



# Table of Contents

<b>Dedication</b>	iii
<b>Preface</b>	xv
<b>Introduction</b>	xix

## Chapter 1: Drawing Sketches for the Solid Models

The Sketching Environment	1-2
Starting a New Session of SolidWorks 2005 	1-3
Starting a New Document in SolidWorks 2005	1-6
Setting Up the Document Options	1-9
Modifying the Dimensioning Standards	1-9
Modifying the Linear and Angular Units	1-10
Modifying the Snap and Grid Settings	1-11
Learning About Sketcher Terms	1-13
Origin	1-13
Inferencing Lines	1-13
Select Tool	1-14
Selecting the Entities Using the Box Selection	1-14
Selecting the Entities Using the Cross Selection	1-14
Drawing Lines 	1-15
Drawing Chain of Continuous Lines	1-16
Drawing Individual Lines	1-17
Drawing Tangent or Normal Arcs Using the Line Tool	1-19
Drawing Construction Lines or Centerlines 	1-20
Drawing Circles 	1-21
Drawing Circles by Defining its Centerpoint	1-21
Drawing Circles by Defining Three Points	1-21
Drawing a Construction Circle	1-22
Drawing Arcs	1-23
Drawing Tangent/Normal Arcs	1-23
Drawing Centerpoint Arcs	1-25
Drawing 3 Point Arcs	1-26
Drawing Rectangles	1-28
Drawing Parallelogram	1-29
Drawing a Rectangle at an Angle	1-29
Drawing Parallelograms	1-29
Drawing Polygons	1-30
Drawing Splines	1-32
Placing Sketched Points	1-33
Drawing Ellipses	1-34
Drawing Elliptical Arcs	1-34
Drawing Parabolic Curves	1-35

Drawing Display Tools	1-36
Zoom to Fit	1-37
Zoom Area	1-37
Zoom In/Out	1-37
Zoom to Selection	1-38
Pan	1-38
Redraw	1-38
Deleting the Sketched Entities	1-38
Tutorial 1	1-39
Tutorial 2	1-48
Tutorial 3	1-52
Tutorial 4	1-57
Self-Evaluation Test	1-62
Review Questions	1-62
Exercise 1	1-63
Exercise 2	1-64

## Chapter 2: Editing and Modifying the Sketches

Editing the Sketched Entities	2-2
Trimming the Sketched Entities	2-2
Extending the Sketched Entities	2-6
Filleting the Sketched Entities	2-6
Chamfering the Sketched Entities	2-8
Offsetting the Sketched Entities	2-10
Mirroring the Sketched Entities	2-11
Mirroring While Sketching (Dynamic Mirror Entities)	2-12
Moving, Rotating, and Scaling the Sketched Entities	2-14
Moving or Copying the Sketched Entities	2-15
Rotating or Copying the Sketched Entities	2-16
Scaling or Copying the Sketched Entities	2-18
Creating Patterns	2-19
Creating Linear Pattern of the Sketched Entities	2-19
Creating Circular Pattern of the Sketched Entities	2-23
Editing Patterns	2-27
Writing Text in the Sketching Environment	2-27
Modifying Sketched Entities	2-29
Modifying a Sketched Line	2-29
Modifying a Sketched Circle	2-29
Modifying a Sketched Arc	2-29
Modifying a Sketched Polygon	2-29
Modifying a Spline	2-30
Modifying a Sketched Point	2-31
Modifying an Ellipse or an Elliptical Arc	2-31
Modifying a Parabola	2-32
Dynamically Modifying and Coping the Sketched Entities	2-32
Splitting the Sketched Entity	2-32

Tutorial 1	2-33
Tutorial 2	2-41
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	2-55

### Chapter 3: Adding Relations and Dimensions to the Sketches

Adding Geometric Relations to the Sketches	3-2
Adding Relations Using the Add Relations PropertyManager	3-2
Automatic Relations	3-7
Dimensioning the Sketch	3-7
Horizontal/Vertical Dimensioning	3-9
Aligned Dimensioning	3-19
Angular Dimensioning	3-20
Diameter Dimensioning	3-22
Radius Dimensioning	3-22
Linear Diameter Dimensioning	3-23
Ordinate Dimensioning	3-23
Additional Dimensioning Options	3-25
Concept of Fully Defined Sketch	3-26
Fully Defined	3-26
Over Defined	3-27
Under Defined	3-27
Dangling	3-27
No Solution Found	3-27
Invalid Solution Found	3-28
Sketch Dimension or Relation Status	3-28
Deleting the Over Defining Dimensions	3-29
Displaying and Deleting Relations	3-30
Opening an Existing File	3-34
Tutorial 1	3-36
Tutorial 2	3-45
Tutorial 3	3-53
Self-Evaluation Test	3-59
Review Questions	3-60
Exercise 1	3-61
Exercise 2	3-61
Exercise 3	3-62

Enhanced

## Chapter 4: Advanced Dimensioning Techniques and Base Feature Options

Advanced Dimensioning Techniques	4-2
Autodimension the Sketches	4-2
Dimensioning of the True Length of an Arc	4-4
Measuring Distances and Viewing Section Properties	4-5
Measuring Distances	4-5
Determining Section Properties of Closed Sketches	4-8
Creating the Base Feature by Extruding the Sketches	4-9
Creating Solid Extruded Features	4-10
Creating Thin Extruded Features	4-14
Creating the Base Features by Revolving the Sketches	4-17
Creating Solid Revolve Features	4-18
Creating Thin Revolved Features	4-20
Dynamically Rotating the View of the Model	4-22
Rotating the View Freely in 3D Space	4-22
Rotating the View Around a Selected Vertex, Edge, or Face	4-22
Modifying the View Orientation	4-23
Restoring Previous View	4-25
Display Modes of the Model	4-25
Wireframe	4-25
Hidden Lines Visible	4-25
Hidden Lines Removed	4-25
Shaded With Edges	4-25
Shaded	4-25
Shadows In Shaded Mode	4-26
Perspective	4-26
Assigning Materials and Textures to the Models	4-26
Assigning Materials to a Model	4-27
Assigning Textures to a Model	4-28
Tutorial 1	4-29
Tutorial 2	4-35
Tutorial 3	4-40
Self-Evaluation Test	4-46
Review Questions	4-46
Exercise 1	4-47
Exercise 2	4-48

## Chapter 5: Creating Reference Geometries

Importance of the Sketching Planes	5-2
Reference Geometry	5-2
Creating New Planes	5-4
Creating Reference Axis	5-9
Creating Reference Points	5-13
Creating Reference Coordinate System	5-16

Other Boss/Base Options	Enhanced	5-16
From		5-16
End Condition		5-19
Direction of Extrusion		5-23
Modeling using the Contour Selection Method		5-23
Creating the Cut Features		5-27
Creating Extruded Cuts	Enhanced	5-27
Handling Multiple Bodies in Cut Feature		5-30
Creating Revolved Cuts		5-33
Concept of the Feature Scope		5-34
Tutorial 1		5-34
Tutorial 2		5-39
Tutorial 3		5-45
Self-Evaluation Test		5-52
Review Questions		5-53
Exercise 1		5-54
Exercise 2		5-55

## Chapter 6: Advanced Modeling Tools-I

Advanced Modeling Tools		6-2
Creating Simple Hole		6-2
Creating the Standard Holes Using the Hole Wizard		6-3
Creating Fillets	Enhanced	6-15
Selection Methods		6-31
Creating Chamfer		6-34
Creating the Shell Feature		6-37
Creating Wrap Feature		6-41
Tutorial 1		6-43
Tutorial 2		6-53
Self-Evaluation Test		6-61
Review Questions		6-61
Exercise 1		6-62
Exercise 2		6-63

## Chapter 7: Advanced Modeling Tools-II

Advanced Modeling Tools		7-2
Creating a Mirror Feature		7-2
Creating a Linear Pattern Feature		7-6
Creating a Circular Pattern Feature		7-11
Creating a Sketch-driven Pattern		7-14
Creating a Curve-driven Pattern		7-15
Creating a Table-driven Pattern		7-19
Creating the Rib Feature		7-20
Creating the Dome Feature		7-26
Creating the Shape Feature		7-28

Creating Indents	<b>New</b>	7-31
		7-35
Tutorial 1		7-37
Tutorial 2		7-45
Tutorial 3		7-50
Self-Evaluation Test		7-60
Review Questions		7-61
Exercise 1		7-62
Exercise 2		7-62
Exercise 3		7-65

## 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 Tool		8-5
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	<b>Enhanced</b>	8-15
Reordering the Features		8-18
Rolling Back the Model		8-19
Renaming the Features		8-21
Creating Folders in the FeatureManager Design Tree		8-21
What's Wrong? Functionality		8-21
Tutorial 1		8-23
Tutorial 2		8-30
Tutorial 3		8-34
Self-Evaluation Test		8-41
Review Questions		8-42
Exercise 1		8-42
Exercise 2		8-44

## Chapter 9: Advanced Modeling Tools-III

Advanced Modeling Tools		9-2
Creating the Sweep Feature	<b>Enhanced</b>	9-2

Creating Cut Sweep Features	Enhanced	9-14
Creating the Loft Feature	Enhanced	9-15
Creating 3D Sketches	Enhanced	9-26
Editing the 3D Sketches		9-28
Creating Curves		9-28
Extruding a 3D Sketch	Enhanced	9-41
Creating Draft Features		9-42
Creating the Deform Feature	Enhanced	9-44
Creating Flex Features	New	9-50
Tutorial 1		9-54
Tutorial 2		9-58
Tutorial 3		9-62
Self-Evaluation Test		9-67
Review Questions		9-68
Exercise 1		9-69
Exercise 2		9-70

## Chapter 10: Assembly Modeling-I

Assembly Modeling	10-2
Types of Assembly Design Approach	10-2
Creating Bottom-Up Assemblies	10-3
Placing Components in the Assembly Document	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	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	10-29
Rotating Individual Components Using the Rotate Component Tool	10-29
Moving and Rotating an Individual Components using the Triad	10-30
Tutorial 1	10-32
Tutorial 2	10-46
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	11-4
Applying Limit Mate	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 Components	11-9
Editing Subassemblies	11-10
Dissolving Subassemblies	11-11
Replacing Components	11-11
Creating Patterns of Components in an Assembly	11-14
Feature-driven Pattern	11-14
Local Pattern	11-15
Mirroring Components	11-16
Simplifying Assemblies using the Visibility Options	11-18
Hiding the Components	11-19
Suppressing Components	11-19
Changing the Transparency Conditions	11-20
Checking Interferences in an Assembly	11-21
Creating Assemblies for Mechanism	11-22
Analyzing Collisions using the Collision Detection Tool	11-23
Creating the Exploded State of an Assembly	Enhanced 11-25
Creating the Explode Line Sketch	11-27
Tutorial 1	11-29
Tutorial 2	11-41
Self-Evaluation Test	11-45
Review Questions	11-46
Exercise 1	11-47

## 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	Enhanced 12-15
Generating Section Views	Enhanced 12-16
Generating Aligned Section Views	Enhanced 12-21
Generating Broken-out Section Views	12-23



Generating Auxiliary Views		12-25
Generating Detail Views	Enhanced	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 an Assembly		12-34
Working with Interactive Drafting in SolidWorks		12-35
Editing and Modifying the Drawing Views		12-35
Changing the View Orientation		12-35
Changing the Scale of Drawing Views		12-36
Deleting Drawing Views		12-36
Rotating Drawing Views		12-36
Modifying Hatch Pattern in Section Views	Enhanced	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 Annotations using the Model Items Tool	Enhanced	13-2
Adding Reference Annotations		13-5
Editing Annotations		13-20
Adding the Bill of Material (BOM) in Drawing	Enhanced	13-21
Adding Balloons to Drawing Views		13-23
Adding Balloons using the Balloon Tool		13-23
Adding Balloons using the AutoBalloon Tool		13-24
Adding New Sheets to Drawing Views		13-25
Editing the Sheet Format		13-26
Creating a User-Defined Sheet Format		13-27
Tutorial 1		13-27
Tutorial 2		13-33
Self-Evaluation Test		13-38
Review Questions		13-39
Exercise 1		13-40

### 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 2005, dreamtech PRESS, India
- SolidWorks for Designers Release 2004, CADCIM Technologies, USA
- SolidWorks for Designers Release 2003, CADCIM Technologies, USA
- SolidWorks for Engineers and Designers Release 2003, dreamtech PRESS, India
- CATIA for Designers V5R13, CADCIM Technologies, USA
- CATIA for Engineers and Designers V5R13, dreamtech PRESS, India
- Pro/ENGINEER Wildfire for Designers Release 2.0, CADCIM Technologies, USA
- Pro/ENGINEER Wildfire for Engineers & Designers Release 2.0, 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
- AutoCAD 2000 Tecniche Avanzate, 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.