

Solid Edge ST7 for Designers (12th Edition)

CADCIM Technologies

*525 St. Andrews Drive
Schererville, IN 46375, USA
(www.cadcim.com)*

Contributing Author

Sham Tickoo

*Professor
Department of Mechanical Engineering Technology
Purdue University Calumet
Hammond, Indiana, USA*





CADCIM Technologies

Solid Edge ST7 for Designers Sham Tickoo

CADCIM Technologies
525 St Andrews Drive
Schererville, Indiana 46375, USA
www.cadcim.com

Copyright ©2015 by CADCIM Technologies, USA. All rights reserved. Printed in the United States of America except as permitted under the United States Copyright Act of 1976.

No part of this publication may be reproduced or distributed in any form or by any means, or stored in the database or retrieval system without the prior permission of CADCIM Technologies.

ISBN 978-1-936646-88-3

NOTICE TO THE READER

Publisher does not warrant or guarantee any of the products described in the text or perform any independent analysis in connection with any of the product information contained in the text. Publisher does not assume, and expressly disclaims, any obligation to obtain and include information other than that provided to it by the manufacturer.

The reader is expressly warned to consider and adopt all safety precautions that might be indicated by the activities herein and to avoid all potential hazards. By following the instructions contained herein, the reader willingly assumes all risks in connection with such instructions.

The Publisher makes no representation or warranties of any kind, including but not limited to, the warranties of fitness for particular purpose or merchantability, nor are any such representations implied with respect to the material set forth herein, and the publisher takes no responsibility with respect to such material. The publisher shall not be liable for any special, consequential, or exemplary damages resulting, in whole or part, from the reader's use of, or reliance upon, this material.

www.cadcim.com

DEDICATION

*To teachers, who make it possible to disseminate knowledge
to enlighten the young and curious minds
of our future generations*

*To students, who are dedicated to learning new technologies
and making the world a better place to live in*

THANKS

*To the faculty and students of the MET department of
Purdue University Calumet for their cooperation*

To employees of CADCIM Technologies for their valuable help

Online Training Program Offered by CADCIM Technologies

CADCIM Technologies provides effective and affordable virtual online training on various software packages including Computer Aided Design and Manufacturing (CAD/CAM), computer programming languages, animation, architecture, and GIS. The training is delivered 'live' via Internet at any time, any place, and at any pace to individuals as well as the students of colleges, universities, and CAD/CAM training centers. The main features of this program are:

Training for Students and Companies in a Classroom Setting

Highly experienced instructors and qualified engineers at CADCIM Technologies conduct the classes under the guidance of Prof. Sham Tickoo of Purdue University Calumet, USA. This team has authored several textbooks that are rated "one of the best" in their categories and are used in various colleges, universities, and training centers in North America, Europe, and in other parts of the world.

Training for Individuals

CADCIM Technologies with its cost effective and time saving initiative strives to deliver the training in the comfort of your home or work place, thereby relieving you from the hassles of traveling to training centers.

Training Offered on Software Packages

CADCIM provides basic and advanced training on the following software packages:

CAD/CAM/CAE: CATIA, Pro/ENGINEER Wildfire, Creo Parametric, Creo Direct, SolidWorks, Autodesk Inventor, Solid Edge, NX, AutoCAD, AutoCAD LT, AutoCAD Plant 3D, Customizing AutoCAD, EdgeCAM, and ANSYS

Architecture and GIS: Autodesk Revit Architecture, AutoCAD Civil 3D, Autodesk Revit Structure, AutoCAD Map 3D, Revit MEP, Navisworks, Primavera Project Planner, and Bentley STAAD Pro

Animation and Styling: Autodesk 3ds Max, Autodesk 3ds Max Design, Autodesk Maya, Autodesk Alias, Foundry NukeX, and MAXON CINEMA 4D

Computer Programming: C++, VB.NET, Oracle, AJAX, and Java

For more information, please visit the following link: <http://www.cadcim.com>

Note

If you are a faculty member, you can register by clicking on the following link to access the teaching resources: <http://www.cadcim.com/Registration.aspx>. The student resources are available at <http://www.cadcim.com>. We also provide **Live Virtual Online Training** on various software packages. For more information, write us at sales@cadcim.com.

Table of Contents

Dedication	iii
Preface	xvii

Chapter 1: Introduction to Solid Edge ST7

Introduction to Solid Edge ST7	1-2
Solid Edge Environments	1-6
System Requirements for Installing Solid Edge ST7	1-8
Important Terms and Definitions	1-8
Getting Started With Solid Edge ST7	1-12
User Interface of Solid Edge	1-13
Prompt Line	1-13
PathFinder	1-13
Docking Window	1-13
Application Button	1-14
Quick Access Toolbar	1-14
Ribbon	Enhanced 1-14
Status Bar	1-15
Record	1-15
Upload to YouTube	1-16
Command Bar	1-16
QuickPick	1-17
Part Environment Tabs	1-17
Assembly Environment Tabs	1-18
Draft Environment Tabs	1-18
Radial Menu	1-18
Simulation Express	1-19
Using Intellisketch	1-19
Units for Dimensions	1-20
Automatic Saving Option	1-20
Color Scheme in Solid Edge	1-20
Self-Evaluation Test	1-22

Chapter 2: Drawing Sketches for Solid Models

Sketching in the Part Environment	2-2
Starting a New Document in the Part Environment	2-2
Starting a New Part file by Using the New Dialog Box	Enhanced 2-3
Transition between Part Environments	2-6
Starting a Sketch in the Part Environment	2-6
Sketching Tools	2-7

Drawing Lines	2-7
Drawing Tangent and Normal Arcs	2-9
Placing Sketched Points	2-10
FreeSketch	2-10
Drawing Circles	2-10
Drawing Ellipses	2-12
Drawing Arcs	2-13
Drawing Rectangles	2-14
Drawing Polygons	2-16
Drawing Curves	2-16
Clean Sketch	2-17
Converting Sketched Entities into Curves	2-19
Filleting Sketched Entities	2-19
Chamfering Sketched Entities	2-20
The Drawing Display Tools	2-21
Zooming an Area	2-21
Zooming Dynamically	2-21
Fitting all Entities into the Current Display	2-21
Panning the Drawings	2-21
Restoring the Original Orientation of the Sketching Plane	2-22
Selecting Sketched Entities	2-22
Deleting Sketched Entities	2-22
Grid	2-23
Reposition Origin	2-24
Tutorial 1	2-25
Tutorial 2	2-29
Tutorial 3	2-33
Self-Evaluation Test	2-37
Review Questions	2-37
Exercise 1	2-38
Exercise 2	2-39
Exercise 3	2-39

Chapter 3: Adding Relationships and Dimensions to Sketches

Geometric Relationships	3-2
Connect Relationship	3-2
Concentric Relationship	3-3
Horizontal/Vertical Relationship	3-3
Collinear Relationship	3-4
Parallel Relationship	3-4
Perpendicular Relationship	3-4
Lock Relationship	3-4
Rigid Set Relationship	3-5
Tangent Relationship	3-5
Equal Relationship	3-5
Symmetric Relationship	3-6
Setting the Symmetry Axis (Ordered Environment)	3-6

Controlling the Display of Relationship Handles	3-7
Conflicts in Relationships	3-7
Deleting Relationships	3-7
Dimensioning the Sketched Entities	3-8
Adding Linear Dimensions	3-8
Adding Aligned Dimensions	3-12
Adding Angular Dimensions	3-13
Adding Diameter Dimensions	3-14
Adding Radial Dimensions	3-15
Adding Symmetric Diameter Dimensions	3-15
Adding Coordinate Dimensions	3-16
Change Coordinate Origin	3-17
Adding Angular Coordinate Dimensions	3-17
Adding Automatic Dimensions (Ordered Environment)	3-18
Understanding the Concept of Fully Constrained Sketches	3-19
Measuring Sketched Entities	3-19
Measuring Distances	3-19
Measuring the Total Length of a Closed Loop or an Open Sketch	3-20
Measuring an Area	3-20
Calculating the Area Properties	3-20
Tutorial 1	3-21
Tutorial 2	3-26
Tutorial 3	3-30
Self-Evaluation Test	3-34
Review Questions	3-35
Exercise 1	3-36
Exercise 2	3-36
Exercise 3	3-37
Exercise 4	3-37
Exercise 5	3-38

Chapter 4: Editing, Extruding, and Revolving the Sketches

Editing the Sketches	4-2
Trimming the Sketched Entities	4-2
Creating Splits in the Sketched Entities	4-3
Extending the Sketched Entities	4-2
Trimming/Extending Entities to a Corner	4-3
Creating Offset Copies	4-4
Creating Symmetric Offset Copies	4-4
Moving/Copying the Sketched Entities	4-6
Rotating the Sketched Entities	4-7
Mirroring the Sketched Entities	4-8
Scaling the Sketched Entities	4-9
Stretching the Sketched Entities	4-10
Editing the Sketched Entities by Dragging	4-10
Writing Text in the Sketch	4-11
Inserting Images into Sketches	4-12

Converting Sketches into Base Features	4-14
Creating Base Features in Synchronous Part Environment	4-15
Creating an Extruded Feature	4-17
Creating Revolved Features	4-17
Creating Primitive Features (Synchronous Environment)	4-18
Creating a Box Feature	4-19
Creating a Cylinder Feature	4-19
Creating a Sphere Feature	4-19
Creating Ordered Features	4-19
Creating Extruded Features	4-20
Creating Revolved Features	4-20
Rotating the View of a Model in 3D Space	4-21
Restoring Standard Views	4-21
Setting the Display Modes	4-22
Shaded with Visible Edges	4-22
Shaded	4-22
Visible and Hidden Edges	4-23
Visible Edges	4-23
Wire Frame	4-23
Drop Shadow	4-23
Improving the Display Quality of a Model	4-23
Tutorial 1	4-23
Tutorial 2	4-27
Tutorial 3	4-29
Self-Evaluation Test	4-31
Review Questions	4-32
Exercise 1	4-33
Exercise 2	4-33
Exercise 3	4-34
Exercise 4	4-34

Chapter 5: Working with Additional Reference Geometries

Additional Sketching and Reference Planes	5-2
Local Reference Planes	5-3
Global Reference Planes	5-3
Creating Reference Planes (Synchronous)	5-3
Creating a Coincident Plane	5-3
Modifying Planes Using the Steering Wheel	5-4
Creating a Plane Normal to an Edge or a Sketched Curve	5-5
Creating a Plane Using Three Points	5-6
Creating a Tangent Plane	5-6
Creating Reference Planes (Ordered)	5-7
Creating a Parallel Plane (Ordered Environment)	5-7
Creating an Angled Plane	5-8
Creating a Perpendicular Plane (Ordered Environment)	5-9
Creating a Plane Using Coincident by Axis (Ordered)	5-10
Displaying the Reference Axes (Ordered Environment)	5-10

Understanding Coordinate Systems	5-10
Creating a Coordinate System	5-11
Using the Other Options of the Extrude Tool	5-13
Creating Cutout Features	5-17
Creating Extruded Cutouts (Ordered Environment)	5-17
Creating Revolved Cutouts (Ordered Environment)	5-19
Creating Cutouts (Synchronous Environment)	5-20
Using the Edges of Existing Features (Synchronous Environment)	5-21
Projecting Edges (Ordered Environment)	5-23
Advanced Drawing Display Tools	5-23
Creating User-defined Named Views	5-23
Using Common Views	5-24
Tutorial 1	5-24
Tutorial 2	5-29
Tutorial 3	5-34
Self-Evaluation Test	5-38
Review Questions	5-39
Exercise 1	5-40
Exercise 2	5-41
Exercise 3	5-42

Chapter 6: Advanced Modeling Tools-I

Advanced Modeling Tools	6-2
Creating Holes (Ordered Environment)	6-2
Creating Holes (Synchronous)	6-10
Creating Rounds (Ordered Environment)	6-12
Creating the Constant Radius Round	6-13
Creating the Variable Radius Round	6-17
Creating Rounds (Synchronous Environment)	6-18
Creating Variable Radius Rounds (Synchronous Environment)	6-18
Creating Chamfers (Ordered Environment)	6-19
Creating Rectangular and Circular Patterns (Ordered Environment)	6-21
Creating Rectangular Patterns	6-21
Creating Circular Patterns	6-26
Creating a Pattern along a Curve (Ordered Environment)	6-28
Creating Patterns (Synchronous Environment)	6-31
Creating Rectangular Patterns	6-31
Creating Circular Patterns	6-33
Creating a Pattern along a Curve (Synchronous Environment)	6-34
Creating Fill Pattern	6-35
Recognizing Hole Patterns	6-38
Mirroring Features and Bodies (Ordered Environment)	6-39
Mirroring Selected Features	6-40
Mirroring Bodies	6-41
Tutorial 1	6-41
Tutorial 2	6-46

Tutorial 3	6-50
Self-Evaluation Test	6-56
Review Questions	6-57
Exercise 1	6-58
Exercise 2	6-59
Exercise 3	6-60
Exercise 4	6-61

Chapter 7: Editing Features

Editing Models in the Synchronous Environment	7-2
Adding Dimensions to the Model	7-2
Dimensioning a Feature	7-2
Dimensioning Holes	7-2
Editing the Round Feature (Synchronous)	7-3
Adding Relations	7-4
Aligning Faces	7-4
Live Rules	7-6
Applying Concentric Relationship	7-7
Applying Tangent Relationship	7-8
Applying Symmetric Relationship	7-8
Applying Parallel Relationship	7-9
Applying Perpendicular Relationship	7-10
Applying Offset Relationship	7-10
Applying Equal Radius Relationship	7-10
Applying Horizontal/Vertical Relationship	7-11
Applying Coplanar Axis Relationship	7-12
Applying Rigid Relationship	7-13
Applying Ground Relationship	7-13
Other Selection Handles	7-13
Reference Plane Handle	7-13
Selection Manager	7-14
Modifying Faces Using the Steering Wheel	7-14
Modifying the Model by Editing Dimensions	7-16
Creating Live Sections	7-16
Modifying the Model by Detaching and Attaching Faces or Features	7-17
Modifying the Model by Isolating Features	7-18
Editing Features in the Ordered Environment	7-19
Suppressing Features	7-20
Unsuppressing the Suppressed Features	7-21
Deleting Features	7-21
Copying and Pasting Features	7-21
Rolling Back a Model to a Feature	7-22
Converting Ordered Features to Synchronous	7-22
Assigning Color to a Part, Feature, or Face	7-23
Playing Back the Construction of Features	7-24
Modifying Faces in the Ordered Environment	7-24

Moving faces	7-24
Rotating faces	7-24
Offsetting faces	7-25
Copying Sketch Objects to Another Sketch	7-25
Checking the Physical Properties of a Model	7-26
Modifying the Display of Construction Entities	7-27
Tutorial 1	7-27
Tutorial 2	7-33
Tutorial 3	7-38
Tutorial 4	7-42
Self-Evaluation Test	7-46
Review Questions	7-47
Exercise 1	7-48
Exercise 2	7-48
Exercise 3	7-49

Chapter 8: Advanced Modeling Tools-II

Advanced Modeling Tools	8-2
Creating Threads <i>Enhanced</i>	8-2
Creating Slots	8-4
Adding Drafts to the Model	8-7
Adding Ribs to the Model (Ordered)	8-9
Adding Thin Wall Features (Ordered)	8-12
Adding a Thin Wall to a Particular Region	8-14
Adding a Lip to the Model	8-16
Creating Web Networks (Ordered)	8-18
Creating Vents	8-19
Creating Mounting Bosses (Ordered)	8-23
Reordering Features	8-27
Tutorial 1	8-28
Tutorial 2	8-31
Tutorial 3	8-35
Tutorial 4	8-39
Self-Evaluation Test	8-42
Review Questions	8-43
Exercise 1	8-44
Exercise 2	8-45
Exercise 3	8-46

Chapter 9: Advanced Modeling Tools-III

Advanced Modeling Tools	9-2
Creating Swept Protrusions	9-2
Creating Swept Cutouts	9-9
Creating Lofted Protrusions	9-9
Creating Lofted Cutouts	9-14

Creating 3D Sketch (Synchronous Environment)	New	9-15
Creating Helical Protrusions		9-18
Creating Helical Cutouts		9-22
Creating Normal Protrusions		9-23
Creating Normal Cutouts		9-23
Working with Additional Bodies		9-24
Inserting a New Body		9-24
Applying Boolean Operations to Bodies		9-25
Tutorial 1		9-29
Tutorial 2		9-37
Tutorial 3		9-45
Self-Evaluation Test		9-51
Review Questions		9-52
Exercise 1		9-53
Exercise 2		9-54
Exercise 3		9-55

Chapter 10: Assembly Modeling-I

The Assembly Environment	10-2
Working with the Assembly Environment	10-2
Types of Assembly Design Approaches	10-2
Creating the Bottom-Up Assembly	10-3
Assembling the First Component	10-3
Assembling the Second Component	10-4
Applying Assembly Relationships	10-5
Creating the Top-Down Assembly	10-16
Creating a Component in the Top-Down Assembly	10-16
Creating the Pattern of Components in an Assembly	10-18
Creating a Reference Pattern	10-18
Mirroring a Component in an Assembly	10-19
Creating Material Removal Features in an Assembly	10-20
Assembly Features	10-20
Assembly-driven Part Features	10-20
Create Part Feature	10-20
Tools for Material Removal in an Assembly	10-21
Subtracting material from a Part Using Another Part	10-22
Moving the Individual Components	10-22
Moving Multiple Components	10-23
Tutorial 1	10-24
Tutorial 2	10-38
Self-Evaluation Test	10-46
Review Questions	10-47
Exercise 1	10-48
Exercise 2	10-51

Chapter 11: Assembly Modeling-II

Creating Subassemblies	11-2
Editing the Assembly Relationships	11-3
Modifying the Values	11-3
Applying Additional Relationships	11-3
Modifying Assembly Relationships	11-5
Editing the Assembly Components	11-6
Modifying Synchronous Components in the Assembly Environment	11-6
Dispersing Subassemblies	11-7
Replacing Components	11-7
Replace Part	11-7
Replace Part with Standard Part	11-8
Replace Part with New Part	11-8
Replace Part with Copy	11-9
Simplifying Assemblies Using Visibility Options	11-10
Hiding and Displaying the Components	11-10
Changing Transparency Conditions	11-10
Interference Detection in Assemblies	11-10
Interference Options	11-10
Checking for the Interference	11-13
Creating Fastener System	11-13
Creating the Exploded State of Assemblies	11-16
Automatic Explode	11-17
Unexploding Assemblies	11-18
Exploding Assemblies Manually	11-19
Changing the Distance between the Components	11-20
Repositioning the Parts	11-21
Removing the Parts	11-21
Drop	11-21
Modify	11-21
Draw	11-22
Tutorial 1	11-22
Tutorial 2	11-29
Tutorial 3	11-37
Self-Evaluation Test	11-40
Review Questions	11-40
Exercise 1	11-41
Exercise 2	11-45

Chapter 12: Generating, Editing, and Dimensioning the Drawing Views

The Draft Environment	12-2
Types of Views Generated in Solid Edge	12-3
Generating Drawing Views	12-4
Generating the Base View	12-4

Generating the Principal View	12-8
Generating the Auxiliary View	12-10
Generating the Section View	12-11
Generating the Revolved Section Views	12-13
Generating the Broken-Out Section View	12-16
Generating the Detail View	12-17
Generating the Broken View	12-18
Inheriting Break Lines to a Principal, Section or Auxiliary View	12-20
Working with Interactive Drafting	12-20
Manipulating Drawing Views	12-20
Aligning Drawing Views	12-20
Modifying the Scale of Drawing Views	12-21
Cropping the Drawing Views	12-21
Moving the Drawing Views	12-22
Rotating the Drawing Views	12-22
Applying the Hatch Pattern	12-22
Modifying the Properties of Drawing Views	12-22
Adding Annotations to Drawing Views	12-24
Displaying Center marks and Center lines in a Drawing View	12-24
Creating a Bolt Hole Circle	12-25
Adding Callouts to Drawing View	12-28
Adding Surface Texture Symbols to a Drawing View	12-29
Specifying the Edge Conditions	12-30
Adding a Feature Control Frame to Drawing Views	12-31
Adding Datum Target to Drawing View	12-32
Adding a Datum Frame to Drawing View	12-33
Adding New Drawing Sheets	12-34
Editing the Default Sheet Format	12-34
Evolving a 3D Model from a 2D Drawing	12-34
Generating the Exploded Views of Assemblies	12-35
Creating Associative Balloons and Parts List	12-37
Parts List Properties Dialog Box	12-38
Setting the Text Properties	12-41
Tutorial 1	12-43
Tutorial 2	12-48
Tutorial 3	12-53
Tutorial 4	12-60
Self-Evaluation Test	12-63
Review Questions	12-64
Exercise 1	12-65
Exercise 2	12-65

Chapter 13: Surface Modeling

Surface Modeling	13-2
Creating Surfaces in Solid Edge	13-2
Creating an Extruded Surface	13-2

Creating a Revolved Surface	13-3
Creating a Swept Surface	13-4
Creating Surfaces Using the BlueSurf Tool (Synchronous and Ordered)	13-6
Inspecting Surface Curvature at a Section	13-14
Creating Surfaces Using the Bounded Tool	13-14
Stitching Multiple Surfaces to Create a Single Surface	13-15
Creating a Surface replacing the another surfaces	13-16
Creating Offset Surfaces	13-17
Copying a Surface	13-17
Creating a Ruled Surface	13-19
Creating a BlueDot (Ordered Environment)	13-20
Creating a Curve at the Intersection of Two Surfaces	13-21
Trimming the Surfaces	13-22
Extending the Surfaces	13-23
Replacing the Faces of a Part with a Surface	13-24
Splitting Faces	13-25
Using the Intersect Tool	13-26
Creating Curves in 3D by Selecting Keypoints	13-26
Creating Curves by Table	13-28
Projecting the Curves on Surfaces	13-29
Creating a Curve at the Projection of Two Curves	13-30
Drawing a Curve on a Surface	13-31
Deriving Curves	13-32
Splitting a Curve	13-32
Splitting a Body	13-32
Adding Thickness to a Surface	13-34
Creating Rounds Using Blending	13-34
Adding a Draft	13-39
Using the Parting Split Tool	13-42
Using the Parting Surface Tool	13-43
Using the Offset Edge Tool	13-43
Tutorial 1	13-44
Tutorial 2	13-50
Self-Evaluation Test	13-57
Review Questions	13-57
Exercise 1	13-58
Exercise 2	13-59
Exercise 3	13-59

Chapter 14: Sheet Metal Design

The Sheet Metal Module	14-2
Setting the Sheet Metal Part Properties	14-3
Creating the Base of the Sheet Metal Parts	14-7
Adding Flanges to a Sheet Metal Part	14-8
Adding Flanges in Synchronous Sheet Metal	14-12
Creating Contour Flanges	14-13

Enhanced

Adding Contour Flanges in Synchronous Sheet Metal	14-17
Creating Lofted Flanges	14-18
Adding the Jog to the Sheet	14-19
Bending the Sheet Metal Part	14-20
Unbending the Sheet Metal Part	14-21
Rebending the Sheet Metal Part	14-21
Filleting or Chamfering Corners of a Sheet Metal Part	14-22
Closing the 2 Bend Corners of a Sheet Metal Part	14-22
Creating Dimples in a Sheet Metal Part	14-25
Creating Louvers in a Sheet Metal Part	14-28
Creating Drawn Cutouts in a Sheet Metal Part	14-30
Creating Beads in a Sheet Metal Part	14-31
Embossing Solids onto a Sheet Metal	14-34
Adding Gussets to a Sheet Metal Part	14-35
Adding Hems	14-38
Converting a Solid Part into a Sheet Metal Part	Enhanced 14-41
Ripping the Corners of a Solid Part	14-43
Adding Sheet Metal Features to a Solid Part	14-43
Creating the Flat Pattern of a Sheet Metal Part	14-44
Creating Flat Patterns in the Flat Pattern Environment	14-44
Saving a Sheet Metal Part in the Flat Pattern Format	14-45
Tutorial 1	14-46
Tutorial 2	14-54
Self-Evaluation Test	14-59
Review Questions	14-59
Exercise 1	14-60
Exercise 2	14-62

Chapter 15: Student Projects

Project 1	15-2
Project 2	15-16
Exercise 1	15-43

Index	I-1
--------------	------------

Preface

Solid Edge ST7

Solid Edge, a product of Siemens, is one of the world's fastest growing solid modeling software. Solid Edge with Synchronous Technology combines the speed and flexibility of direct modeling with precise control of dimension-driven design through precision sketching, region selection, face selection, and handle selection. Solid Edge ST7 integrates the synchronous modeling with the traditional modeling into a single environment. It is an integrated solid modeling tool, which not only unites the synchronous modeling with traditional modeling but also addresses every design-through-manufacturing process. Solid Edge ST7 allows the users to convert any selected ordered feature in existing models into a synchronous feature. This solid modeling package allows the manufacturing companies to get an insight into the design intent, thereby promoting collaboration and allowing the companies have an edge over their competitors. This package is remarkably user-friendly and helps the users to be productive from day one.

In Solid Edge, the 2D drawing views can be easily generated in the drafting environment after creating solid models and assemblies. The drawing views that can be generated include orthographic views, isometric views, auxiliary views, section views, detail views, and so on. You can use any predefined drawing standard file for generating the drawing views. You can display model dimensions in the drawing views or add reference dimensions whenever you want. The bidirectional associative nature of this software ensures that any modification made in the model is automatically reflected in the drawing views.

The **Solid Edge ST7 for Designers** textbook has been written to help the readers use Solid Edge ST7 effectively. This textbook covers both Synchronous and Ordered environments of Solid Edge ST7 such as Part, Assembly, Sheet Metal, and Drafting. A number of mechanical engineering industry examples are used as tutorials in this textbook so that the users can relate the knowledge gained with the actual mechanical industry designs. Some of the other salient features of this textbook are as follows:

- **Tutorial Approach**

The author has adopted the tutorial point-of-view and the learn-by-doing approach throughout the textbook. This approach guides the users through the process of creating the models in the tutorials.

- **Heavily Illustrated Text**

The text in this book is heavily illustrated with about 1100 line diagrams and screen capture images.

- **Real-World Projects as Tutorials**

The author has used about 50 real world mechanical engineering projects as tutorials in this textbook. This enables the readers to relate the tutorials to the real-world models in the mechanical engineering industry. In addition, there are about 44 exercises that are also based on the real-world mechanical engineering projects.

- **Tips and Notes**

Additional information related to various topics is provided to the users in the form of tips and notes.

- **Learning Objectives**

The first page of every chapter summarizes the topics that are covered in the chapter.

- **Self-Evaluation Test, Review Questions, and Exercises**

Every chapter ends with Self-Evaluation Test so that the users can assess their knowledge of the chapter. The answers to Self-Evaluation Test are given at the end of the chapter. Also, the Review Questions and Exercises are given at the end of each chapter and they can be used by the Instructors as test questions and exercises.

Symbols Used in the Textbook

**Note**

The author has provided additional information in the form of notes.

**Tip**

The author has provided useful information to the users about the topic being discussed in the form of tips.

**New**

This icon indicates that the command or tool being discussed is new.

**Enhanced**

This icon indicates that the command or tool being discussed is enhanced.

Formatting Conventions Used in the Textbook

Please refer to the following list for the formatting conventions used in this textbook.

- Names of tools, buttons, options, panels, tabs, and ribbon are written in boldface. Example: The **Extrude** tool, the **Finish Sketch** button, the **Modify** panel, the **Sketch** tab, and so on.
- Names of dialog boxes, drop-downs, drop-down lists, list boxes, areas, edit boxes, check boxes, and radio buttons are written in boldface. Example: The **Revolve** dialog box, the **Create 2D Sketch** drop-down in the **Sketch** panel of the **Model** tab, the **Placement** drop-down of **Hole** dialog box, the **Distance** edit box in the **Extrude** dialog box, the **Extended Profile** check box in the **Rib** dialog box, the **Drilled** radio button in the **Hole** dialog box, and so on.
- Values entered in edit boxes are written in boldface. Example: Enter **5** in the **Radius** edit box.
- Names and paths of the files are written in italics. Example: *C:\Solid Edge\c03, c03tut03.prt*, and so on

The methods of invoking a tool/option from the **Ribbon**, **Quick Access** toolbar, and **Application Menu** are given in a shaded box.

Ribbon: Get Started > Launch > New
Quick Access Toolbar: New
Application Menu: New

Naming Conventions Used in the Text Tool

If you click on an item in a toolbar or a panel of the **Ribbon** and a command is invoked to create/edit an object or perform some action, then that item is termed as **tool**.

For example:

To Create: **Line** tool, **Dimension** tool, **Extrude** tool

To Edit: **Fillet** tool, **Draft** tool, **Trim Surface** tool

Action: **Zoom All** tool, **Pan** tool, **Copy Object** tool

If you click on an item in a toolbar or a panel of the **Ribbon** and a dialog box is invoked wherein you can set the properties to create/edit an object, then that item is also termed as **tool**, refer to Figure 1.

For example:

To Create: **Create iPart** tool, **Parameters** tool, **Create** tool

To Edit: **Styles Editor** tool, **Document Settings** tool

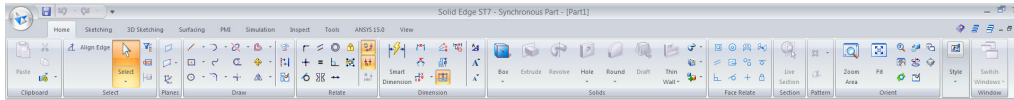


Figure 1 The Home tab in the Part environment

Button

The item in a dialog box that has a 3d shape like a button is termed as **Button**. For example, **OK** button, **Cancel** button, **Apply** button, and so on.

Dialog Box

The naming conventions used for the components in a dialog box are mentioned in Figure 2.

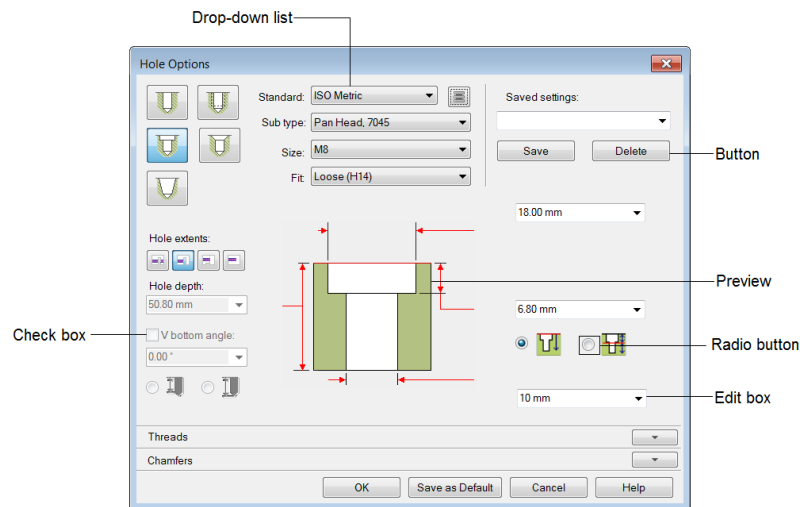


Figure 2 The components in a dialog box

Drop-down

A drop-down is one in which a set of common tools are grouped together. You can identify a drop-down with a down arrow on it. These drop-downs are given a name based on the tools grouped in them. For example, **Rectangle** drop-down, **Line** drop-down, **More planes** drop-down, and so on; refer to Figure 3.



Figure 3 The Rectangle, Line, and More planes drop-downs

Drop-down List

A drop-down list is the one in which a set of options are grouped together. You can set parameters by using these options. You can identify a drop-down list with a down arrow on it. For example, the **Selection Type** drop-down list, the **Select from** drop-down list, and so on, refer to Figure 4.

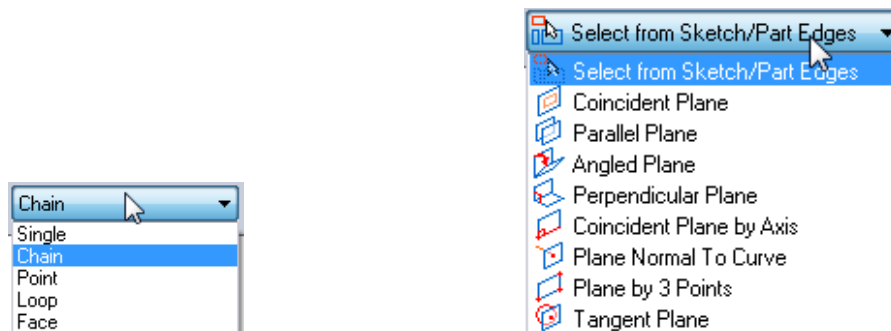


Figure 4 The Selection Type and Select from drop-down lists

Free Companion Website

It has been our constant endeavor to provide you the best textbooks and services at affordable price. In this endeavor, we have come out with a Free Companion website that will facilitate the process of teaching and learning of Solid Edge ST7. If you purchase this textbook, you will get access to the files on the Companion website.

The following resources are available for the faculty and students in this website:

Faculty Resources

- **Technical Support**

You can get online technical support by contacting techsupport@cadcim.com.

- **Instructor Guide**

Solutions to all review questions and exercises in the textbook are provided in this guide to help the faculty members test the skills of the students.

- **PowerPoint Presentations**

The contents of the book are arranged in PowerPoint slides that can be used by the faculty for their lectures.

- **Part Files**

The part files used in illustrations, tutorials, and exercises are available for free download.

Student Resources

- **Technical Support**

You can get online technical support by contacting *techsupport@cadcim.com*.

- **Part Files**

The part files used in illustrations and tutorials are available for free download.

- **Additional Students Projects**

Various projects are provided for the students to practice.

If you face any problem in accessing these files, please contact the publisher at *sales@cadcim.com* or the author at *stickoo@purduecal.edu* or *tickoo525@gmail.com*.

Stay Connected

You can now stay connected with us through Facebook and Twitter to get the latest information about our textbooks, videos, and teaching/learning resources. To stay informed of such updates, follow us on Facebook (*www.facebook.com/cadcim*) and Twitter (@cadcimtech). You can also subscribe to our YouTube channel (*www.youtube.com/cadcimtech*) to get the information about our latest video tutorials.