

Table of Contents

Dedication	iii
Preface	xv
Introduction	xix

Chapter 1: Drawing Sketches for the Solid Models

The Sketching Environment	1-2
Starting a New Document in SolidWorks 2004	1-3
Setting Up the Document Options	1-7
Modifying the Dimensioning Standards	1-8
Modifying the Linear and Angular Units	1-8
Modifying Snap and Grid Settings	1-9
Learning About Sketcher Terms	1-11
Origin	1-12
Inferencing Lines	1-12
Select Tool	1-13
Drawing Lines	1-13
Drawing Continuous Lines	1-14
Drawing Individual Lines	1-14
Drawing Tangent or Normal Arcs Using the Line Tool	1-17
Drawing Construction Lines or Centerlines	1-18
Drawing Circles	1-18
Drawing a Construction Circle	1-19
Drawing Arcs	1-20
Drawing Tangent/Normal Arcs	1-20
Drawing Centerpoint Arcs	1-22
Drawing 3 Point Arcs	1-23
Drawing Rectangles	1-25
Drawing Parallelogram	1-26
Drawing a Rectangle at an Angle	1-26
Drawing Parallelograms	1-26
Drawing Polygons	1-27
Drawing Splines	1-29
Drawing Points	1-30
Drawing Ellipses	1-31
Drawing Elliptical Arcs	1-32
Drawing Parabolic Curve	1-33
Drawing Display Tools	1-34
Zoom to Fit	1-35
Zoom Area	1-35
Zoom In/Out	1-35
Zoom to Selection	1-35

Evaluation copy. Do not reproduce. For more information, visit www.cadcam.com

Pan	1-36
Redraw	1-36
Deleting the Sketched Entities	1-36
Tutorial 1	1-36
Tutorial 2	1-47
Tutorial 3	1-51
Tutorial 4	1-56
Self-Evaluation Test	1-61
Review Questions	1-61
Exercise 1	1-62
Exercise 2	1-63

Chapter 2: Editing and Modifying the Sketches

Editing the Sketched Entities	2-2
Trimming the Sketched Entities	2-2
Extending the Sketched Entities	2-3
Filletting the Sketched Entities	2-4
Chamfering the Sketched Entities	2-5
Offsetting the Sketched Entities	2-7
Mirroring the Sketched Entities	2-9
Moving, Rotating, and Scaling the Sketched Entities	2-11
Creating Patterns	2-15
Creating Linear Pattern of the Sketched Entities	2-16
Creating Circular Pattern of the Sketched Entities	2-21
Editing Patterns	2-25
Writing Text in the Sketching Environment	2-26
Modifying Sketched Entities	2-27
Modifying a Sketched Line	2-27
Modifying a Sketched Circle	2-27
Modifying a Sketched Arc	2-27
Modifying a Sketched Polygon	2-28
Modifying a Spline	2-28
Modifying a Sketched Point	2-29
Modifying an Ellipse or an Elliptical Arc	2-29
Modifying a Parabola	2-29
Dynamically Modifying and Coping the Sketched Entities	2-29
Splitting the Sketched Entity	2-30
Tutorial 1	2-30
Tutorial 2	2-40
Tutorial 3	2-46
Self-Evaluation Test	2-52
Review Questions	2-52
Exercise 1	2-53
Exercise 2	2-54
Exercise 3	2-55
Exercise 4	2-56

Chapter 3: Adding Relations and Dimensions to the Sketches

Adding Geometric Relations to the Sketch	3-2
Adding Relations Using the Add Relations PropertyManager	3-2
Automatic Relations	3-7
Dimensioning the Sketch	3-8
Horizontal/Vertical Dimensioning	3-10
Aligned Dimensioning	3-20
Angular Dimensioning	3-20
Diameter Dimensioning	3-22
Radius Dimensioning	3-23
Linear Diameter Dimensioning	3-23
Ordinate Dimensioning	3-24
Additional Dimensioning Options	3-26
Concept of Fully Defined Sketch	3-27
Fully Defined	3-27
Over Defined	3-28
Under Defined	3-28
Dangling	3-28
No Solution Found	3-28
Invalid Solution Found	3-29
Sketch Dimension or Relation Status	3-29
Deleting the Over Defining Dimensions	3-30
Displaying and Deleting Relations	3-31
Viewing and Examining Relations	3-35
Opening an Existing File	3-36
Tutorial 1	3-38
Tutorial 2	3-48
Tutorial 3	3-55
Self-Evaluation Test	3-61
Review Questions	3-62
Exercise 1	3-63
Exercise 2	3-63
Exercise 3	3-64

Chapter 4: Advanced Dimensioning and Base Feature Options

Advanced Dimensioning Techniques	4-2
Autodimension the Sketches	4-2
Dimensioning of the True Length of an Arc	4-5
Measuring Distances and Viewing Section Properties	4-6
Measuring Distances	4-6
Determining Section Properties of Closed Sketches	4-10
Creating the Base Feature by Extruding the Sketches	4-11
Creating the Solid Extruded Features	4-12
Creating the Thin Extruded Features	4-17
Creating the Base Features by Revolving the Sketches	4-20

Creating the Solid Revolve Features	4-21
Creating the Thin Revolved Features	4-24
Dynamically Rotating the View of the Model	4-26
Rotating the View Freely in 3D Space	4-26
Rotating the View Around a Selected Vertex, Edge, or Face	4-26
Modifying the View Orientation	4-27
Restoring Previous View	4-29
Display Modes of the Model	4-29
Wireframe	4-29
Hidden Lines Visible	4-29
Hidden Lines Removed	4-30
Shaded With Edges	4-30
Shaded	4-30
Shadows In Shaded Mode	4-30
Perspective	4-30
Assigning Material and Texture to the Model	4-32
Assigning Material to a Model New	4-32
Assigning Textures to a Model New	4-33
Tutorial 1	4-34
Tutorial 2	4-41
Tutorial 3	4-46
Self-Evaluation Test	4-52
Review Questions	4-53
Exercise 1	4-54
Exercise 2	4-55

Chapter 5: Creating Reference Geometries

Importance of the Sketching Planes	5-2
Reference Geometry	5-2
Creating New Planes	5-5
Creating Reference Axis	5-10
Creating Reference Points New	5-14
Creating Reference Coordinate System	5-16
Other Boss/Base Options	5-17
End Condition	5-17
Direction of Extrusion New	5-21
Modeling using the Contour Selection Method Enhanced	5-21
Creating the Cut Features	5-25
Creating Extruded Cuts	5-25
Handling Multiple Bodies in Cut Feature	5-28
Creating Revolved Cuts	5-30
Concept of the Feature Scope	5-31
Tutorial 1	5-32
Tutorial 2	5-38
Tutorial 3	5-45
Self-Evaluation Test	5-54

Review Questions	5-55
Exercise 1	5-56
Exercise 2	5-57

Chapter 6: Advanced Modeling Tools-I

Advanced Modeling Tools	6-2
Creating Simple Hole	6-2
Creating the Standard Holes Using Hole Wizard	Enhanced 6-3
Creating Fillets	Enhanced 6-15
Selection Methods	6-31
Creating Chamfer	Enhanced 6-33
Creating the Shell Feature	Enhanced 6-37
Creating Wrap Feature	New 6-41
Tutorial 1	6-43
Tutorial 2	6-53
Self-Evaluation Test	6-62
Review Questions	6-63
Exercise 1	6-64
Exercise 2	6-65

Chapter 7: Advanced Modeling Tools-II

Advanced Modeling Tools	7-2
Creating Mirror Feature	7-2
Creating Linear Pattern Feature	7-8
Creating Circular Pattern Feature	7-15
Creating Sketch-driven Pattern	7-19
Creating Curve-driven Pattern	7-21
Creating Table-driven Pattern	7-24
Creating the Rib Feature	7-26
Creating the Dome Feature	Enhanced 7-31
Creating the Shape Feature	7-35
Displaying the Section View of the Model	Enhanced 7-37
Tutorial 1	7-39
Tutorial 2	7-48
Tutorial 3	7-54
Self-Evaluation Test	7-65
Review Questions	7-65
Exercise 1	7-66
Exercise 2	7-68
Exercise 3	7-69

Chapter 8: Editing Features

Editing the Features of the Model	8-2
Editing Using Edit Feature Option	8-2

Editing the Sketches of the Sketched Features	8-4
Changing the Sketch Plane of the Sketches	8-4
Editing by Double-clicking the Entities and Features	8-4
Editing using the Move/Size Features	8-6
Editing the Sketches with the Move/Size Features Tool Active	8-9
Editing the Features and Sketches by Cut, Copy, and Paste	8-10
Cut, Copy, and Paste Features and Sketches from One Document to the other	8-11
Coping Features Using Drag and Drop	8-11
Deleting the Features	8-13
Deleting the Bodies	8-13
Suppressing the Features	8-14
Unsuppressing the Features	8-15
Unsuppressing the Features with Dependents	8-15
Hiding the Bodies	8-15
Move/Copy Bodies	8-16
Reordering the Features	8-18
Rolling Back the Model	8-20
Renaming the Features	8-21
Creating Folders in the FeatureManager Design Tree	8-21
What's Wrong? Functionality	8-22
Tutorial 1	8-23
Tutorial 2	8-31
Tutorial 3	8-38
Self-Evaluation Test	8-46
Review Questions	8-47
Exercise 1	8-48
Exercise 2	8-49

Chapter 9: Advanced Modeling Tools-III

Advanced Modeling Tools	9-2
Creating the Sweep Feature Enhanced	9-2
Creating Cut Sweep Features Enhanced	9-14
Creating Loft Feature Enhanced	9-15
Creating 3D Sketches	9-25
Editing 3D Sketches	9-27
Creating Curves	9-27
Extruding a 3D Sketch New	9-40
Creating Draft Features	9-40
Creating the Deform Feature New	9-42
Tutorial 1	9-47
Tutorial 2	9-51
Tutorial 3	9-55
Self-Evaluation Test	9-62
Review Questions	9-62
Exercise 1	9-63
Exercise 2	9-65

Chapter 10: Assembly Modeling-I

Assembly Modeling	10-2
Types of Assembly Design Approach	10-2
Creating Bottom-Up Assemblies	10-4
Placing Components in the Assembly File	10-4
Assembling the Components	10-9
Creating Top-down Assemblies	10-26
Creating the Component in the Top-down Assembly	10-27
Moving the Individual Components	10-28
Moving the Individual Components by Dragging NEW	10-28
Moving the Individual Components using the Move Component Tool	10-28
Rotating the Individual Components	10-29
Rotating the Individual Components by Dragging NEW	10-29
Rotating the Individual Components using the Rotate Component Tool	10-30
Moving and Rotating an Individual Components using the Triad NEW	10-30
Tutorial 1	10-32
Tutorial 2	10-47
Self-Evaluation Test	10-52
Review Questions	10-53
Exercise 1	10-54

Chapter 11: Assembly Modeling-II

Advanced Assembly Mates	11-2
Applying the Symmetric Mate	11-2
Applying the Cam Mate	11-2
Applying Gear Mate NEW	11-4
Applying Limit Mate NEW	11-4
Creating Subassemblies	11-5
Bottom-up Subassembly Design	11-5
Top-down Subassembly Design	11-5
Deleting Components and Subassemblies	11-6
Editing Assembly Mates	11-7
Replacing the Mated Entities	11-8
Editing the Components	11-9
Editing the Subassemblies	11-10
Dissolving the Subassembly	11-11
Replacing the Components	11-11
Creating Patterns of the Components in an Assembly	11-14
Feature-driven Pattern	11-14
Local Pattern	11-15
Mirroring the Components	11-16
Simplifying the Assemblies using Visibility Options	11-18
Hiding the Components	11-19
Suppressing the Components	11-19
Changing the Transparency Conditions	11-20

Checking the Interferences in the Assembly	11-21
Creating the Assemblies for Mechanism	11-22
Analyzing the Collisions using the Collision Detection Tool	11-23
Creating the Explode State of the Assembly	11-25
Creating Systematic Explode State	11-26
Creating the Explode Line Sketch	11-28
Tutorial 1	11-30
Tutorial 2	11-41
Self-Evaluation Test	11-46
Review Questions	11-47
Exercise 1	11-48

Chapter 12: Working With Drawing Views-I

The Drawing Mode	12-2
Starting a Drawing Document 	12-2
Starting a New Drawing Document Using the New SolidWorks Document Dialog box	12-2
Starting a New Drawing Document From Within the Part/Assembly Document 	12-4
Type of Views	12-5
Generating Standard Drawing Views	12-6
Generating Model Views 	12-6
Generating the Three Standard Views	12-9
Generating Standard Views Using the Relative View Tool	12-11
Generating Standard Views Using the Predefined View Tool	12-12
Generating the Derived Views	12-15
Generating Projected Views	12-15
Generating Section Views	12-16
Generating Aligned Section Views	12-21
Generating Broken-out Section Views	12-23
Generating Auxiliary Views	12-25
Generating Detail Views	12-27
Cropping Drawing Views	12-29
Generating Broken View	12-30
Alternate Position View	12-32
Creating the Drawing view of the Exploded State of the Assembly	12-34
Working with Interactive Drafting in SolidWorks	12-35
Editing and Modifying the Drawing Views	12-35
Changing the View Orientation	12-36
Changing the Scale of the Drawing Views	12-36
Deleting the Views	12-36
Rotating the Drawing View	12-36
Modifying the Hatch Pattern of the Section View 	12-37
Tutorial 1	12-39
Tutorial 2	12-45
Self-Evaluation Test	12-50

Review Questions	12-51
Exercise 1	12-52

Chapter 13: Working With Drawing Views-II

Adding Annotations to the Drawing Views	13-2
Generating the Annotations using the Model Items Tool	13-2
Adding the Reference Annotations	13-5
Editing the Annotations	13-21
Adding the BOM in the Drawing Enhanced	13-21
Adding Balloons to the Drawing Views	13-23
Adding Balloons using the Balloon Tool	13-23
Adding Balloons using the AutoBalloon Tool New	13-25
Adding New Sheets to the Drawing Views	13-26
Editing the Sheet Format	13-27
Creating a User-Defined Sheet Format	13-27
Tutorial 1	13-28
Tutorial 2	13-34
Self-Evaluation Test	13-39
Review Questions	13-40
Exercise 1	13-41

Student Project	1
------------------------	---

Index	1
--------------	---

About the Author

Sham Tickoo is a professor of Mechanical Engineering Technology at Purdue University Calumet. He has been in the education and training since 1981 and joined Purdue University in the year 1987. Since then he has been teaching drafting and design, AutoCAD, AutoLISP, 3ds max, 3ds viz, Mechanical Desktop, Pro/Engineer, Autodesk Inventor, SolidWorks etc.

Prof. Tickoo is also one of the best known authors of CAD/CAM books in the world market. He has authored a number of books on the latest CAD/CAM technology. He is one of the very few authors in the world whose books are translated and sold in the countries like Italy, China, Philippines, and Russia. The list of some of his books is given below:

SolidWorks for Engineers and Designers Release 2004, dreamtech PRESS, India
SolidWorks for Designers Release 2003, CADCIM Technologies, USA
SolidWorks for Engineers and Designers Release 2003, dreamtech PRESS, India
Pro/ENGINEER Wildfire for Designers, CADCIM Technologies, USA
Pro/ENGINEER Wildfire for Engineers & Designers, dreamtech PRESS, India
Pro/ENGINEER for Designers Release 2001, CADCIM Technologies, USA
Autodesk Inventor for Designers Release 6, CADCIM Technologies
Autodesk Inventor for Engineers & Designers Release 6, dreamtech PRESS, India
Autodesk Inventor for Designers Release 5, CADCIM Technologies
AutoCAD: A Problem Solving Approach, Delmar Publishers, USA
AutoCAD LT: A Problem Solving Approach, Delmar Publishers, USA
Customizing AutoCAD, Delmar Publishers, USA
3D Studio VIZ, Goodheart-Willcox Publishers, USA
3D Studio MAX, Goodheart-Willcox Publishers, USA
Mechanical Desktop Instructor, McGraw-Hill, USA
AutoCAD 2000 Tecniche Avanzate, APOGeO education, Italy
AutoCAD 2000 Fondamenti, APOGeO education, Italy
AutoCAD 2000: A Problem Solving Approach, China Machine Press, China
Customizing AutoCAD 2000, China Machine Press, China
AutoCAD 2002, Russian Edition

In addition to his academic career, he has been employed as a design engineer, quality control engineer, software developer, and has also worked as a consultant for a number of companies in US and Canada.

Prof. Tickoo has developed a software package called “SMLayout” that has been registered by Autodesk Inc. as a third party software product. This software package is currently being used by some companies in Canada, United States, and Columbia. SMLayout is written in the AutoLISP programming language. It generates flat layout drawing of some complicated geometric shapes like: transitions, cones, cone-cylinder intersections, sphere-cylinder intersections, and cylinder-cylinder intersections. This software package addresses the needs of steel fabricators and manufacturers of sheet metal products. The programs also provide a dimensioning option for automatic dimensioning of the drawing.

In March 2000, Prof. Tickoo was issued a US patent for a product called “Self Adjusting Cargo Organizer” that he designed for vehicles.