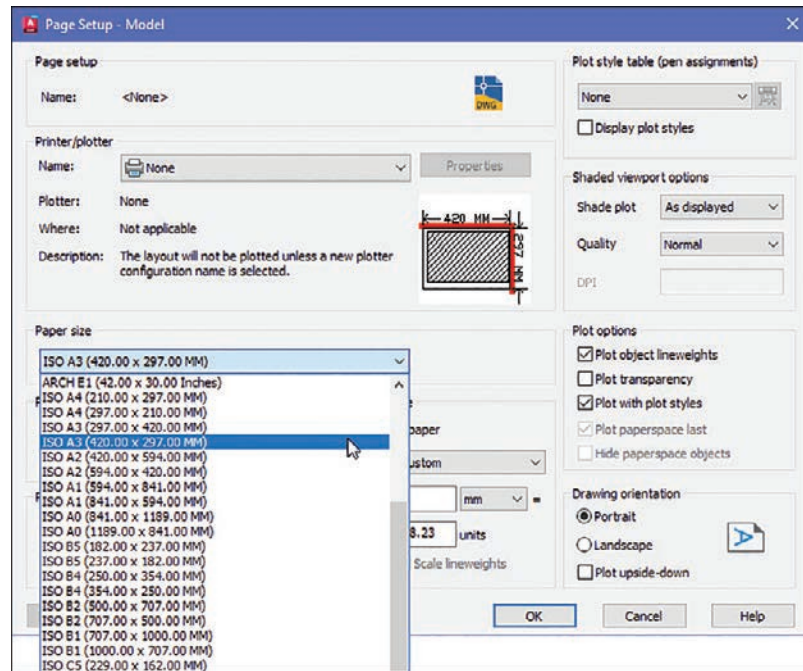
- 1 Open the **Application** menu and select **Print**, then **Page Setup**.

The **Page Setup Manager** dialog box appears.

- 2 Click **Modify**.

The **Page Setup - Model** dialog box appears (Figure 1-31).

Figure 1-31



- 3 Scroll down the available **Paper size** options and select the **ISO A3 (420.00 x 297.00)** option.

- 4 Click **OK**.

Setting Grid and Snap Values

- 1 Right-click the **Grid** button at the bottom of the screen.

- 2 Click **Grid Settings**.

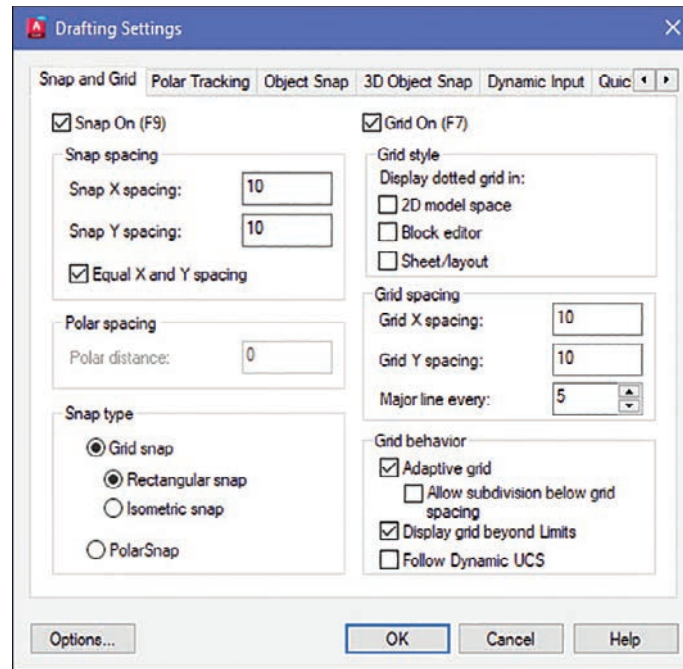
The **Drafting Settings** dialog box appears (Figure 1-32).

- 3 Select **Grid On** and **Snap On** and set the snap spacing to **5** and the grid spacing to **10**.

- 4 Enter **Zoom** at the command prompt, type **A**, and press **Enter**.

The screen is now ready for starting a drawing using millimeter values.

Figure 1-32



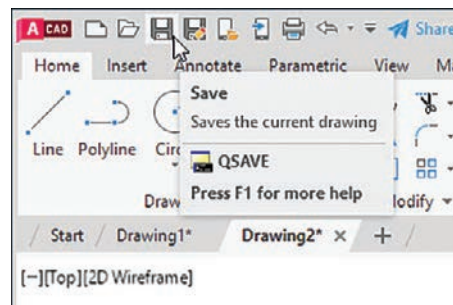
1-11 Save and Save As

Use the **Save** command to save your work. If you start a new drawing, the first time you click **Save**, the **Save As** command displays the **Save Drawing As** dialog box. You can select **Save As** at any time if you want to save your drawing using a different name or in a different location, but most of the time you will use the **Save** command to simply save your work.

Using the Save and Save As Commands

- 1 Click the **Save** button on the **Quick Access Toolbar** at the top of the screen, above the Home panel (Figure 1-33).

Figure 1-33

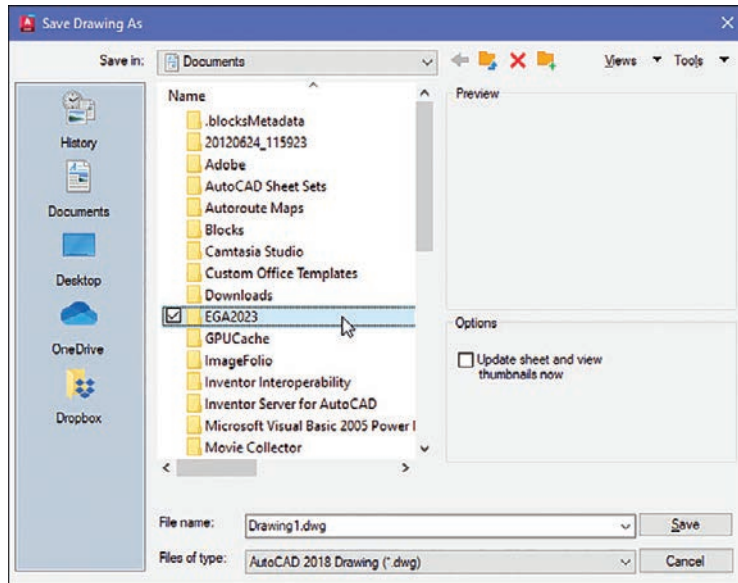


NOTE

The small group of tool buttons to the right of the **Application Menu** button and above the ribbon is called the **Quick Access Toolbar**. You will find frequently used commands here, including New, Open, Save, and Print. You can customize the Quick Access Toolbar to add your own frequently used commands.

Because this drawing has not yet been saved, the **Save Drawing As** dialog box appears (Figure 1-34). In this example, the file name **Drawing1.dwg** appears. This is the default name that was created automatically when the new drawing was opened.

Figure 1-34

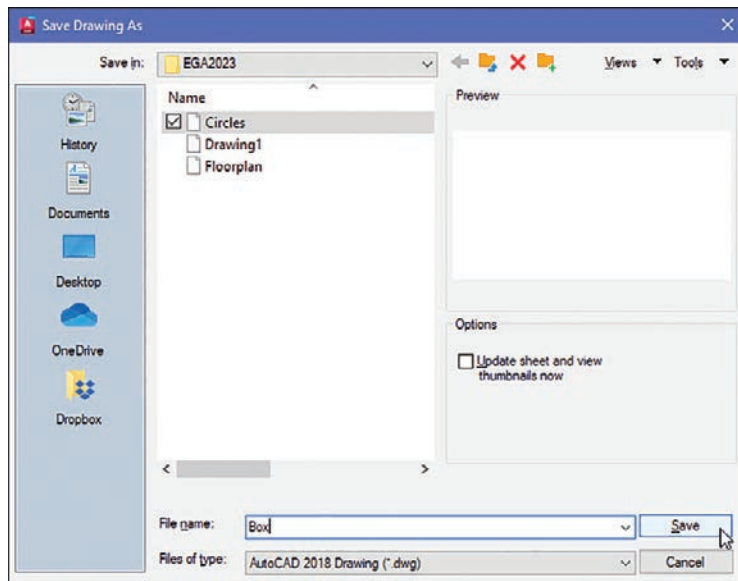


2 Create a new folder where you can save your work.

Figure 1-34 shows a folder named **EGA2023** created under **Documents**.

3 Save the drawing in the folder **EGA2023** located in the **Documents** folder and enter the name **Box** (Figure 1-35).

Figure 1-35



NOTE

The **Open** option can also be accessed by pressing **Ctrl+O**.

- 4 Click **Save**.

1-12 Open

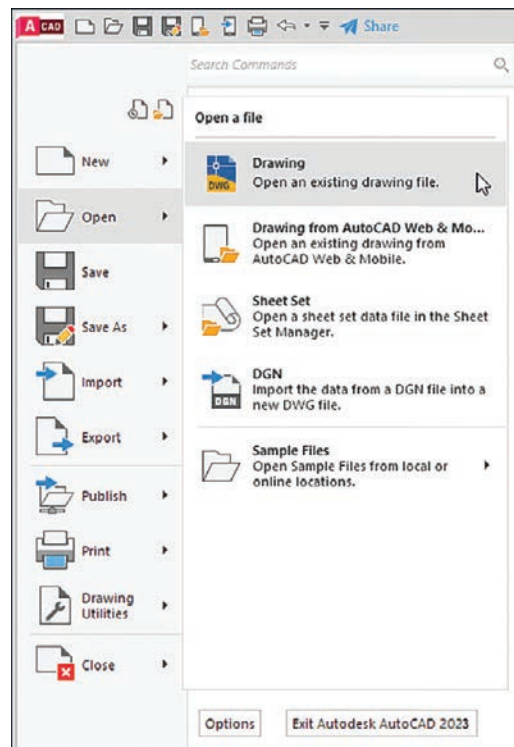
Use the **Open** command to call up an existing drawing so that you can continue working on it.

Using Open

Access existing drawings using **Open** and the **Select File** dialog box.

- 1 On the **Application** menu, click **Open** (Figure 1-36), then click **Drawing**.

Figure 1-36



The **Select File** dialog box appears (Figure 1-37).

- 2 Click the **Views** option at the top of the **Select File** dialog box and click the **Thumbnails** and **Preview** options.

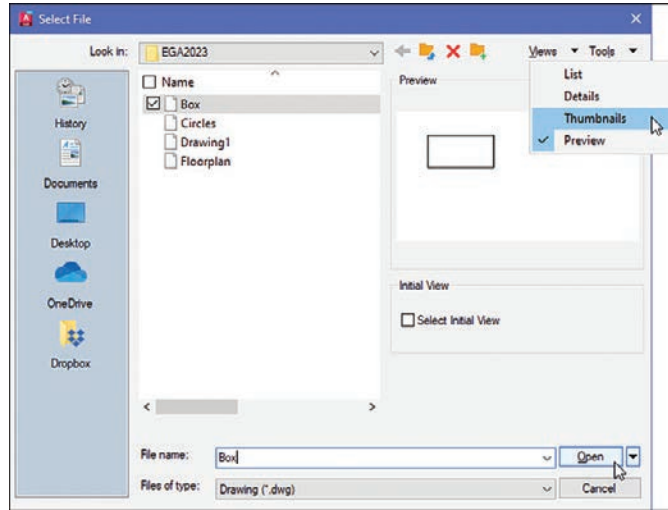
Thumbnails of the drawing files appear.

- 3 Click the desired file.

A preview appears.

- 4 Click **Open**.

Figure 1-37



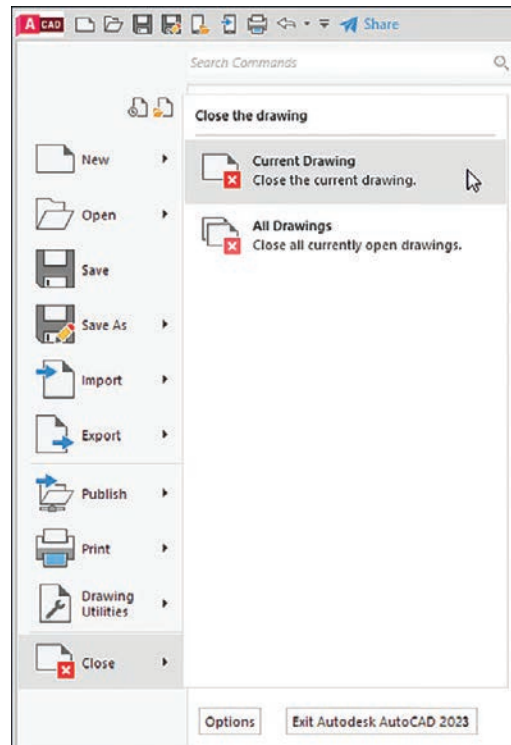
1-13 Close

The **Close** command allows you to close the current drawing.

- 1 On the **Application** menu, click **Close** and then click **Current Drawing** (Figure 1-38).

The system exits the AutoCAD program.

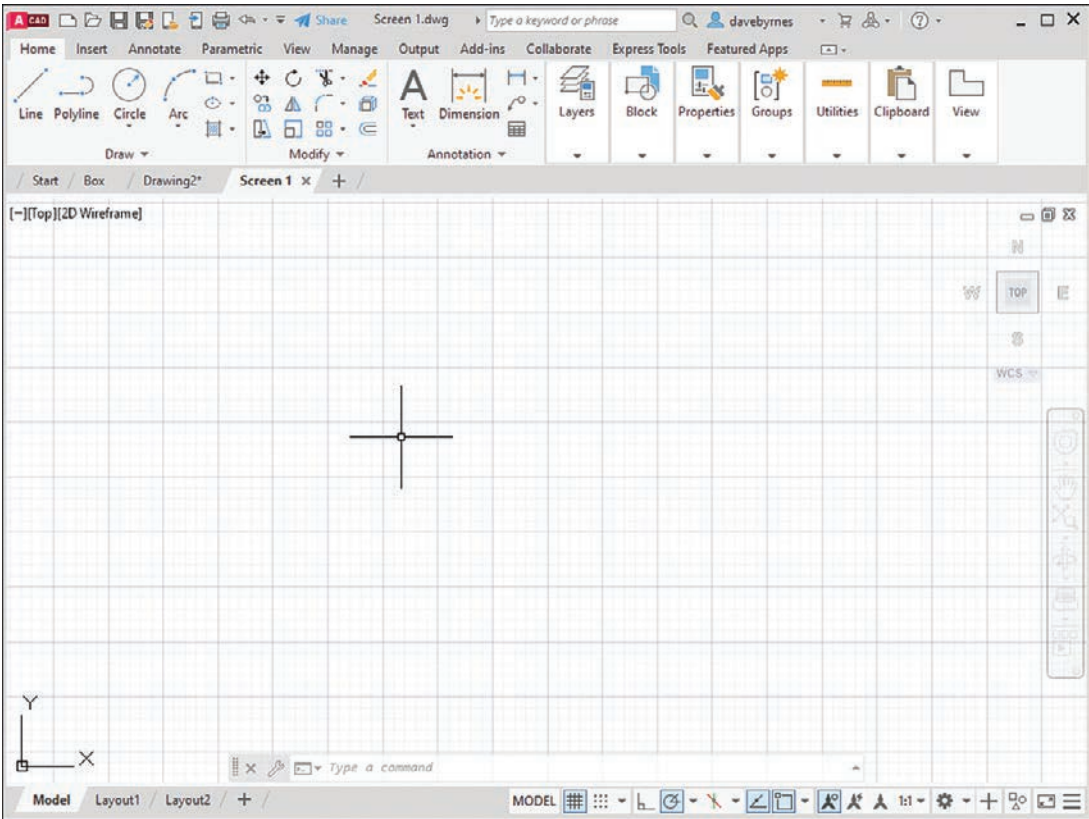
Figure 1-38



1-14 EXERCISE PROBLEMS

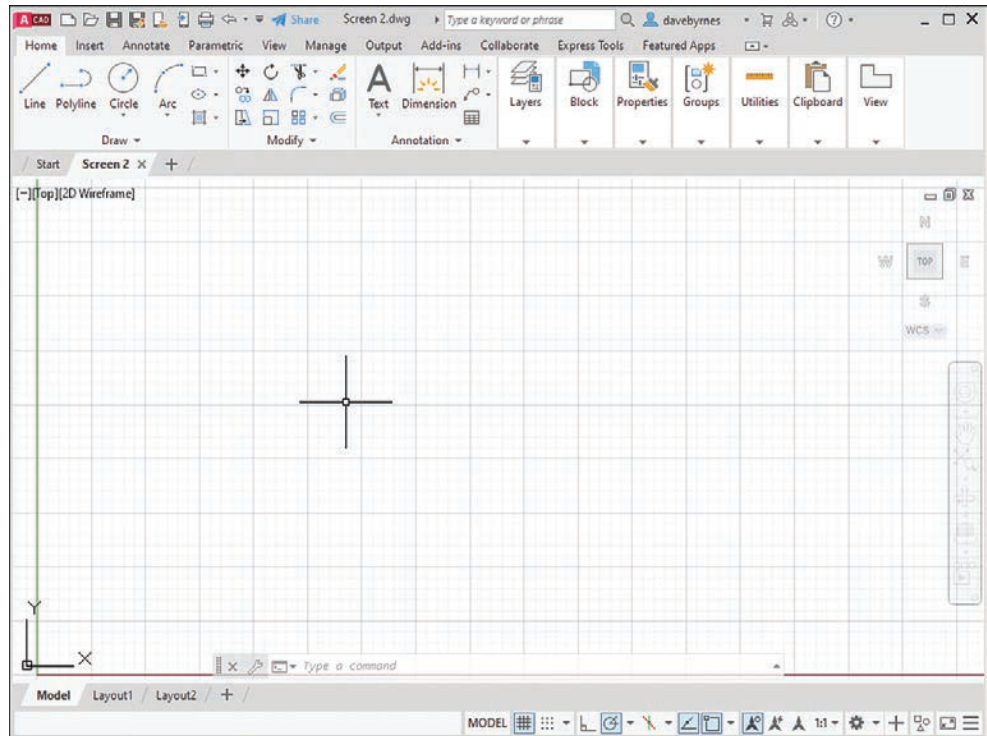
EX1-1

Create a drawing screen as shown in Figure EX1-1. Select the **acadiso** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **10** and snap spacing to **5**. Set the sheet size to **ISO A3 (297.00 x 420.00)**. Name the drawing **Screen 1**.



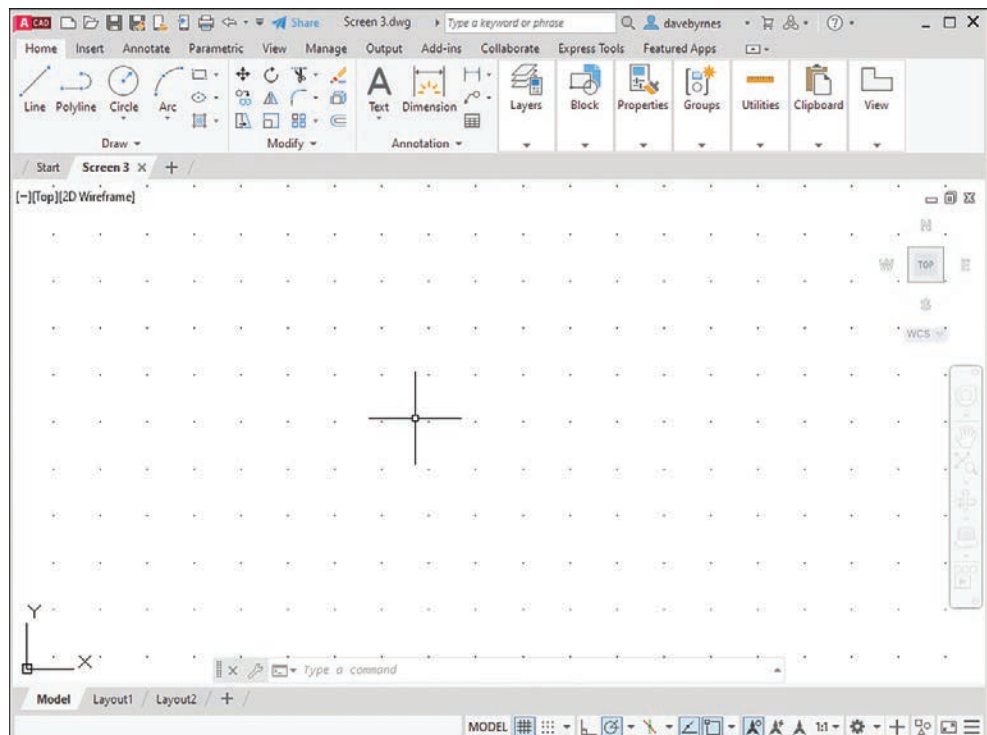
EX1-2

Create a drawing screen as shown in Figure EX1-2. Select the **acad** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **0.50** and snap spacing to **0.25**. Locate the origin in the lower-left corner of the drawing screen. Name the drawing **Screen 2**.



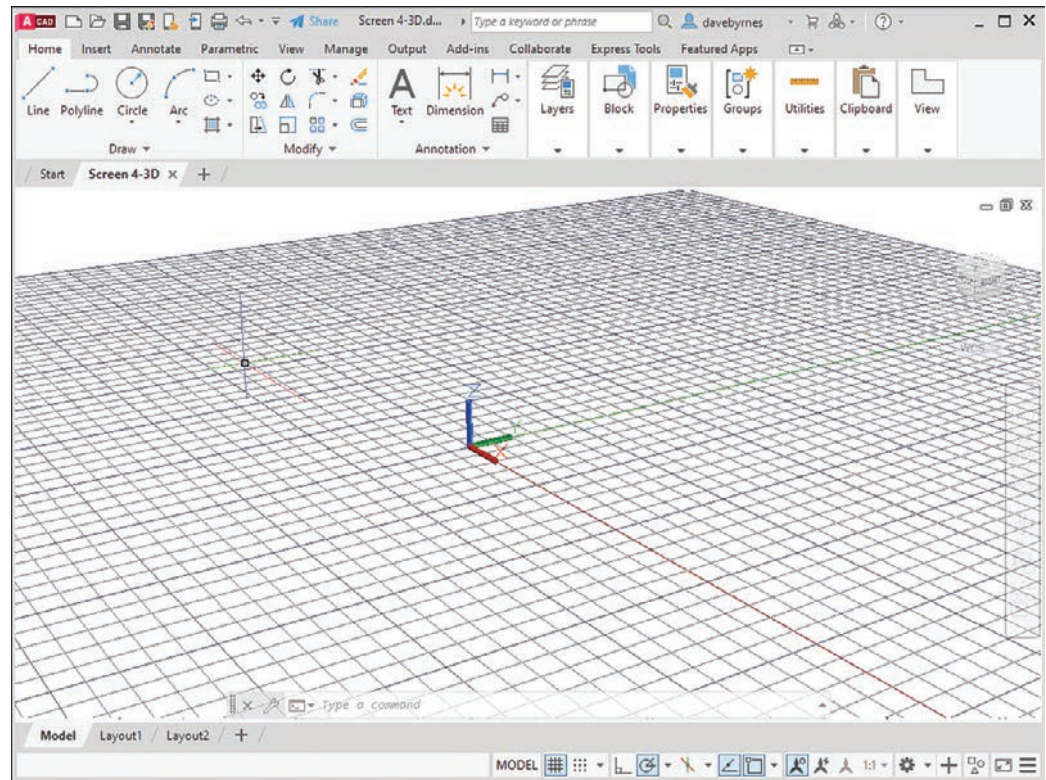
EX1-3

Create a drawing screen as shown in Figure EX1-3. Select the **acadiso** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **50** and snap spacing to **10**. Set the grid background to **dotted**. Name the drawing **Screen 3**.



EX1-4

Create a drawing screen as shown in Figure EX1-4. Select the **acadiso3D** template, turn on the **Grid** and **Snap** functions, and set the grid spacing to **20** and snap spacing to **5**. Name the drawing **Screen 4-3D**.



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Index

Symbols

* (asterisk), 8

° (degree symbol), 61

\$ (dollar sign), 8

- (hyphen), 8

% (percent sign), 8

± (plus/minus symbol), 61

_ (underscore), 8

2D construction

arrays, 65–68

Path Array option, 67–68

Polar Array option, 67

rectangular, 66

break points in, 71–72

chamfers, 72–73, 412–413

circles, drawing, 42–44

center marks, 45–46

centerlines, 400–401

circular bases, 709–710, 712

diameter, 6–8, 43

in L-shaped bracket drawing, 689

with object snap. *See* object snap modes (Osnap)

quadrant-sensitive applications, 44

radius, 42

sketches, 173–174, 180

tangent tangent radius, 44

three points, 43

two points, 43

closed areas, creating, 32

copying objects in

Copy command (Modify panel), 63

multiple copies, 64

dimensioning, 415

ellipses, 51–53, 348–349

axis endpoint, 51–52

center points of, 52–53

defining with Osnap Intersection mode, 94

drawing, 51–53

elliptical arcs, 52–53

elliptical bases, 710–712

in L-shaped bracket model, 690

as representations of projected holes, 223

sketching, 173–174

exercise problems, 78–88

extending objects in, 70

fillets, 73–74

definition of, 234–235

dimensioning, 409

lines. *See* lines

mirroring objects in, 65

moving objects in, 62–63

offset distance, 64–65

overview of, 29

points, 56–57

hyperbolas, 141–143

parabolas, 141

randomly selecting, 29–31

shape of, 57

size of, 57

polygons

center points, 55–56

definition of, 55–56

drawing, 55–56

edge distance, 56

polylines, 46–50

in castings model, 738

creating from basic shapes, 738

creating from line segments, 717–718

creating splined polylines from, 135

drawing, 47

extruding, 716–717

line thickness, 50

polyline arcs, 48–50

verifying as single entity, 47

rectangles, 54

drawing, 54

exploding, 54

in L-shaped bracket drawing, 689

rotating objects in, 68–69

splines, 50–51

tables, 74–77

text, 57–61

dimension, 372–374, 404

editing, 137

height/placement, 378–380

multiline, 57–60

trimming objects in, 69–70

2D WCS (world coordinate system), returning to, 675

2D Wireframe visual style, 684, 688

2H pencil lead, 170

2-Point option (Circle tool), 43

A3 (metric) paper size, aligning drawing limits with, 15–16

3 Point tool (Coordinates panel), 681, 689, 697

3D modeling. *See also dimensions; solid modeling*
 3D Modeling workspace, 676–677
 3D Modeling mode, accessing, 676–677
 selecting, 676
 drawing problems, 688
 orthographic views
 of 3D objects, 690–692
 creating, 692
 overview of, 669
 perspective and parallel grids, 673–675
 Thickness system variable, 693–698
 creating objects with, 696–698
 drawing curves with, 695–696
 drawing hexagons/arcs with, 693–694
 exercise problems, 699–703
 overview of, 693
 UCSs (user coordinate systems), 677–682
 creating, 679
 defining, 681–682
 definition of, 677
 drawing problems, 686–687, 688–690
 restoring, 680–681
 returning to WCS (world coordinate system), 680
 saving, 679–680
 solid boxes, drawing, 677–678
 UCS axis, rotating, 683–684
 visual errors, 688
 visual styles, changing, 683–684
 WCS (world coordinate system)
 background color, 671
 default settings for, 669–670
 returning to 2D WCS, 675, 680
 returning to original orientation, 673
 viewpoint, 672

3D Modeling mode, accessing, 676–677
3D Modeling workspace. *See 3D modeling*
3D Object Snap tab (Drafting Settings), 689
3D Vertex snap option, 689
3H pencil lead, 170
3-Point option (Circle tool), 43

A

abbreviations, dimensioning, 414
acad template, 2–5, 25, 670
acad3D template, 669. *See also 3D modeling*
 3D Modeling mode, accessing, 676–677
 perspective and parallel grids, 673–675
acadiso template, 19, 25, 212
acadiso3D template, 27, 669, 673–675
acceleration, uniform, 654–655

access controller model, 772
Access to Help tool, 6
acme threads
 definition of, 544–545
 external, 546–548
addendum (a), 627
Advanced Attribute Editor dialog box, 131
AGMA (American Gear Manufacturers Association), 629
aligned dimensions, 384–385
 creating, 384–385
 definition of, 384
 text options, 385
aligned sections, 300
Aligned tool (Dimensions panel), 384–385
 creating aligned dimensions with, 384–385
 overview of, 384
 text options, 385
American Gear Manufacturers Association (AGMA), 629
American National Standards Institute (ANSI), 247, 629
American National threads, 544–545
American Society of Mechanical Engineers, 247
Ang option (Construction Line command), 40–41
angle bisectors, constructing, 138–139
angles, 294
 angle units value, 13
 angular dimensions, 369, 391–393
 creating, 392
 overdimensioning, avoiding, 393
 angular lines, 40–41
 bisecting, 138–139
 dihedral, 340
angular tolerances, 448–449, 504–505
 creating, 449
 precision of, 449
Angular tool (Dimensions panel), 391–393
 creating angular dimensions with, 392
 overdimensioning and, 393
Annotate panel, 5. *See also Dimensions panel*
 Multileader tool, 287
 Multiline Text tool, 57–61, 584
 creating multiline text, 57–58
 text font, 59–60
 text height, 59
 text justification, 60–61
 Table tool, 74–77, 589
 Text tool, 583
ANSI (American National Standards Institute), 247, 629

ANSI37 pattern, 413

aperture size, Object Snap, 91

Apparent Intersect object snap, 94–95

Application menu commands, 2

Close, 23–24

Drawing Utilities, 11–12, 19

New, 2, 3–5, 8, 582

Open, 23

Print, 15

Save, 10, 21–22

approval signatures, 588–589

architectural unit system, 11–12

arcs

drawing with thickness, 693–694

elliptical, 52–53

of ogee curves, 139–140

polyline, 48–50

Array command, 629, 630, 632, 742

cam displacement diagrams, 651

harmonic cam motion, 653

Rectangular, 541, 545

threads, drawing

 detailed thread representations, 539

 external acme threads, 547

 external square threads, 545

 schematic thread representations, 541

uniform acceleration/deceleration, 654

arrays, 65–68

Path Array option, 67–68

Polar Array option, 67

rectangular, 66

arrowheads

drawing, 288–289

resizing, 289

assembly drawings

assembly numbers, 580–581

BOM (bill of materials), 589–591

design layouts, 593–598

 copying, 596

 creating, 594–595

 creating drawings from, 597–598

 definition of, 593

drawing problems, 593–602

 assembly drawing examples, 599–602

 design layouts, 594–595

 layers, 596–597

exercise problems, 603–624

families of drawings, 580

formats for, 582–584

overview of, 579–581

release block, 588–589

revision block, 586–587

revisions, 587

scale of, 586

templates, adding, 582–584

title block, 584–586

 attributes of, 132–134

 company name, 586

 drawing numbers, 586

 drawing titles, 585

 overview of, 584–585

 release date, 586

 scale, 586

 sheet number, 586

tolerance block, 588

asterisk (*), 8

Attribute Definition dialog box, 126–128, 132

attributes, block, 126–132

adding, 126–128

creating new blocks with, 129

editing, 131–132

inserting existing blocks with, 130

title blocks with, 132–134

automobiles, rack-and-pinion steering for, 645

auxiliary views

axis system, rotating, 330–331

definition of, 327

drawing problems, 331–332

 irregular surfaces, 337–340

 lines, transferring between views, 334–335

 rounded surfaces, 335–337

 secondary views, 346–348

exercise problems, 350–366

irregular surfaces, 337–340

lines, transferring between views, 332–335

 drawing problem, 334–335

 grip points, 333–334

 line length measurement, 332–333

 line rotation, 334

oblique surfaces, 342–343

overview of, 327–328

partial views, 340–341

projecting information from normal view into,
328–331

rounded surfaces, 335–337

secondary views, 343–349

 creating, 345–346

 drawing problem, 346–348

 of an ellipse, 348–349

 overview of, 343–344

sectional views, 341–342

slanted surfaces, 342

axis endpoint, ellipses, 51–52

axis system, rotating, 330–331

B

background color, changing, 671

backlash, 627

ball bearings

drawing problems, 646–647

overview of, 645–646

baseline dimensions, 396–397, 451–453

Baseline tool (Dimensions panel), 396–397, 452–453

bearings

ball bearings, 645–646

drawing problems, 646–647

overview of, 645–646

bushings

drawing problems, 647–650

overview of, 647–648

bevel gears

definition of, 637

drawing, 639–641

face angle lines, 640

matched sets of, 640–641

terminology for, 637–639

bilateral tolerance, 441

bill of materials (BOM), 589–591

bisectors of angles, constructing, 138–139

Block command, 535, 629

Block Definition panel, 106

Create command, 107–108

Define Attributes tool, 126–128

Block Editor, 548

Block panel, 106, 107–108, 109, 110, 126, 567

blocks

attributes, 126–132

adding, 126–128

creating new blocks with, 129

editing, 131–132

inserting existing blocks with, 130

title blocks with, 132–134

changing shape of, 113–114

combining, 114

creating, 107–108

creating new drawing files from, 115–117

definition of, 106

exercise problems, 149–167

exploding, 109

inserting, 107–108, 110–113

at different angles, 110–112

with different scale factors, 112–113

scaling, 109

title blocks, attributes of, 132–134

Tool Palettes

changing scale of, 567–568

modifying, 568

Blocks palette, 108, 110–111, 113–114, 116, 567–568

bolts

drawing problems, 558–559

head shapes

hexagon, 550–552

square, 552–554

overview of, 548

BOM (bill of materials), 589–591

Boolean operations, 722

subtraction, 723–724

unions, 722–723

Boolean panel, Union tool, 687

bosses, 235

boundaries, setting, 13–16

aligning with standard A3 (metric) paper size, 15–16

drawing problem, 20

options for, 13–14

Box tool, 683, 686, 706–709. See also boxes

boxes

drawing from given dimensions, 707–708

drawing with corner option, 706

cubes, drawing, 708

dynamic grips, 708–709

visual styles, 706–707

boxes, 706–709

in combined solid objects drawing, 727–728, 730

creating with thickness, 696

cubes, 708

drawing from given dimensions, 707–708

drawing with corner option, 706

dynamic grips, 708–709

spheres, 709

three-dimensional, 202–203

visual styles, 706–707

Break at Point command (Modify panel), 71–72

Break command (Modify panel), 71–72

breaks, 303–305

break points, 71–72

freehand lines, 303

S-breaks, 303–305

broken-out sectional views, 302

bronze bushings, 647

Browse templates command, 2

bushings

drawing problems, 647–650

overview of, 647–648

C

callouts, thread, 536–537

- English units, 536–537
- metric units, 536

cam displacement diagrams

- axes in, 650
- cam followers
 - drawing problems, 655–657
 - exercise problems, 658–659
 - types of, 655
- cam motions, 652–655
 - harmonic motion, 653–654
 - modified uniform motion, 652–653
 - uniform acceleration/deceleration, 654–655
 - uniform motion, 652
- dwells in, 651
- overview of, 650–651, 667
- setting up, 651–652

castings, 737–742

- holes, creating, 740
- orthographic views, 234–238
 - definition of, 234–235
 - drawing problems, 237–238
- overview of, 737
- polylines, creating, 738
- rounded edges, creating, 741–742
- sectional views, 305
- shapes
 - drawing, 737–738
 - extruding, 739

center distance (CD), 627, 635

Center Mark tool, 45–46, 400–401, 630, 632

center marks, 45–46, 400–401

- creating, 45–46
- diameter dimensions, 389
- removing from radius dimensions, 387

Center object snap, 95–96

center points

- of ellipses, 52–53
- polygons, 55–56
- wedges, 713–714

centerlines, adding linear dimensions to, 389–390

Centerlines panel, Center Mark tool, 45–46

chain dimensions, 451–453

Chamfer command (Modify panel), 72–73, 540, 743

chamfers, 72–73, 412–413

change orders (COs), 587

chisels, grinding wheel for, 615

Circle Properties dialog box, 124

Circle tool (Draw panel), 6–8, 42–44, 93, 693.

See also circles, drawing

- 2-Point option, 43
- 3-Point option, 43
- Diameter option, 43
- radius, 42
- tangent tangent radius option, 44

circles, drawing, 42–44. See also spheres

- center marks, 45–46
- centerlines, 400–401
- circular bases
 - cones with, 712
 - cylinders with, 709–710
- diameter, 6–8, 43
- in L-shaped bracket drawing, 689
- with object snap. **See** object snap modes (Osnap)
- quadrant-sensitive applications, 44
- radius, 42
- sketches, 173–174, 180
- tangent tangent radius, 44
- three points, 43
- two points, 43

circular pitch (CP), 627

circular thickness (CT), 627

circular views, of cylinders, 226

circularity tolerances, 492

Circumscribe option (Polygon tool), 56

clearance fits, 460–461, 627

- exercise problems, 482–484
- geometric tolerances, 529, 530–531

Close command (Application menu), 23–24

closed areas, 32

closed splines, 50

closing drawings, 23–24

color

- background, 671
- layers, 119–121
- text, 61

combined solid objects

- drawing, 727–731
 - boxes, 727–728, 730
 - drawing setup, 727
 - wedges, creating, 730–731
- internal open volume, 728–729
- top cutout, 729

combining

- blocks, 114
- spur gears, 636–637

command line, 4, 6–7

command sequences, exiting, 31

command tools, 7–8. See also individual tools

commands. See also individual commands

- accessing in panels, 5
- help information for, 6
- tooltips for, 6
- company name, on drawings, 586**
- compound lines, 213–214**
 - definition of, 213
 - drawing problems, 214
- compression springs, 564, 566**
- Conceptual visual style, 678, 685, 686, 695**
- conditions**
 - fixed, 469–471
 - floating, 468–469
- Cone tool (Modeling panel), 710–711**
 - cones with circular bases, 712
 - cones with elliptical bases, 711–712
- cones, 711–712**
 - drawing with circular bases, 711–712
 - drawing with elliptical bases, 711–712
 - in intersecting solids model, 733
- Construction Line commands, 38**
 - Ang option, 40–41
 - Hor option, 40–41
 - Offset option, 41–42
 - Ver option, 40–41
 - Xline option, 38–40
- Continue tool (Dimensions panel), 398**
- continued dimensions, 398**
- control vertex splines, 50**
- Control Vertex tool (Draw panel), 51**
- coordinate dimensions, 416–417**
- coordinate systems**
 - UCSs (user coordinate systems), 677–682
 - creating, 679
 - defining, 681–682
 - definition of, 677
 - restoring saved UCS, 680–681
 - returning to WCS (world coordinate system), 680
 - saving, 679–680
 - solid box, drawing, 677–678
 - WCS (world coordinate system)
 - background color, 671
 - default settings for, 669–670
 - returning to 2D WCS, 675, 680
 - returning to original orientation, 673
 - viewpoint, 672
- Coordinates panel, 680–681**
 - 3 Point tool, 681, 689, 697
 - Origin command, 689, 698
 - World command, 698, 735
 - X Axis Rotate UCS, 734
 - Y Axis Rotate UCS, 734
- Copy command (Modify panel), 63–64, 332, 596**
 - beveled gears, drawing, 640
 - S-breaks, creating, 305
- Copy Edges tool (Solid Editing panel), 750–751**
- Copy Faces tool (Solid Editing panel), 750**
- copying objects**
 - Copy command (Modify panel), 63
 - design layouts, 596
 - edges, 750–751
 - faces, 750
 - multiple copies, 64
- corner points, wedges, 713**
- COs (change orders), 587**
- counterbore, 233–234**
- countersunk holes, 234**
- CP (circular pitch), 627**
- Create command (Block Definition panel), 107–108**
- Create New Drawing dialog box, 8–9**
- Create New Multileader Style dialog box, 289**
- Create panel, Wedge tool, 731**
- crests, thread, 535–536**
- cross-hatching. See hatching**
- CT (circular thickness), 627**
- cubes, drawing, 708**
- curves**
 - curved leader lines, 402–403
 - drawing with thickness, 695–696
 - ogee, 139–140
 - parabolic, 141
 - sketching, 173–174
- customization, Quick Access Toolbar, 21–22**
- cuts, sectional. See sectional views**
- cutting plane lines, 287–290**
 - drawing, 287–290
 - patterns for, 287
- Cylinder tool, 679, 682, 709–711, 739–740, 742. See also cylinders**
- cylinders, 709–711**
 - creating with thickness, 697–698
 - drawing with circular base, 709–710
 - drawing with elliptical base, 710–711
 - in intersecting solids model, 734–735
 - orthographic views, 226–230
 - circular views, 226
 - drawing conventions and, 229–230
 - drawing problems, 227, 228–229
 - rectangular views, 226
 - with slanted and rounded surfaces, 228–229
- cylindricity tolerances, 493**

D

d (dedendum), 627

Dashed cutting plane lines, 287–290

database information, displaying for drawn solid objects, 743

datum, definition of, 494

Datum Identifier (Geometric Tolerance dialog box), 494

datum surfaces, 465

deceleration, uniform, 654–655

Decimal Degrees angle units, 13

decimal unit system, 11

dedendum (d), 627

Define Attributes tool (Block Definition panel), 126–128

degree symbol (°), 61

Degrees/Minutes/Seconds angle units, 13

design layouts, 593–598

copying, 596

creating, 594–595

creating drawings from, 597–598

definition of, 593

detail drawings

design layouts, 593–598

copying, 596

creating, 594–595

creating drawings from, 597–598

definition of, 593

drawing notes, 592

drawing problems

assembly drawing examples, 599–602

design layouts, 594–598

layers, 596–597

exercise problems, 603–624

first-angle projection, 591–592

detailed thread representations

definition of, 538

drawing, 538–539

internal, 539–540

deviation tolerances, 445–446

diagrams, cam displacement. See cam displacement diagrams

dialog boxes

Advanced Attribute Editor, 131

Attribute Definition, 126–128, 132

Circle Properties, 124

Create New Drawing, 8–9

Create New Multileader Style, 289

Dimension Style Manager, 45

precision of dimension values, 446, 449

tolerances, creating, 444–446, 447

Drafting Settings, 17–18, 20, 38

Apparent Intersect object snap, 94–95

Object Snap, 90–91

Snap and Grid options, 17–18

Drawing Units, 11–13, 380–381

Drawing Window Colors, 671

Edit Attributes, 128, 130

Enhanced Attribute Editor, 131

Geometric Tolerance

Datum Identifier, 494

leader lines, 500

positional tolerance, 498

straightness value, 494–495

Help, 6

Insert Table, 75

Layer Properties Manager, 118–122, 207–208

Leader Settings, 402, 499

Linetype Manager, 205, 288

Load or Reload Linetypes, 121, 205, 288

Material Condition, 499

Modify Dimension Style, 45, 389, 447, 496, 497

precision options, 383

Scale factor, 377–378

Text tab, 378–380

zero suppression, 382

Modify Multileader Style, 289

Modify Table Style: Standard, 76

Multileader Style Manager, 289, 403–404, 501

Options, 671

accessing, 91

Aperture Size option, 91

Page Setup Manager, 15–16, 20

Page Setup - Model, 15, 20

Point Style, 56–57

Primary Units, 449

Printer/Plotter, 16

Quick Properties, 208

Save Drawing As, 9–11, 21–22

Select Color, 208

Select File, 23

Select Linetype, 120, 208

Select Template, 3–5, 9, 19, 582–584, 674

Symbol, 495

Table Style, 75, 76

UCS, 679

View Manager, 672

Viewports, 691

Write Block, 115–117

diameter

of circles, 43

dimensions, 388–389

center marks and lines, 389

creating, 388–389

- outside, 627
- pitch, 627
- root, 627
- of threads, 535–536
- diameter symbol (Ø), 61, 390**
- Diameter tool (Dimensions panel), 6–8, 388–389, 498. See also diameter**
 - center marks and lines, 389
 - diameter dimensions, creating, 388–389
- diametral pitch (DP), 627**
- dihedral angles, 340**
- dimension lines, 369**
- Dimension Style Manager, 45**
 - precision of dimension values, 446, 449
 - tolerances, creating
 - limit tolerances, 447
 - plus or minus tolerances, 444–446
- Dimension tool, 449**
- dimensions, 451–453. See also tolerances**
 - aligned, 384–385
 - creating, 384–385
 - definition of, 384
 - text options, 385
 - angular, 369, 391–393
 - creating, 392
 - overdimensioning, avoiding, 393
 - baseline, 396–397
 - center marks, 400–401
 - chain, 451–453
 - chamfers, 412–413
 - common errors in, 370
 - continued, 398
 - conventions for, 369
 - coordinate, 416–417
 - creating, 495–497
 - diameter, 388–389
 - center marks and lines, 389
 - creating, 388–389
 - dimension lines, 369
 - double dimensioning, 450–451
 - exercise problems, 420–440
 - extension lines, 369
 - fillets and rounds, 409
 - holes and hole patterns, 406–407
 - importance of, 367–368
 - irregular surfaces, 410–411
 - keys and keyseats, 414
 - knurls, 413–414
 - leader lines, 369, 401–404
 - creating with Multileader, 403–404
 - creating with Quick Leader tool, 401–403
 - curved, 402–403

- linear, 369, 370–375
 - adding to given centerlines, 389–390
 - creating, 370–375
 - default dimension text, changing, 372–374
 - editing, 374–375
 - horizontal dimensions, 371–372
 - vertical dimensions, 371
- nominal, 468
- ordinate, 393–397
- orthographic views, 417–418
- placement of, 406–407
- points, 415
- polar, 411–412
- precision of, 446, 449
- quick filters, 398
- Quick tool for, 399–400
- radius, 369
 - altering default, 386–387
 - of circles, 42
 - creating, 385–387
 - overview of, 385
 - removing center mark from, 387
 - very large radii, 419
- rectangular, 454
- rounded shapes
 - external, 410
 - internal, 409
- sectional views, 417
- series of, 399–400
- styles of, 376–380
 - drawing scale, 377–378
 - drawing units, 380–381
 - precision options, 383
 - text height/placement, 378–380
 - zero suppression, 382
- symbols and abbreviations, 414
- symmetry and centerline, 415
- terminology for, 368–369
- text angle, 404
- tolerances, 405
- very large radii, 419
- Dimensions panel**
 - Aligned tool, 384–385
 - creating aligned dimensions with, 384–385
 - overview of, 384
 - text options, 385
 - Angular tool, 391–393
 - creating angular dimensions with, 392
 - overdimensioning and, 393
 - Baseline tool, 396–397, 452–453
 - Center Mark tool, 400–401
 - Continue tool, 398

- Diameter tool, 388–389, 498
 - center marks and lines, 389
 - diameter dimensions, creating, 388–389
- Dimension Style Manager, 444–446
- Jogged tool, 419
- Linear tool, 370–375, 389–390, 443, 447, 452–453
 - changing default dimension text with, 372–374
 - dimension styles, 376–380
 - editing dimensions with, 374–375
 - horizontal dimensions, 371–372
 - vertical dimensions, 371
- Ordinate tool, 393–397
- overview of, 367–368
- Quick tool, 399–400
- Radius tool, 385–387
 - altering default dimensions with, 386–387
 - creating dimensions with, 385–386
 - overview of, 385
 - removing center mark with, 387
- Text Angle tool, 404
- Tolerance tool, 494, 498

directrix, 141

displacement diagrams

- axes in, 650
- cam followers
 - drawing problems, 655–657
 - exercise problems, 658–659
 - types of, 655
- cam motions, 652–655
 - harmonic motion, 653–654
 - modified uniform motion, 652–653
 - uniform acceleration and deceleration, 654–655
 - uniform motion, 652
- dwells in, 651
- overview of, 650–651, 667
- setting up, 651–652

Dist command, 332–333

dollar sign (\$), 8

double dimensioning, 450–451

double-hatched lines, 413

DP (diametral pitch), 627

Drafting & Annotation workspace, 676

Drafting Settings dialog box, 17–18, 20, 38

- 3D Object Snap, 689
- Apparent Intersect object snap, 94–95
- Object Snap, 90–91
- Snap and Grid options, 17–18

Draw panel, 5

- Circle tool, 42–44
 - 2-Point option, 43

- 3-Point option, 43
- Diameter option, 6–8, 43
- drawing circles about midpoint of, 93
- radius, 42
- tangent tangent radius option, 44
- Control Vertex tool, 51
- Ellipse tool, 51–53, 94
 - axis endpoints, 51–52
 - center points, 52–53
 - elliptical arcs, 53
 - holes in slanted surfaces, 222
- Fit tool, 51
- Gradient tool, 298
- Hatch tool, 292–294, 413, 542, 641
- Helix tool, 145, 719–720
- Line command, 29–32, 92, 95–96, 97, 217
 - closed area for, 32
 - randomly selected points for, 29–31
- Point tool, 56–57
- Polygon tool, 55–56
 - center points, 55–56
 - hexagon-shaped heads, drawing, 550
 - square-shaped heads, drawing, 552–554
- Polyline command, 47–50, 302, 303, 695
 - drawing polylines with, 47
 - line thickness, 50
 - polyline arcs, 48–50
 - verifying polyline is single entity with, 47
- Ray command, 38
- Rectangle tool, 54, 689
- Spline Fit tool, 51, 655

Drawing & Annotation workspace, 676

drawing area (drawing editor), 5

drawing limits, 13–16

- aligning with standard A3 (metric) paper size, 15–16
- drawing problem, 20
- options for, 13–14

drawing problems

- assembly and detail drawings, 593–602
 - assembly drawing examples, 599–602
 - design layouts, 594–598
 - layers, 596–597
- auxiliary views, 331–332
 - irregular surfaces, 337–340
 - lines, transferring between views, 334–335
 - rounded surfaces, 335–337
 - secondary views, 346–348
- ball bearings, 646–647
- bushings, 647–650
- cam followers, 655–657

- fasteners
 - bolts, 558–559
 - screws, 557–558
- geometric tolerances
 - fixed fasteners, 513–515
 - floating fasteners, 511–512
- new drawings, 18–20
 - drawing units, 19
 - precision of, 19
- object snap modes (Osnap), 98–101
- orthographic views
 - castings, 237–238
 - compound lines, 214
 - cylinders, 227, 228–229
 - hidden lines, 212–213
 - irregular surfaces, 231–232
 - oblique surfaces, 217
 - rounded surfaces, 219–220
 - thin-walled objects, 239–240, 241–245
- sectional views, 295
- shape parameters, 145–147
- sketches, 173–174, 179–180
- spur gears
 - center distance between gears, 635–636
 - meshing spur gears, 629–634
- thread representations, 557–559
- tolerances, 461–462
 - hole and shaft basis, 461–462
 - preferred and standard sizes, 467–468
 - shaft diameters, 457–458
 - standard fits (inch values), 463
 - standard fits (metric values), 461–462
- UCSs (user coordinate systems), 686–687, 688–690
- drawing revisions, 587**
- drawing screen, 2**
 - command line, 4, 6–7
 - drawing area, 5
 - drawing limits, 13–16
 - aligning with standard A3 (metric) paper size, 15–16
 - drawing problem, 20
 - options for, 13–14
 - grid and snap values, 17–18, 20
 - panels, 5–6
 - Quick Access Toolbar, 21–22
 - ribbon, 4
 - starting new drawings from, 2–5
 - tabs, 5–6
 - User Coordinate System (UCS) icon, 5
- drawing units, 380–381**
 - angle units value, 13
 - drawing problem, 19
 - precision of, 13
 - selecting, 11
- Drawing Units dialog box, 11–13, 380–381**
- Drawing Utilities command (Application menu), 11–12, 19**
- Drawing Window Colors dialog box, 671**
- drawings. See also 2D construction; 3D modeling; dimensions**
 - assembly
 - assembly numbers, 580–581
 - BOM (bill of materials), 589–591
 - design layouts, 593–598
 - drawing problems, 593–602
 - exercise problems, 603–624
 - families of drawings, 580
 - formats for, 582–584
 - overview of, 579–581
 - release block, 588–589
 - revision block, 586–587
 - revisions, 587
 - scale of, 586
 - templates, 582–584
 - templates, adding, 582–584
 - title block, 584–586
 - tolerance block, 588
 - auxiliary views
 - axis system, rotating, 330–331
 - definition of, 327
 - drawing problems, 331–332, 334–340, 346–348
 - exercise problems, 350–366
 - irregular surfaces, 337–340
 - lines, transferring between views, 332–335
 - oblique surfaces, 342–343
 - overview of, 327–328
 - partial views, 340–341
 - projecting information from normal view into, 328–331
 - rounded surfaces, 335–337
 - secondary views, 343–349
 - sectional views, 341–342
 - slanted surfaces, 342
 - axis system, rotating, 330–331
 - closing, 23–24
 - conventions for
 - cylinders, 229–230
 - sections, 301
 - creating from blocks, 115–117
 - detail drawings
 - design layouts, 593–598
 - drawing notes, 592

- drawing problems, 593–602
 - exercise problems, 603–624
 - first-angle projection, 591–592
 - drawing limits, 13–16
 - aligning with standard A3 (metric) paper size, 15–16
 - drawing problem, 20
 - options for, 13–14
 - drawing problems, 18–20
 - assembly drawing examples, 599–602
 - design layouts, 594–598
 - drawing units, 19
 - layers, 596–597
 - precision of, 19
 - drawing units
 - angle units value, 13
 - drawing problem, 19
 - precision of, 13
 - selecting, 11
 - exercise problems, 25–27
 - families of, 580
 - layers
 - changing object layer, 123–124
 - color of, 119–121
 - creating, 118–119
 - definition of, 117–118
 - drawing on, 122–123
 - exercise problems, 149–167
 - linotype, 119–121, 124–125
 - matching, 125–126
 - properties of, 118
 - turning off, 126
 - naming conventions for, 5, 8
 - notes, 592
 - opening, 23
 - precision of, 19
 - revisions, 587
 - saving, 9–11, 21–22
 - sectional views
 - aligned sections, 300
 - breaks, 303–305
 - broken-out views, 302
 - of castings, 305
 - cutting plane lines, 287–290
 - drawing conventions, 301
 - drawing problem, 295
 - exercise problems, 306–323
 - gradients, 298
 - half and partial views, 302
 - hatching, 290–294
 - holes in, 297–298
 - locations of, 296–297
 - multiple sections, 300
 - offset sections, 299
 - overview of, 285–287
 - removed views, 303
 - section lines, 296
 - sketches
 - curves, 173–174
 - developing personal style of, 169
 - drawing problems for, 173–174, 179–180
 - exercise problems, 185–197
 - feathering, 171
 - graph paper for, 170
 - importance of, 169
 - isometric, 176–178, 186–189, 190–196
 - lines, 171–172
 - oblique, 180–181, 189
 - orientations for, 178, 184
 - pencils for, 170
 - perspective, 182–183, 186-C04.03354
 - proportions, 172–173
 - starting new, 2–5, 8–9
 - Create New Drawing dialog box, 8–9
 - drawing limits, 13–16, 20
 - drawing units, 11–13, 19
 - grid and snap values, 17–18, 20
 - Save Drawing As dialog box, 9–11, 21–22
 - Select Template dialog box, 9
 - thread representations, 538–542
 - callouts, 536–537
 - crests, 535–536
 - detailed, 538–540
 - drawing problems, 557–559
 - internal, 539–540
 - major/minor diameter, 535–536
 - roots of, 535–536
 - schematic, 538, 540–542
 - simplified, 538, 542
 - terminology for, 535–536
 - driven helical gears, 643**
 - driver helical gears, 643**
 - Dtext, 444**
 - dwells, 651**
 - .dwg filename extension, 8**
 - dynamic grips, drawing boxes with, 708–709**
 - Dynamic Input, drawing lines with, 35–38**
- E**
- ECOs (engineering change orders), 587**
 - edges**
 - copying, 750–751
 - editing, 744
 - extracting, 751

- imprinting objects, 751
- polygon edge distance, 56
- Edit Attributes dialog box, 128, 130**
- Edit panel**
 - Subtract tool, 722, 723–724
 - Union tool, 722–723
- Edit Polyline tool (Modify panel), 135, 141, 143, 695, 738**
- Edit Spline tool (Modify panel), 136–137**
- editing**
 - block attributes, 131–132
 - dimensions, 374–375
 - edges, 744
 - copying, 750–751
 - extracting, 751
 - imprinting objects, 751
 - faces, 744–751
 - basic extruding of, 745
 - copying, 750
 - extruding along a path, 745–746
 - extruding multiple, 746–747
 - moving, 747
 - offsetting, 747–748
 - rotation, 748–749
 - tapering, 749
 - polylines, 135
 - solid models, 682–683
 - splines, 136–137
 - text, 137
- Ellipse tool (Draw panel), 51–53, 94, 222, 305.**
 - See also ellipses*
- ellipses, 51–53, 348–349**
 - axis endpoint, 51–52
 - center points of, 52–53
 - defining with Osnap Intersection mode, 94
 - drawing, 51–53
 - elliptical arcs, 52–53
 - elliptical bases
 - cones with, 711–712
 - cylinders with, 710–711
 - in L-shaped bracket model, 690
 - as representations of projected holes, 223
 - sketching, 173–174
- Endpoint object snap, 32, 92–93, 140, 236, 657, 679, 682, 686, 697, 725**
- endpoints, axis, 51–52**
- engineering change orders (ECOs), 587**
- engineering orders (EOs), 587**
- engineering unit system, 11**
- English units**
 - for spur gears, 626–627
 - thread callouts, 536–537
- Enhanced Attribute Editor dialog box, 131**
- EOs (engineering orders), 587**
- Equal option (Viewport Configuration panel), 736**
- Erase tool (Modify panel), 32–34, 217, 289, 631**
 - groups of lines, 34
 - individual lines, 33–34
- erasing lines, 32–34**
 - groups of lines, 34
 - individual lines, 33–34
- errors, dimensioning, 370**
- Esc key, 31**
- exercise problems**
 - 2D construction, 78–88
 - advanced commands, 149–167
 - assembly and detail drawings, 603–624
 - auxiliary views, 342–343
 - cam displacement diagrams, 658–659
 - dimensions, 420–440
 - drawing, 25–27
 - fasteners, 569–578
 - gears, 658, 661–665
 - geometric tolerances, 516–533
 - clearance fits, 516–531
 - dimensions and tolerances, 516–528
 - interference fits, 529–530, 531
 - locational fit, 530, 531
 - transition fits, 530
 - transitional fits, 531
 - orthographic views, 251–283
 - sectional views, 306–323
 - sketches, 185–197
 - solid modeling, 752–778
 - Thickness system variable, 699–703
 - threads, 569–578
 - tolerances, 472–485
 - clearance fits, 482–484
 - interference fits, 483, 484
 - locational fit, 483, 484
 - transition fits, 484–485
- Explode command (Modify panel), 54, 109, 289**
- exploding**
 - blocks, 109
 - rectangles, 54
- expressions, tolerance, 442–443**
- Extend command (Modify panel), 70, 345, 348**
 - beveled gears, drawing, 641
 - uniform acceleration and deceleration, 655
- extension lines, 369**
- extension springs, 564–566**
- external acme threads, 546–548**
- external rounded shapes, 410**

external square threads, 545–546

Extract Edges tool (Solid Editing panel), 751

extracting edges, 751

Extrude Faces tool (Solid Editing panel), 745–747

Extrude tool (Modeling panel), 716–718

extrusion, 716–718

in castings model, 739

of faces

along a path, 745–746

basic extrusion, 745

multiple faces at same time, 746–747

F

face angle lines, 640

face width (FW), 627

faces

basic extruding of, 745

copying, 750

editing, 744–751

extruding along a path, 745–746

moving, 747

offsetting, 747–748

rotating, 748–749

tapering, 749

families of drawings, 580

fasteners

bolts

drawing problems, 558–559

head shapes, 550–554

overview of, 548

exercise problems, 569–578

fixed, 512–515

drawing problems, 513–515

overview of, 512–513

floating, 510–512

drawing problems, 511–512

overview of, 510

keys, 562–563

nuts

hexagon-shaped finished nuts, 554–556

locknuts, 556–557

overview of, 554

rivets, 563

screws, 559–561

drawing problems, 557–558

head shapes, 550–554

overview of, 549

setscrews, 560–561

shoulder, 567–568

standard, 559–560

springs, 564–566

compression, 564, 566

extension, 564–566

overview of, 564

torsional, 564

studs, 550

Tool Palettes, 567–568

accessing, 567

blocks, changing scale of, 567–568

shoulder screws, creating, 567

washers, 561–562

lock, 561

plain, 561–562

star, 561

feathering, 171

files

naming conventions for, 8

saving, 9–11

Filled square property (Layers), 118

Fillet command (Modify panel), 73–74, 147, 237, 409, 630, 737, 741

fillets, 73–74

definition of, 234–235

dimensioning, 409

finished nuts, 554–556

First point option, 72

first-angle projection, 591–592

fit splines, 50

Fit tool (Draw panel), 51

fixed conditions, 469–471

fixed fasteners, 512–515

drawing problems, 513–515

overview of, 512–513

flatness tolerances, 488

floating conditions, 468–469

floating fasteners, 510–512

drawing problems, 511–512

overview of, 510

FN (force fits), 462

followers, cam

drawing problems, 655–657

types of, 655

fonts, multiline text, 59–60

form, tolerances of, 488–493

circularity, 492

cylindricity, 493

flatness, 488

straightness, 489

MMC (maximum material condition), 489–492

overview of, 489

RFS (regardless of feature size), 489–492

formats, assembly drawing, 582–584

fractional unit system, 11

freehand lines, breaks with, 303
freehand sketching. See sketching
FW (face width), 627

G

gears

bevel
 definition of, 637
 drawing, 639–641
 face angle lines, 640
 matched sets of, 640–641
 terminology for, 637–639
exercise problems, 658, 661–665
helical, 643
modules, 627
overview of, 625
racks, 644–645
spur
 center distance between, 635
 combining, 636–637
 drawing problems, 629–634, 635–636
 drawings of, 628–629
 English and metric units for, 626–627
 meshing spur gears, 631–632
 pinions, 644
 selecting, 633–634
 terminology for, 626–627
tooth shape and proportions, 629, 630–631,
 632–633
worm, 641–642

general hatch patterns, 292

Geometric Tolerance dialog box

Datum Identifier, 494
leader lines, 500
positional tolerance, 498
straightness value, 494–495

geometric tolerances

angularism, 504–505
circularity, 492
creating, 493–500
 basic dimension, 495–497
 datum, 494
 leader lines, 499–500
 limit tolerances, 497–498
 positional tolerance, 495–496, 498–499
 straightness value, 494–495
cylindricity, 493
defined tolerance zone, 487
drawing problems
 fixed fasteners, 513–515
 floating fasteners, 511–512
exercise problems, 516–533

clearance fits, 516–528, 530–531
dimensions and tolerances, 516–528
interference fits, 529–530, 531
locational fit, 530, 531
transition fits, 530, 531
fixed fasteners, 512–515
 drawing problems, 513–515
 overview of, 512–513
flatness, 488
floating fasteners, 510–512
 drawing problems, 511–512
 overview of, 510
locational tolerance, 502
overview of, 487–488
parallelism, 504
perpendicularity, 502–503
positional tolerances, 508–509
profiles, 505–506
runouts, 506–507
straightness, 489–492
tolerances of form, 488–493
 circularity, 492
 cylindricity, 493
 flatness, 488
 straightness, 489–492
tolerances of orientation, 500
virtual conditions, 509–510

gib head keys, 562–563

Gradians angle units, 13

Gradient tool (Draw panel), 298

gradients, creating, 298

graph paper, 170

Grid command, 4, 17–18, 20, 651

grids, 673–675

grinding wheels, 615, 773

grips

definition of, 101–102
extending lines with, 103–104
Grips menu, accessing, 102
mirroring objects with, 106
moving lines with, 333–334
moving objects with, 104
rotating objects with, 104–105
scaling objects with, 105
turning off, 102

Grips menu

accessing, 102
Mirror command, 106
Rotate command, 104–105
Scale command, 103–104
Stretch command, 103–104

H

half sectional views, 302

harmonic motion, 653–654

hatch lines, 413, 641

Hatch Pattern panel, 296

Hatch tool (Draw panel), 292–294, 413, 542, 641

hatching

hatch creation, 292–293

hatch lines, 641

hatch patterns

double-hatched lines, 413

selecting, 293–294, 296

spacing and angle of, 294

overview of, 290–294

HB pencil lead, 170

head shapes (bolt/screw), 550–554

hexagon, 550–552

square, 552–554

height, of multiline text, 59

helix, 719–720

constructing, 143–145

definition of, 145

helical gears, 643

Helix tool (Draw panel), 145, 719–720

Help dialog box, 6

help information, accessing, 6

hexagons

drawing with thickness, 693–694

hexagon-shaped finished nuts, 554–556

hexagon-shaped heads, 550–552

hidden layers

changing, 208–209

creating, 207–208

hidden lines, 203–209

conventions for, 204–205

definition of, 203–204

drawing, 205–209

drawing problems, 212–213

hidden layers, 207–208

hidden linetypes, 205–207

properties of, 208–209

hidden linetypes, adding, 205–207

holes, 220–226. See also tolerances

in castings model, 740

choosing shafts for, 456–457

definition of, 220–221

designing from fastener size, 471

dimensioning, 406–407

examples of, 221

hole basis tolerances, 461

hole callouts, 232–234

counterbore, 233–234

countersunk holes, 234

ream, 232

spotface, 233

locations of, 455

in oblique surfaces, 224–226

projected, 223

in sectional views, 297–298

in slanted surfaces, 222–223

hollow rivets, 563

Home panels, 5

Hor option (Construction Line command), 40–41

horizon lines, 182

horizontal dimensions, 371–372

horizontal lines, 40–41

hyperbolas, 141–143

hyphen (-), 8

I

icon, UCS (user coordinate system), 5, 685, 686

Imprint tool (Solid Editing panel), 751

imprinting objects, 751

inch values, for standard fit tolerances, 462–463

drawing problems, 463

fits defined by inch values, 462–463

Inscribe option (Polygon tool), 56

Insert command (Block panel), 107–108, 109, 110, 126, 567

Insert Table dialog box, 74, 75

interference fits, 460–461

exercise problems, 483, 484

geometric tolerances, 529–530, 531

internal detailed thread representations, 539–540

internal open volume, 728–729

internal rounded shapes, 409

internal thread representations

detailed thread representations, 539–540

orthographic views of, 542–543

schematic, 541–542

sectional views of, 544

International Organization for Standardization (ISO), 247

International System of Units (SI) symbols, 592

Intersect tool (Solid Editing panel), 724

Intersection object snap, 93–94, 138–139, 143, 144, 217, 227, 234, 236, 245, 348, 653

intersections, 241–245, 724

- intersecting solids model, 732–736
 - completing, 735
 - cone, 733
 - cylinder, 734–735
 - drawing setup, 732
 - viewport configuration, 736
- Intersection object snap, 93–94, 138–139, 143, 144, 227, 653

irregular surfaces

- definition of, 217
- dimensioning, 410–411
- orthographic views, 230–232
 - definition of, 230–231
 - drawing problems, 231–232
- projection into auxiliary views, 337–340

ISO (International Organization for Standardization), 247

isometric sketches

- description of, 176–178
- exercise problems, 186–189, 190–196

J

Jogged tool (Dimensions panel), 419

justifying text, 60–61

K

keys, 414, 562–563

keyseats, 414, 562

keyways, 414, 562–563

knuckle threads, 544–545

knurls, 413–414

L

large radii, dimensioning, 419

lay, 466

Layer Properties, 118–122, 207

Layer Properties Manager dialog box, 118–122, 207–208

layers, 596–597

- changing an object's layer, 123–124
- color of, 119–121
- creating, 118–119
- definition of, 117–126
- drawing on, 122–123
- exercise problems, 149–167
- hidden
 - changing, 208–209
 - creating, 207–208
- linotype, 119–121, 124–125
- matching, 125–126

properties of, 118

turning off, 126

Layers panel, 118–122, 207

layouts, design, 593–598

- copying, 596
- creating, 594–595
- creating drawings from, 597–598
- definition of, 593

LC (clearance locational fits), 462

lead, of worm thread, 642

lead, pencil, 170

leader lines, 369, 401–404

- creating with Multileader, 403–404
- creating with Quick Leader tool, 401–403
- curved, 402–403
- definition of, 499–500

Leader Settings dialog box, 402, 499

Leaders panel, 287, 289, 403–404

Lightbulb property (layers), 118

limits

- drawing, 13–16
 - aligning with standard A3 (metric) paper size, 15–16
 - for drawing problem, 20
 - options for, 13–14
- tolerances, 446–448
 - changing existing dimension to, 448
 - creating, 447, 497–498
 - example of, 446–447

Limits command, 16

Line command (Draw panel), 29–32, 217, 693.

See also lines

- closed area for, 32
- harmonic cam motion, 653
- with object snap modes, 92, 95–98
- randomly selected points for, 29–31

line profile tolerance, 505–506

linear dimensions, 369, 370–375

- adding to given centerlines, 389–390
- creating, 370–375
- default dimension text, changing, 372–374
 - with Mtext option, 373–374
 - with text option, 372–373
- editing, 374–375
- horizontal dimensions, 371–372
- vertical dimensions, 371

Linear tool (Dimensions panel), 370–375, 443,

447, 452–453. See also linear dimensions

- creating linear dimensions with, 370–375
- default dimension text, changing, 372–374
 - with Mtext option, 373–374
 - with text option, 372–373

- dimension styles, 376–380
 - drawing scale, 377–378
 - drawing units, 380–381
 - precision options, 383
 - text height/placement, 378–380
 - zero suppression, 382
- editing dimensions with, 374–375
- horizontal dimensions, 371–372
- linear dimensions, 389–390
- vertical dimensions, 371

lines

- compound, 213–214
 - definition of, 213
 - drawing problems, 214
- cutting plane lines, 287–290
 - drawing, 287–290
 - patterns for, 287
- dimension, 369, 389
- drawing to quadrant point, 96
- drawing with Construction Line commands, 38–42
 - Ang option, 40–41
 - Hor option, 40–41
 - Offset option, 41–42
 - Ver option, 40–41
 - Xline option, 38–40
- drawing with Dynamic Input, 35–38
- drawing with Line command, 29–32
 - closed area for, 32
 - randomly selected points for, 29–31
- drawing with object snap. **See** object snap modes (Osnap)
- erasing, 32–34
 - groups of lines, 34
 - individual lines, 33–34
- extending, 103–104, 369
- grip points, 333–334
- hatch, 413, 641
- hidden, 203–209
 - conventions for, 204–205
 - definition of, 203–204
 - drawing, 205–209
 - drawing problems, 212–213
 - hidden layers, creating, 207–208
 - hidden linetypes, adding, 205–207
 - properties of, 208–209
- leader lines, 369, 401–404
 - creating with Multileader, 403–404
 - creating with Quick Leader tool, 401–403
 - curved, 402–403
- line profile tolerance, 505–506
- measuring length of, 332–333

- phantom, 581
- polylines, 46–50
 - in castings model, 738
 - creating from basic shapes, 738
 - creating from line segments, 717–718
 - creating splined polylines from, 135
 - drawing, 47
 - extruding, 716–717
 - line thickness, 50
 - polyline arcs, 48–50
 - verifying as single entity, 47
- precedence of, 209–210
- rotating and moving, 334
- section lines, 296
- sketching, 170
- snap points, 34–35
 - definition of, 34
 - drawing problems, 35
- splines, 50–51
- Thickness system variable, 693–698
 - creating objects with, 696–698
 - drawing curves with, 695–696
 - drawing hexagons/arcs with, 693–694
 - exercise problems, 699–703
 - overview of, 693
- transferring between views, 332–335
 - drawing problem, 334–335
 - grip points, 333–334
 - line length measurement, 332–333
 - line rotation, 334

Linetype command, 205

Linetype Manager dialog box, 205, 288

linotype, layers, 119–121, 124–125

List command, 743

lists, for solid objects, 743

LN (interference fits), 462

Load or Reload Linetypes dialog box, 121, 205, 288

locational fit

- exercise problems, 483, 484
- geometric tolerances, 530, 531

locational tolerance, 502

locations

- of holes, 455
- of sectional views, 296–297

lock washers, 561

locking nuts, 556–557

locknuts, 556–557

Loft tool (Modeling panel), 721–722

lofts and lofting, 721–722

L-shaped bracket drawing problem, 686–687, 688–690

circles, 689
ellipses, 690
rectangles, 689

LT (transitional locational fits), 462

M

major diameter, threads, 535–536

Make Current tool, 123

Massprop command, 744

Match Layer tool, 125–126

Material Condition dialog box, 499

materials, bill of. See BOM (bill of materials)

maximum length, calculating, 454

maximum material condition (MMC), 489–492

meshing spur gears, 631–632

metric units

for spur gears, 626–627

for standard fits, 458–461

clearance fits, 460–461, 482–484

drawing problem, 461–462

hole and shaft basis, 461

interference fits, 460–461, 483, 484

locational fit, 483, 484

nominal sizes, 461

transition fits, 460–461, 484–485

thread callouts, 536

Midpoint object snap, 93

minimum length, calculating, 454

minivise model

assembly drawings of, 621–623, 776

exploded view of, 776–777

minor diameter, threads, 535–536

Mirror command, 65, 106, 305

mirroring objects, 65, 106

mleader command, 401–404

MMC (maximum material condition), 489–492

model space, 583

Model Viewports panel, 690

Modeling panel

Box tool, 706–709

boxes, drawing from given dimensions,
707–708

boxes, drawing with corner option, 706

cubes, drawing, 708

dynamic grips, 708–709

visual styles, 706–707

Cone tool, 710–711

cones with circular bases, 712

cones with elliptical bases, 711–712

Cylinder tool, 709–711, 739–740, 742

cylinders, drawing with circular base, 709–710

cylinders, drawing with elliptical base,
710–711

Extrude tool, 716–718

Loft tool, 721–722

Revolve tool, 718–719

Sphere tool, 709

Sweep tool, 720

Torus tool, 715–716

Wedge tool, 713–715

alignment with existing wedges, 714–715

center points, 713–714

corner points, 713

modified uniform motion, 652–653

**Modify Dimension Style dialog box, 45, 389,
447, 496, 497**

precision options, 383

Scale factor, 377–378

Text tab, 378–380

zero suppression, 382

Modify Multileader Style dialog box, 289

Modify panel, 332

Array command, 742

Break at Point command, 71–72

Break command, 71–72

Chamfer command, 72–73, 540, 743

Copy command, 63–64

Edit Polyline tool, 135, 141, 143, 695, 738

Edit Spline tool, 136–137

Erase tool, 32–34, 217, 289, 631

groups of lines, 34

individual lines, 33–34

Explode command, 54, 109, 289

Extend command, 70, 345, 348

beveled gears, drawing, 641

uniform acceleration and deceleration, 655

Fillet command, 73–74, 147, 237, 409, 630, 737,
741

Mirror command, 65

Move command, 62–63, 289, 743

Move tool, 62–63, 584

Offset command, 64–65, 147, 217, 332–333,
557, 640

Path Array tool, 67–68

Polar Array tool, 67, 143, 144, 657

Rectangular Array tool, 66

Rotate command, 68–69, 104–105, 289, 332,
333–334

Trim command, 69–70, 217, 237, 631, 738

Modify Table Style dialog box, 76

modules, gear, 627

motions, cam, 652–655

- harmonic motion, 653–654
- modified uniform motion, 652–653
- uniform acceleration/deceleration, 654–655
- uniform motion, 652
- Move command (Modify panel), 62–63, 289, 584, 743**
 - auxiliary views, moving, 329
 - S-breaks, creating, 305
- Move Faces tool (Solid Editing panel), 747**
- moving objects, 62–63**
 - faces, 747
 - with grips, 104
- Mtext option, 137, 373–374, 449**
- Multileader Style Manager, 289, 403–404, 501**
- Multileader tool (Leaders panel), 287, 403–404**
- multiline text, 57–60**
 - creating, 57–58
 - symbol options, 61
 - text color, 61
 - text font, 59–60
 - text height, 59
 - text justification, 60–61
- Multiline Text Editor, 57–60, 137**
 - accessing, 59
 - creating text with, 57–58
 - symbol options, 61
 - text color, 61
 - text font, 59–60
 - text height, 59
 - text justification, 60–61
- Multiline Text tool (Annotate panel), 57–61, 584**
 - creating multiline text, 57–58
 - text font, 59–60
 - text height, 59
 - text justification, 60–61
- multiple copies, 64**
- multiple faces, extruding at same time, 746–747**
- multiple sections, 300**

N

- Named UCS combo control, 680**
- Named Views panel, 202**
- naming conventions, 5, 8, 585**
- Nearest object snap, 97–98**
- New command (Application menu), 2, 8, 582**
- nominal dimensions, 468**
- nominal sizes, 461**
- notes, drawing, 592**
- numbers**

- assembly, 580–581

- drawing, 586

nuts, 548

- hexagon-shaped finished nuts, 554–556

- locknuts, 556–557

- overview of, 554

O

Ø (diameter symbol), 61, 390, 503

object snap modes (Osnap), 93, 681–682

- accessing, 89–91

- aperture size, 91

- Apparent Intersect, 94–95

- Center, 95–96

- drawing problems, 98–101

- enabling, 91

- Endpoint, 32, 92–93, 140, 236, 657, 686, 697, 725

- exercise problems, 149–167

- Intersection, 93–94, 138–139, 142, 144, 217, 227, 234, 236, 245, 348, 653

- Nearest, 97–98

- overview of, 89

- Perpendicular, 96–97

- Quadrant, 96

- Snap command versus, 89

- Tangent, 97, 236, 653, 656

objects. *See also individual objects*

- blocks

- attributes, 126–134

- changing shape of, 113–114

- combining, 114

- creating, 107–108

- creating new drawing files from, 115–117

- definition of, 106

- exercise problems, 149–167

- exploding, 109

- inserting, 108–109, 110–113

- scaling, 109

- changing layer of, 123–124

- copying

- Copy command (Modify panel), 63

- design layouts, 596

- edges, 750–751

- faces, 750

- multiple copies, 64

- creating with thickness, 696–698

- boxes, 696

- cylinders, 697–698

- returning to original settings, 698

- UCSs (user coordinate systems), 697

- helix, 143–145
- mirroring, 106
- moving, 104
- rotating, 68–69, 104–105
 - axis system, 330–331
 - faces, 748–749
 - lines, 334
 - UCS (user coordinate system) axes, 683–684
- scaling, 105
- spirals, 143
- stretching, 103–104
- oblique sketches**
 - description of, 180–181
 - exercise problems, 189
- oblique surfaces, 215–217**
 - auxiliary views of, 342–343
 - definition of, 215
 - drawing problems, 217
 - holes in, 224–226
- OD (outside diameter), 627**
- Offset command (Modify panel), 64–65, 147, 217, 332–333, 557, 640**
- offset distance, 64–65**
- Offset Faces tool (Solid Editing panel), 747–748**
- offset lines, 41–42**
- Offset option (Construction Line command), 41–42**
- offset sections, 299**
- offsetting faces, 747–748**
- ogee curves, 139–140**
- Open command**
 - Application menu, 23
 - Select Template dialog box, 4
- open splines, 50**
- opening drawings, 23**
- Options dialog box, 91, 671**
- ordinate dimensions, 393–397**
- Ordinate tool (Dimensions panel), 393–397**
- orientation**
 - sketch, 178, 184
 - tolerances of, 500
- origin points, 679**
- Origin tool (Coordinates panel), 689, 698**
- orthographic views**
 - of 3D objects, 690–692
 - creating, 692
 - viewports for, 690–691
 - castings, 234–238
 - definition of, 234–235
 - drawing problems, 237–238
 - compound lines, 213–214
 - definition of, 213

- drawing problems, 214
- creating, 692
- cylinders, 226–230
 - circular views, 226
 - drawing conventions and, 229–230
 - drawing problems, 227, 228–229
 - rectangular views, 226
 - with slanted and rounded surfaces, 228–229
- definition of, 199
- dimensioning, 417–418
- drawing problems, 212–213
 - compound lines, 214
 - cylinders, 227, 228–229
 - hidden lines, 212–213
 - irregular surfaces, 231–232
 - oblique surfaces, 217
 - rounded surfaces, 219–220
 - thin-walled objects, 241–245
- exercise problems, 251–283
- first-angle projection, 249–250, 591–592
- hidden lines, 203–209
 - conventions for, 204–205
 - definition of, 203–204
 - drawing, 205–209
 - drawing problems, 212–213
 - hidden layers, 207–208
 - hidden linetypes, 205–207
 - properties of, 208–209
- hole callouts, 232–234
 - counterbore, 233–234
 - countersunk holes, 234
 - ream, 232
 - spotface, 233
- holes, 220–226
 - definition of, 220–221
 - examples of, 221
 - in oblique surfaces, 224–226
 - projected, 223
 - in slanted surfaces, 222–223
- in intersecting solids drawing, 736
- intersections, 241–245
- irregular surfaces, 230–232
 - definition of, 217, 230–231
 - drawing problems, 231–232
- modification of existing parts, 245–249
- oblique surfaces, 215–217
 - definition of, 215
 - drawing problems, 217
- precedence of lines in, 209–210
- projection between, 211
- rounded surfaces, 217–220
 - definition of, 217–219

- drawing problems, 219–220
- slanted surfaces in, 210
- thin-walled objects, 238–240
 - definition of, 238–239
 - drawing problems, 239–240
- third-angle projection, 200–201, 249–250
- of thread representations, 542–543
- for three-dimensional box drawing, 202–203

Osnap. *See* **object snap modes (Osnap)**

outside diameter (OD), 627

overdimensioning, avoiding, 393

P

Padlock property (Layers), 118

Page Setup Manager dialog box, 15–16, 20

Page Setup - Model dialog box, 15, 20

Palettes panel, Tool Palettes

- accessing, 567–568
- blocks, changing scale of, 567–568
- blocks, modifying, 568
- shoulder screws, 567–568

panels. *See also individual panels*

- accessing commands in, 5
- overview of, 5–6

paper, graph, 170

paper size, drawing limits for, 13–16

- aligning with standard A3 (metric) paper size, 15–16
- options for, 13–14

paper space, 583

parabolas, 141

parabolic curves, 141

parallel grids, 673–675

parallelism tolerances, 504

partial auxiliary views, 340–341

partial sectional views, 302

parts list. *See* **BOM (bill of materials)**

Path Array tool (Modify panel), 67–68

paths, extruding faces along, 745–746

patterns, hatch

- selecting, 293–294, 296
- spacing and angle of, 294

PD (pitch diameter), 627

pencils, 170

percent sign (%), 8

Perpendicular object snap, 96–97

perpendicularity tolerances, 502–503

perspective grids, 673–675

perspective sketches

- description of, 182–183
- exercise problems, 186–189

phantom lines, 287–290, 581

pickbox, 33

pictorial drawings, 182–183

pinions, 644

pitch

- of knurls, 413–414
- pitch diameter (PD), 627

plain washers, 561–562

plan views, 670

plus or minus tolerance, 443–446

- creating
 - with Dimension Style Manager, 444–446
 - with Dtext, 444
 - with Text option, 443–444
 - with Text Override tool, 444
- definition of, 441
- overview of, 443

plus/minus symbol (\pm), 61

Point Style dialog box, 56–57

Point tool (Draw panel), 56–57

points, 56–57

- dimensioning to, 415
- hyperbolas, 141–143
- parabolas, 141
- randomly selecting, 29–31
- shape of, 57
- size of, 57

Polar Array tool (Modify panel), 67, 143, 144, 657

polar dimensions, 411–412

Polygon tool (Draw panel), 47–50, 55–56, 302, 303, 695

- center points, 55–56
- drawing polylines, 47
- hexagon-shaped heads, drawing, 550
- line thickness, 50
- polyline arcs, 48–50
- square-shaped heads, drawing, 552–554
- verifying that polyline is single entity, 47

polygons

- center points, 55–56
- definition of, 55–56
- drawing, 55–56
- edge distance, 56

polylines, 46–50

- in castings model, 738
- creating from basic shapes, 738
- creating from line segments, 717–718
- creating splined polylines from, 135
- drawing, 47
- extruding, 716–717
- line thickness, 50

- polyline arcs, 48–50
- verifying that polyline is single entity, 47
- positional tolerance, 508–509**
 - basic dimension, 495–497
 - creating, 495–496, 498–499
 - definition of, 495–496, 498–499
 - limit tolerances, 497–498
- Pratt & Whitney keys, 562–563**
- precision**
 - of angular tolerances, 449
 - dimension styles, 383
- preferred pitches, 627**
- preferred sizes, 463–464, 467–468**
- Preset Views (View Manager), 672**
- pressure angle, 627**
- Primary Units dialog box, 449**
- Primitive panel**
 - Box tool, 683, 686
 - Cylinder command, 682
- Print command (Application menu), 15**
- Printer/Plotter dialog box, 16**
- profile tolerances, 505–506**
- projected holes, representations of, 223**
- projection**
 - into auxiliary views
 - irregular surfaces, 337–340
 - rounded surfaces, 335–337
 - first-angle, 591–592
 - between orthographic views, 211
- Properties palette, 124–125, 496, 694**
 - limit tolerances, changing existing dimension to, 448
 - text justification, 60–61
- proportions, in sketches, 172–173**
- Ptype command, 57**

Q

- qleader command, 401–403, 499**
- Quadrant object snap, 96**
- quadrant-sensitive applications, 44**
- Quick Access Toolbar, 21–22**
 - Save command, 10, 21–22
 - Workspace Switching button, 669
- Quick Leader tool, 401–403**
- Quick Properties dialog box, 208**
- Quick tool (Dimensions panel), 399–400**

R

- rack-and-pinion steering, 645**
- racks, 644–645**
- radial dimensions, 369**

- Radians angle units, 13**
- radius dimensions**
 - altering default, 386–387
 - of circles, 42
 - creating, 385–387
 - overview of, 385
 - removing center mark from, 387
 - very large radii, 419
- Radius tool (Dimensions panel), 385–387**
 - overview of, 385
 - radius dimensions
 - altering default, 386–387
 - creating, 385–386
 - removing center mark from, 387
- Ray command (Draw panel), 38**
- RC (running and sliding fits), 462**
- RD (root diameter), 627**
- ream callouts, 232**
- Rectangle tool (Draw panel), 54, 689**
- rectangles, 54**
 - drawing, 54
 - exploding, 54
 - in L-shaped bracket drawing, 689
- Rectangular Array tool (Modify panel), 66**
- rectangular arrays, 66**
- rectangular dimensions, 454**
- rectangular views, of cylinders, 226**
- regardless of feature size (RFS), 489–492**
- release block, assembly drawings, 588–589**
- release date, 586**
- removed sectional views, 303**
- representations, of threads, 538–542**
 - detailed
 - definition of, 538
 - drawing, 538–539
 - internal, 539–540
 - schematic
 - definition of, 538
 - drawing, 540–542
 - internal representations, 541–542
 - simplified, 542
 - definition of, 538
 - drawing, 542
- resizing**
 - arrowheads, 289
 - cutting plane lines, 289
 - solid models, 682–683
- restoring UCSs (user coordinate systems), 680–681**
- revision block, assembly drawings, 586–587**
- revisions, 587**
- Revolve tool (Modeling panel), 718–719**

revolved solid objects, 718–719
RFS (regardless of feature size), 489–492
ribbon, 4. See also panels
rivets, 563
root diameter (RD), 627
roots, of threads, 535–536
Rotate Axis command, 686
Rotate command (Modify panel), 68–69, 104–105, 289, 332, 333–334, 632, 640
Rotate Faces tool (Solid Editing panel), 748–749
rotating objects, 68–69, 104–105
axis system, 330–331
faces, 748–749
lines, 334
UCS (user coordinate system) axes, 683–684
roughness, 466
rounded edges, castings model, 741–742
rounded shapes
external, 410
internal, 409
rounded surfaces
cylinders, 228–229
orthographic views, 217–220
definition of, 217–219
drawing problems, 219–220
projection into auxiliary views, 335–337
rounds, 234–235, 409
runout tolerances, 506–507
runouts, 234–236, 506–507

S

Save As command, 593, 595
Save command, 10, 21–22
Save Drawing As dialog box, 9–11, 21–22
saving
drawings, 9–11, 21–22
UCSs (user coordinate systems), 679–680
S-breaks, 303–305
scale, 105
of blocks, 109, 112–113, 567–568
defining, 586
of drawings, 377–378
Scale command (Grips menu), 103–104
schematic thread representations
definition of, 538
drawing, 540–542
internal representations, 541–542
scientific unit system, 11
screws, 559–561
drawing problems, 557–558
head shapes, 550–554

hexagon, 550–552
square, 552–554
overview of, 549
setscrews, 560–561
shoulder screws, 567–568
standard, 559–560

S-curves, constructing, 139–140

SE Isometric command (Visualize panel), 677, 739

secondary auxiliary views, 343–349

creating, 345–346
drawing problem, 346–348
of an ellipse, 348–349
overview of, 343–344

section lines, 296

sectional views

aligned sections, 300
auxiliary views, 341–342
breaks, 303–305
freehand lines, 303
S-breaks, 303–305
broken-out views, 302
of castings, 305
cutting plane lines, 287–290
drawing, 287–290
patterns for, 287
dimensioning, 417
drawing conventions, 301
drawing problem, 295
exercise problems, 306–323
gradients, 298
half and partial views, 302
hatching
hatch creation, 292–293
hatch patterns, 293–294
overview of, 290–294
holes in, 297–298
of internal thread representations, 544
locations of, 296–297
multiple sections, 300
offset sections, 299
overview of, 285–287
removed views, 303
section lines, 296
thread representations in, 539–540

Select Color dialog box, 208

Select File dialog box, 23

Select Linetype dialog box, 120, 208

Select Template dialog box, 3–5, 9, 19, 582–584, 674

setscrews, 560–561

shafts, 456–457, 461. See also tolerances

shape parameters, drawing problems for, 145–147

sheet number, stating on title block, 586

sheet size. See drawing limits

shoulder screws, 567–568

SI (International System of Units) symbols, 592

signatures, approval, 588–589

simplified thread representations, 542

definition of, 538

drawing, 542

sketching

curves, 173–174

drawing problems, 173–174, 179–180

exercise problems, 185–197

feathering, 171

graph paper for, 170

importance of, 169

isometric sketches

description of, 176–178

exercise problems, 186–189, 190–196

lines, 170

oblique sketches

description of, 180–181

exercise problems, 189

orientations for, 178, 184

pencils for, 170

personal style of, 169

perspective sketches

description of, 182–183

exercise problems, 186–189

proportions, 172–173

slanted surfaces, 210

auxiliary views of, 342

of cylinders, 228–229

holes in, 222–223

slots, sketching, 174

snap angle, 330–332, 339, 345, 348

Snap command, 4, 17–18, 20, 29, 89, 651, 681.

See also object snap modes (Osnap)

snap points, 34–35

definition of, 34

drawing problems, 35

Snapang, 330–332, 339, 345, 348

solid box

drawing, 677–678

UCSs (user coordinate systems), 677–678

solid cuts, indicating

hatch creation, 292–293

hatch patterns

selecting, 293–294, 296

spacing and angle of, 294

overview of, 290–292

Solid Editing panel, 744, 751

Copy Edges tool, 750–751

Copy Faces tool, 750

Extrude Faces tool, 745–747

Imprint tool, 751

Intersect tool, 724

Move Faces tool, 747

Offset Faces tool, 747–748

Rotate Faces tool, 748–749

Subtract tool, 740

Taper Faces tool, 749

Union tool, 735, 740

solid modeling

boxes, 706–709

in combined solid objects drawing, 727–728, 730

creating with thickness, 696

cubes, 708

drawing from given dimensions, 707–708

drawing with corner option, 706

dynamic grips, 708–709

spheres, 709

three-dimensional, 202–203

visual styles, 706–707

castings model, 737–742

holes, 740

overview of, 737

polylines, 738

rounded edges, 741–742

shapes, drawing, 737–738

shapes, extruding, 739

combined solid objects drawing, 727–731

boxes, 727–728, 730

drawing setup, 727

internal open volume, 728–729

top cutout, 729

wedges, 730–731

cones, 711–712

drawing with circular bases, 712

drawing with elliptical bases, 711–712

cylinders, 709–711

drawing with circular bases, 709–710

drawing with elliptical bases, 710–711

displaying database information for, 743

displaying structural characteristics for, 744

edges

copying, 750–751

editing, 744

extracting, 751

imprinting objects, 751

polygon edge distance, 56

editing, 682–683

exercise problems, 752–778

- extrusion, 716–718, 739
- faces
 - basic extruding of, 745
 - copying, 750
 - editing, 744–751
 - extruding along a path, 745–746
 - extruding multiple, 746–747
 - moving, 747
 - offsetting, 747–748
 - rotating, 748–749
 - tapering, 749
- helix, 719–720
- intersecting solids model, 732–736
 - completing, 735
 - cone, drawing, 733
 - cylinder, drawing, 734–735
 - drawing setup, 732
 - viewport configuration, 736
- intersection, 724
- lofts, 721–722
- overview of, 705–706
- resizing, 683
- revolved solid objects, 718–719
- spheres, 709
- subtraction, 722, 723–724
- sweeps, 720
- thread representations in, 742–743
- torus, 715–716
- UCSs (user coordinate systems), 677–682, 725–726
 - creating, 679
 - defining, 681–682
 - definition of, 677
 - drawing problems, 686–687, 688–690
 - restoring, 680–681
 - returning to WCS (world coordinate system), 680
 - saving, 679–680
 - UCS icon, 5
- unions, 722–723
- wedges, 713–715
 - aligning with existing wedges, 714–715
 - center points, 713–714
 - combined solid objects, 730–731
 - corner points, 713
- spacing, of hatch patterns, 294**
- Sphere tool (Modeling panel), 709**
- spheres, 709**
- spined polylines, creating from given polyline, 135**
- spirals, 143**
- Spline command, 657**
- Spline Fit tool (Draw panel), 51, 655**
- splines, 50–51, 136–137**
- spotface callouts, 233**
- spotfacing, 235**
- springs, 564–566**
 - compression, 564, 566
 - extension, 564–566
 - overview of, 564
 - torsional, 564
- spur gears**
 - center distance between, 635
 - combining, 636–637
 - drawing problems, 629–634
 - drawings of, 628–629
 - English and metric units for, 626–627
 - meshing spur gears, 632, 635–636
 - pinions, 644
 - selecting, 633–634
 - terminology for, 626–627
- square keys, 414, 562–563**
- square threads**
 - definition of, 544–545
 - external, 545–546
- square-shaped heads, 552–554**
- standard A3 (metric) paper size, aligning drawing limits with, 15–16**
- standard fit tolerances, 450**
 - inch values, 462–463
 - drawing problems, 463
 - fits defined by inch values, 462–463
 - metric units, 458–461
 - clearance fits, 460–461, 482–484
 - hole and shaft basis, 461–462
 - interference fits, 460–461, 483, 484
 - locational fit, 483, 484
 - nominal sizes, 461
 - transition fits, 460–461, 484–485
 - metric values, 461–462
- standard screws, 559–560**
- standard sizes, 463–464, 467–468**
- star washers, 561**
- starting drawings, 2–5, 8–9**
 - Create New Drawing dialog box, 8–9
 - drawing limits, 13–16, 20
 - drawing units, 11–13, 19
 - grid and snap values, 17–18, 20
 - precision of, 19
 - Save Drawing As dialog box, 9–11, 21–22
 - Select Template dialog box, 9
- steering, rack-and-pinion, 645**
- Stock Drive Products website, 645**
- straightness tolerances, 489–492, 494–495**

MMC (maximum material condition), 489–492
 overview of, 489
 RFS (regardless of feature size), 489–492
Stretch command (Grips menu), 103–104
**structural characteristics, displaying for drawn
 solid objects, 744**
studies, tolerance, 453–454
studs, 550
Subtract tool
 Edit panel, 722, 723–724
 Solid Editing panel, 740
subtraction, 722, 723–724
Sun property (Layers), 118
surface control symbols, 467
surfaces
 cylinders, 228–229
 irregular
 definition of, 217
 dimensioning, 410–411
 orthographic views, 230–232
 oblique
 definition of, 215
 drawing problems, 217
 holes in, 224–226
 orthographic views, 215–217
 profile tolerances, 505–506
 rounded
 definition of, 217–219
 drawing problems, 219–220
 orthographic views, 217–220
 slanted, 210
 of cylinders, 228–229
 holes in, 222–223
 surface finishes, 465–468
 datum surfaces, 465
 definition of, 465
 design applications of, 466
 lay, 466
 roughness, 466
 surface control symbols, 467
 surface texture, 466
Surveyor angle units, 13
SW Isometric view, 685
Sweep tool (Modeling panel), 720
sweeps, 720
Symbol dialog box, 495
symbols, 61
 dimensioning, 414
 surface control, 467
**Symbols and Arrows (Modify Dimension Style
 dialog box), 389**
symmetry

dimensioning and, 415
 symmetrical tolerances, 445
system variables
 definition of, 693
 Snapang, 330–332, 339, 345, 348
 Thickness, 693–698
 creating objects with, 696–698
 drawing curves with, 695–696
 drawing hexagons/arcs with, 693–694
 exercise problems, 699–703
 overview of, 693

T

Table command (Annotate panel), 74–77, 589
Table Style dialog box, 75, 76
tables, 74–77
Tangent object snap, 97, 236, 653, 656
**tangent tangent radius, drawing circles with,
 44**
Taper Faces tool (Solid Editing panel), 749
tapering faces, 749
tapping bits, 543
Teflon bushings, 647
templates, 2–5. See also 3D modeling
 acad, 2–5, 25, 670
 acad3D, 669
 3D Modeling mode, accessing, 676–677
 perspective and parallel grids, 673–675
 acadiso, 19, 25, 212
 acadiso3D, 27, 669, 673–675
 for assembly drawings, 582–584
 selecting, 2, 19
text, 57–61
 angle of, 404
 default dimension text, changing, 372–374
 with Mtext option, 373–374
 with text option, 372–373
 dimension, 404
 editing, 137
 height/placement, 378–380
 multiline, 57–60
 creating, 57–58
 symbol options, 61
 text color, 61
 text font, 59–60
 text height, 59
 text justification, 60–61
 options for, 378–380
 aligned dimensions, 385
 angular dimension and tolerance, 449
 default dimension text, changing, 372–373
 plus or minus tolerances, creating, 443–444

Text Angle tool (Dimensions panel), 404

Text Override tool, 444, 484–485

Text tool, 583

texture, surface, 466

thickness command, 693, 695, 696, 698

Thickness system variable, 50, 693–698

creating objects with, 696–698

boxes, 696

cylinders, 697–698

returning to original settings, 698

UCSs (user coordinate systems), 697

drawing curves with, 695–696

drawing hexagons/arcs with, 693–694

exercise problems, 699–703

overview of, 693

thin-walled objects, 238–240

definition of, 238–239

drawing problems, 239–240

third-angle projection, 200–201

threads

acme

definition of, 544–545

external, 544–545, 546–548

American National, 544–545

bolts, 548, 550–554

callouts, 536–537

English units, 536–537

metric units, 536

crests of, 535–536

exercise problems, 569–578

head shapes, 550–554

hexagon, 550–552

square, 552–554

knuckle, 544–545

major/minor diameter of, 535–536

nuts, 548

orthographic views of, 542–543

representations of, 538–542

detailed, 538–540

drawing problems, 557–559

internal, 539–540

schematic, 538, 540–542

simplified, 538, 542

in solid models, 742–743

roots of, 535–536

screws, 550–554

sectional views of, 544

square

definition of, 544–545

external, 545–546

studs, 550

terminology for, 535–536

three points, drawing circles with, 43

title block, assembly drawings, 584–586

attributes of, 132–134

company name, 586

drawing numbers, 586

drawing titles, 585

overview of, 584–585

release date, 586

scale, 586

sheet number, 586

Tolerance tool (Dimensions panel), 494, 498

tolerances, 405, 450

angular, 448–449

creating, 449

precision of, 449

baseline dimensions, 451–453

bilateral, 441

chain dimensions, 451–453

definition of, 441

deviation, 445–446

double dimensioning, 450–451

drawing problems, 461–462

shaft diameters, 457–458

standard fits (inch values), 463

standard fits (metric values), 461–462

exercise problems, 472–485

fixed conditions, 469–471

floating conditions, 468–469

geometric

angularism, 504–505

circularity, 492

creating, 493–500

cylindricity, 493

defined tolerance zone, 487

drawing problems, 511–512, 513–515

exercise problems, 516–533

fixed fasteners, 512–515

flatness, 488

floating fasteners, 510–512

locational tolerance, 502

overview of, 487–488

parallelism, 504

perpendicularity, 502–503

positional tolerance, 495–496, 498–499, 508–509

profiles, 505–506

runouts, 506–507

straightness, 489–492

tolerances of form, 488–493

tolerances of orientation, 500

virtual conditions, 509–510

hole basis, 461

hole locations, 455
hole size, 471
limit, 446–448, 497–498
 changing existing dimension to, 448
 creating with Dimension Style Manager, 447
 example of, 446–447
nominal dimensions, 468
plus or minus, 443–446
 creating with Dimension Style Manager, 444–446
 creating with Dtext, 444
 creating with Text option, 443–444
 creating with Text Override tool, 444
 definition of, 441
 overview of, 443
preferred and standard sizes, 463–464, 467–468
rectangular dimensions, 454
shaft basis, 461
shafts, choosing, 456–457
standard fits (inch values), 462–463
 drawing problems, 463
 fits defined by inch values, 462–463
standard fits (metric values), 458–462
surface finishes, 465–468
 datum surfaces, 465
 definition of, 465
 design applications of, 466
 lay, 466
 roughness, 466
 surface control symbols, 467
 surface texture, 466
symmetrical, 445
tolerance block, 588
tolerance expressions, 442–443
tolerance studies, 453–454
unilateral, 441

Tool Palettes, 567–568
 accessing, 567
 blocks
 changing scale of, 567–568
 modifying, 568
 shoulder screws, creating, 567

tooltips, 6
top drawings. See assembly drawings
Top view, 675
torsional springs, 564
torus, 715–716
Torus tool (Modeling panel), 715–716
transferring lines between views, 332–335
 drawing problem, 334–335
 grip points, 333–334

 line length measurement, 332–333
 line rotation, 334
transition fits, 460–461
 exercise problems, 484–485
 geometric tolerances, 530, 531
Trim command (Modify panel), 69–70, 217, 237, 631, 738
trimming objects, 69–70
Tutorial-iMfg template, 582–584
two points, drawing circles with, 43

U

UCS dialog box, 679
UCSs (user coordinate systems), 677–682
 creating, 679, 697
 defining, 681–682
 definition of, 677
 drawing problems, 686–687, 688–690
 restoring saved UCS, 680–681
 returning to WCS (world coordinate system), 680
 saving, 679–680
 solid box, drawing, 677–678
 solid modeling, 725–726
 UCS axis, rotating, 683–684
 UCS icon, 5, 685, 686

underscore (_), 8
Undo tool, 34
Unified National Coarse (UNC), 536
Unified National Extra Fine (UNEF), 536
Unified National Fine (UNF), 536
Unified National (UN), 536
uniform acceleration/deceleration, 654–655
uniform hatch patterns, 292
uniform motion, 652
unilateral tolerance, 441
Union tool, 687, 722–723, 735, 740
unions, 687, 722–723, 735, 740
units, drawing, 380–381
units systems. See drawing units
user coordinate systems. See UCSs (user coordinate systems)

V

Ver option (Construction Line command), 40–41
vertical dimensions, creating, 371
vertical lines, 40–41
view command, 672
View Cube, 672, 688
View Manager dialog box, 672

viewpoint, changing, 672

Viewport Configuration (Model Viewports panel), 690

Viewport Configuration panel, 736

Viewport with sun property (Layers), 118

viewports

configuration of, 690–691

in intersecting solids drawing, 736

Viewports dialog box, 691

views

auxiliary

axis system, rotating, 330–331

definition of, 327

drawing problems, 331–332, 334–340, 346–348

exercise problems, 350–366

irregular surfaces, 337–340

lines, transferring between views, 332–335

oblique surfaces, 342–343

overview of, 327–328

partial views, 340–341

projecting information from normal view into, 328–331

rounded surfaces, 335–337

secondary views, 343–349

sectional views, 341–342

slanted surfaces, 342

orthographic

of 3D objects, 690–692

castings, 234–238

compound lines, 213–214

creating, 692

cylinders, 226–230

definition of, 199

dimensioning, 417–418

drawing problems, 212–213, 214, 217, 219–220, 227, 228–229, 231–232

exercise problems, 251–283

first-angle projection, 249–250, 591–592

hidden lines, 203–209

hole callouts, 232–234

holes, 220–226

in intersecting solids drawing, 736

irregular surfaces, 217, 230–232

modification of existing parts, 245–249

oblique surfaces, 215–217

precedence of lines in, 209–210

projection between, 211

rounded surfaces, 217–220

slanted surfaces in, 210

thin-walled objects, 238–240

third-angle projection, 200–201, 249–250

for three-dimensional box drawing, 202–203

plan, 670

sectional

aligned sections, 300

breaks, 303–305

broken-out views, 302

of castings, 305

cutting plane lines, 287–290

dimensioning, 417

drawing conventions, 301

drawing problem, 295

exercise problems, 306–323

gradients, 298

half and partial views, 302

hatching, 290–294

holes in, 297–298

locations of, 296–297

multiple sections, 300

offset sections, 299

overview of, 285–287

removed views, 303

section lines, 296

thread representations in, 539–540

transferring lines between, 332–335

drawing problem, 334–335

grip points, 333–334

line length measurement, 332–333

line rotation, 334

virtual conditions, 509–510

visé model. See minivisé model

visual errors, 3D modeling, 688

visual styles, 684

2D Wireframe, 684, 688

changing, 683–684

Conceptual, 678, 685, 686, 695

Visualize panel, SE Isometric command, 677, 739

W

washers, 561–562

lock, 561

plain, 561–562

star, 561

Wblock command, 115–117, 467, 535, 538, 539–541, 564

WCS (world coordinate system)

background color, 671

default settings for, 669–670

returning to 2D WCS, 675, 680

returning to original orientation, 673

viewpoint, 672

Wedge tool (Modeling panel), 713–715, 731

- alignment with existing wedges, 714–715
- center points, 713–714
- corner points, 713

wedges, 713–715

- aligning with existing wedges, 714–715
- center points, 713–714
- in combined solid objects drawing, 730–731
- corner points, 713

whole depth, 627

windows, command line, 6–7

Woodruff keys, 562–563

working depth, 627

working drawings. See assembly drawings; detail drawings

Workspace Switching tool, 669, 676, 734

workspaces

- 3D Modeling. **See** 3D modeling
- accessing, 669
- definition of, 669
- Drafting & Annotation, 676

- Drawing & Annotation, 676
- switching, 734

World command (Coordinates panel), 698

world coordinate system. See WCS (world coordinate system)

World tool (Coordinates panel), 735

worms and worm gears, 641–642

Write Block dialog box, 115–117

X

X Axis Rotate UCS command (Coordinates panel), 734

Xline option (Construction Line command), 38–40

Y-Z

Y Axis Rotate UCS command (Coordinates panel), 734

zero suppression, 382

zoom command, 16, 739