

Table of Contents

Dedication	iii
Preface	xvii

Chapter 1: Introduction

Introduction to Autodesk Inventor 2012	1-2
Part Module	1-2
Assembly Module	1-2
Presentation Module	1-3
Drawing Module	1-3
Sheet Metal Module	1-3
Getting Started with Autodesk Inventor	1-3
Quick Access Toolbar	1-7
Ribbon and Tabs	1-7
Sketch Tab	1-8
Model Tab	1-8
Sheet Metal Tab	1-9
Assemble Tab	1-9
Place Views Tab	1-9
Presentation Tab	1-10
Tools Tab	1-10
View Tab	1-10
Navigation Bar	1-11
Browser Bar	1-11
Units for Dimensions	1-12
Important Terms and their Definitions	1-12
Feature-based Modeling	1-12
Parametric Modeling	1-12
Bidirectional Associativity	1-12
Adaptive	1-12
Design Doctor	1-13
Constraints	1-13
Consumed Sketch	1-15
Select Other Behavior	1-16
Hotkeys	1-16
Part Module	1-16
Assembly Module	1-17
Drawing Module	1-18
Customizing Hotkeys	1-18
Creating the Sketch	1-20
Marking Menu	1-20
Color Scheme	1-22
Self-Evaluation Test	1-23
Review Questions	1-24

Enhanced

New

Chapter 2: Drawing Sketches for Solid Models

The Sketching Environment	2-2
The Open Dialog Box	2-2
Import DWG	2-5
Starting a New File	2-5
Introduction to the Sketching Environment	2-7
Setting Up the Sketching Environment	2-7
Understanding the Drawing Display Tools	2-9
Zoom All	2-10
Zoom	2-10
Zoom Window	2-10
Zoom Selected	2-11
Pan	2-11
Orbit	2-11
Constrained Orbit	2-11
Sketching Entities	2-12
Positioning Entities by Using Dynamic Input	2-13
Drawing Lines	2-14
Drawing Circles	2-17
Drawing Ellipses	2-18
Drawing Arcs	2-19
Drawing Rectangles	2-21
Drawing Polygons	2-22
Placing Points	2-23
Creating Fillets	2-23
Creating Chamfers	2-24
Drawing Splines	2-26
Deleting Sketched Entities	2-27
Finishing a Sketch	2-28
Tutorial 1	2-28
Tutorial 2	2-34
Tutorial 3	2-37
Tutorial 4	2-41
Self-Evaluation Test	2-45
Review Questions	2-45
Exercise 1	2-46
Exercise 2	2-47
Exercise 3	2-48

Chapter 3: Adding Constraints and Dimensions to Sketches

Adding Geometric Constraints to a Sketch	3-2
Perpendicular Constraint	3-2
Parallel Constraint	3-2
Tangent Constraint	3-3
Coincident Constraint	3-3
Concentric Constraint	3-3

Collinear Constraint	3-4
Horizontal Constraint	3-4
Vertical Constraint	3-4
Equal Constraint	3-5
Fix Constraint	3-5
Symmetric Constraint	3-5
Smooth Constraint	3-6
Viewing the Constraints Applied to a Sketched entity	3-6
Controlling Constraints and applying them automatically while sketching	3-7
Selecting Constraints	3-7
Applying Constraints	3-8
Deleting Geometric Constraints	3-8
Adding Dimensions to Sketches	3-8
Linear Dimensioning	3-9
Aligned Dimensioning	3-11
Angular Dimensioning	3-12
Diameter Dimensioning	3-13
Radius Dimensioning	3-14
Linear Diameter Dimensioning	3-14
Creating Driven Dimensions	3-16
Understanding the Concept of Fully-Constrained Sketches	3-17
Measuring Sketched Entities	3-17
Measuring Distances	3-17
Measuring Angles	3-19
Measuring Loops	3-20
Adding Linear Measurements	3-21
Clearing Accumulated Dimensions	3-21
Evaluating Region Properties	3-21
Tutorial 1	3-22
Tutorial 2	3-27
Tutorial 3	3-32
Tutorial 4	3-37
Self-Evaluation Test	3-41
Review Questions	3-41
Exercise 1	3-42
Exercise 2	3-43
Exercise 3	3-43
Exercise 4	3-44
Exercise 5	3-44

Chapter 4: Editing, Extruding, and Revolving the Sketches

Editing Sketched Entities	4-2
Extending Sketched Entities	4-2
Trimming Sketched Entities	4-2
Splitting Sketched Entities	4-3
Offsetting Sketched Entities	4-4
Mirroring Sketched Entities	4-5

Moving Sketched Entities	4-6
Rotating Sketched Entities	4-9
Creating Patterns	4-11
Creating Rectangular Patterns	4-11
Creating Circular Patterns	4-14
Writing Text in the Sketching Environment	4-17
Writing Regular Text	4-17
Writing Text Aligned to a Geometry	4-19
Inserting Images and Documents in Sketches	4-21
Editing Sketched Entities by Dragging	4-22
Tolerances	4-22
Converting the Base Sketch into a Base Feature	4-24
Extruding the Sketch	4-25
Revolving the Sketch	4-29
Direct Manipulation of Features by Using the Mini Toolbar <i>Enhanced</i>	4-32
Command Options	4-32
Manipulators	4-32
Rotating the View of a Model in 3D Space	4-33
Rotating the View of a Model Using the Orbit	4-33
Changing the View Using the ViewCube	4-35
Navigating the Model	4-38
Controlling the Display of Models	4-39
Setting the Visual Styles	4-39
Setting the Shadow Options	4-41
Setting the Camera Type	4-42
Tutorial 1	4-42
Tutorial 2	4-46
Tutorial 3	4-49
Tutorial 4	4-53
Self-Evaluation Test	4-56
Review Questions	4-56
Exercise 1	4-57
Exercise 2	4-58
Exercise 3	4-59
Exercise 4	4-59
Exercise 5	4-60

Chapter 5: Other Sketching and Modeling Options

Need for other Sketching Planes	5-2
Defining a New Sketching Plane	5-2
Work Features	5-3
Creating Work Planes	5-3
Creating Work Axes	5-13
Creating Work Points	5-17
Other Extrusion Options	5-21
Other Revolution Options	5-27
The Concept of Sketch Sharing	5-28

Tutorial 1	5-29
Tutorial 2	5-35
Tutorial 3	5-39
Self-Evaluation Test	5-44
Review Questions	5-44
Exercise 1	5-46
Exercise 2	5-46
Exercise 3	5-47
Exercise 4	5-48

Chapter 6: Advanced Modeling Tools-I

Advanced Modeling Tools	6-2
Creating Holes	6-2
Creating Fillets	6-12
Creating Edge Fillets	6-12
Creating Face Fillets	6-20
Creating Full Round Fillets	6-21
Creating Chamfers	6-22
Mirroring Features and Models	6-25
Creating Rectangular Patterns	6-28
Creating Circular Patterns	6-31
Creating Rib Features <i>Enhanced</i>	6-33
Thickening or Offsetting the Faces of Features	6-37
Creating the Embossed and Engraved Features	6-40
Applying Images on a Feature	6-42
Assigning Different Colors/Styles to a Model	6-43
Tutorial 1	6-44
Tutorial 2	6-50
Tutorial 3	6-56
Tutorial 4	6-63
Tutorial 5	6-67
Self-Evaluation Test	6-73
Review Questions	6-74
Exercise 1	6-75
Exercise 2	6-75

Chapter 7: Editing Features and Adding Automatic Dimensions to Sketches

Concept of Editing Features	7-2
Editing Features of a Model	7-2
Updating Edited Features	7-4
Editing Features Dynamically by Using 3D Grips	7-5
Editing the Sketches of Features	7-6
Redefining the Sketching Plane of a Sketched Feature	7-8
Suppressing Features	7-9
Unsuppressing the Suppressed Features	7-10

Moving the Faces of a Solid	7-10
Free Move Method	7-10
Direction and Distance Method	7-11
Points and Plane Method	7-11
Deleting Features	7-12
Copying and Pasting Features	7-12
Adding Automatic Dimensions to Sketches	7-15
Projecting Entities in the Sketching Environment	7-16
Projecting Edges or Faces	7-16
Projecting Cutting Edges	7-17
Project to 3D Sketch	7-18
Tutorial 1	7-18
Tutorial 2	7-25
Tutorial 3	7-30
Self-Evaluation Test	7-36
Review Questions	7-37
Exercise 1	7-38
Exercise 2	7-39
Exercise 3	7-40
Exercise 4	7-41
Exercise 5	7-42

New

Chapter 8: Advanced Modeling Tools-II

Advanced Modeling Tools	8-2
Creating Sweep Features	8-2
Sweep Along Edge	8-8
Creating Lofted Features	8-9
Creating Coil Features	8-19
Creating Threads	8-23
Creating Shell Features	8-26
Applying Drafts	8-29
Creating Split Features	8-32
Trimming Surfaces	8-35
Extending Surfaces	8-36
Deleting Faces	8-37
Replacing Faces with Surfaces	8-39
Creating Planar Boundary Patches	8-40
Stitching Surfaces	8-42
Working with the Sculpt Tool	8-43
Working with the Bend Part Tool	8-44
Reordering the Features	8-47
Using the Sketch Doctor	8-49
Using the Design Doctor	8-49
Tutorial 1	8-51
Tutorial 2	8-54
Tutorial 3	8-57
Tutorial 4	8-59

Enhanced

Tutorial 5	8-62
Self-Evaluation Test	8-65
Review Questions	8-65
Exercise 1	8-67
Exercise 2	8-68

Chapter 9: Assembly Modeling-I

Assembly Modeling	9-2
Types of Assemblies	9-2
Top-down Assemblies	9-2
Bottom-up Assemblies	9-3
Creating Top-down Assemblies	9-4
Creating Components in the Assembly Module	9-4
Creating Bottom-Up Assemblies	9-7
Placing Components in the Assembly File	9-7
Assembling Components by Using the Constrain Tool	9-8
Assembly Tab	9-8
Motion Tab	9-15
Transitional Tab	9-17
Constraint Set Tab	9-18
Specifying the Limits for Constraining	9-18
Assembling Parts by Using the Assemble Tool	9-19
Using ALT+Drag to Apply Assembly Constraints	9-20
Moving Individual Components	9-21
Rotating Individual Components in 3D Space	9-21
Tutorial 1	9-22
Tutorial 2	9-37
Tutorial 3	9-45
Self-Evaluation Test	9-55
Review Questions	9-56
Exercise 1	9-57

Chapter 10: Assembly Modeling-II

Editing Assembly Constraints	10-2
Editing Components	10-3
Editing Components in the Assembly File	10-3
Editing Components by Opening their Part Files	10-5
Creating subassemblies	10-5
Creating a Subassembly Using the Bottom-up Design Approach	10-6
Creating a Subassembly Using the Top-down Design Approach	10-6
Checking Degrees of Freedom of a Component	10-6
Creating the Pattern of Components in an Assembly	10-7
Component	10-7
Associative Tab	10-8
Rectangular Tab	10-9
Circular Tab	10-9

Replacing a Component from the Assembly File with another Component	10-10
Replacing a Single Instance of the Selected Component	10-10
Replacing all Instances of the Selected Component	10-11
Mirroring Subassemblies or Components of an Assembly	10-12
Copying Subassemblies or Components of an Assembly	10-15
Deleting Components	10-15
Editing the Pattern of Components	10-16
Making a Pattern Instance Independent	10-16
Deleting Assembly Constraints	10-17
Creating Assembly Section Views in the Assembly File	10-17
Analyzing Assemblies for Interference	10-18
Creating Design View Representations	10-20
Design View Representation Area	10-20
Positional Representation Area	10-21
Level of Detail Representation	10-21
Assembly by Driving Assembly Constraints	10-21
Creating Positional Representations	10-25
Viewing the Bill of Material of the Current Assembly	10-25
Working with Assembly Features	10-27
Tutorial 1	10-27
Tutorial 2	10-30
Tutorial 3	10-31
Self-Evaluation Test	10-38
Review Questions	10-38
Exercise 1	10-40

Enhanced

Chapter 11: Working with Drawing Views-I

The Drawing Module	11-2
Types of Views	11-3
Generating Drawing Views	11-5
Generating the Base View	11-5
Generating Projected Views	11-10
Generating Auxiliary Views	11-11
Generating Section Views	11-12
Generating Detail Views	11-14
Generating Broken Views	11-16
Generating Break Out Views	11-18
Generating Overlay Views	11-20
Generating Slice Views	11-22
Drafting Drawing Views	11-23
Editing Drawing Views	11-23
Deleting Drawing Views	11-24
Moving Drawing Views	11-25
Copying Drawing Views	11-25
Rotating Drawing Views	11-26
Changing the Orientation of Drawing Views	11-27
Assigning Different Hatch Patterns to Components in Assembly Section Views	11-28

Editing the Default Hatch Style of the Sectioned Objects	11-30
Excluding Components from Assembly Section Views	11-31
Tutorial 1	11-33
Tutorial 2	11-39
Self-Evaluation Test	11-44
Review Questions	11-44
Exercise 1	11-45

Chapter 12: Working with Drawing Views-II

Modifying Drawing Standards	12-2
Inserting Additional Sheets into Drawing	12-3
Activating a Drawing Sheet	12-3
Displaying Dimensions in Drawing Views	12-3
Retrieving Parametric Dimensions in Drawing Views	12-4
Adding Reference Dimensions	12-5
Modifying the Model Dimensions	12-5
Editing Drawing Sheets	12-5
Creating Dimension Styles	12-7
Applying Dimension Styles	12-8
Modifying a Dimension and its Appearance Using the Shortcut Menu	12-8
Adding the Parts List	12-9
Source Area	12-9
BOM Settings and Properties Area	12-10
Table Wrapping Area	12-10
Editing the Parts List	12-11
Column Chooser	12-12
Group Settings	12-12
Filter Settings	12-13
Export	12-13
Table Layout	12-13
Save Item Overrides to BOM	12-14
Member Selection	12-14
Adding/Removing Custom Parts	12-14
Shortcut Menu Options	12-14
Setting the Standard for the Parts List	12-15
Adding Balloons to Assembly Drawing Views	12-15
Adding Balloons to Selected Components	12-16
Adding Automatic Balloons	12-16
Adding Text to a Drawing Sheet	12-18
Adding Multiline Text without a Leader	12-19
Adding Multiline Text with Leader	12-20
Tutorial 1	12-21
Tutorial 2	12-26
Tutorial 3	12-31
Self-Evaluation Test	12-36
Review Questions	12-36
Exercise 1	12-37

Chapter 13: Presentation Module

The Presentation Module	13-2
Creating the Presentation View	13-3
Assembly Area	13-3
Explosion Method Area	13-4
Defining Units for Presentation Files	13-6
Tweaking Components in the Presentation View	13-6
Create Tweak Area	13-6
Transformations Area	13-8
Animating an Assembly	13-9
Rotating the Presentation View Precisely	13-15
Tutorial 1	13-16
Tutorial 2	13-23
Self-Evaluation Test	13-30
Review Questions	13-30
Exercise 1	13-32

Chapter 14: Working with Special Design Tools

Adaptive Parts	14-2
Defining Parameters	14-2
Working with iParts	14-7
Types of iPart Factories	14-7
Creating iPart Factories	14-7
Procedure to Create an iPart	14-13
Inserting an iPart into an Assembly	14-14
Changing the iParts in the Assembly File	14-17
Creating 3D Sketches	14-17
Line	14-18
Spline	14-18
Bend	14-19
Include Geometry	14-19
Intersection Curve	14-19
Helical Curve	14-20
Tutorial 1	14-21
Tutorial 2	14-25
Tutorial 3	14-29
Tutorial 4	14-33
Tutorial 5	14-36
Self-Evaluation Test	14-44
Review Questions	14-44
Exercise 1	14-46

Chapter 15: Working with Sheet Metal Components

The Sheet Metal Module	15-2
------------------------	------

Setting Sheet Metal Component Parameters	15-4
Setting the Sheet Metal Rule	15-4
Setting the Material Style	15-12
Creating Sheet Metal Components	15-14
Folding Sheet Metal Components	15-18
Adding Flanges to Sheet Metal Components	15-20
Creating Cuts in Sheet Metal Components	15-25
Creating Seams at the Corners of Sheet Metal Components	15-26
Bending the Faces of a Sheet Metal Component	15-30
Rounding the Corners of Sheet Metal Components	15-33
Chamfering the Corners of Sheet Metal Components	15-34
Punching 3D Shapes into Sheet Metal Components	15-35
Creating Hems	15-38
Creating Contour Flanges	15-40
Creating the Flat Patterns of Sheet Metal Components	15-42
Adding or Removing Material from the Flat Pattern	15-43
Tutorial 1	15-44
Tutorial 2	15-50
Self-Evaluation Test	15-56
Review Questions	15-57
Exercise 1	15-58

Chapter 16: Introduction to Weldments

Understanding Weldment Assemblies	16-2
Main Types of Welds in Autodesk Inventor	16-3
Cosmetic Welds	16-3
Fillet Welds	16-4
Groove Welds	16-4
Adding Welds to Assemblies	16-5
Assembling the Components of Weldment Assemblies	16-5
Preparing Assemblies for Weldments	16-6
Adding Welds	16-6
Creating Fillet Welds	16-6
Creating Cosmetic Welds	16-8
Creating Groove Welds	16-10
Tutorial 1	16-11
Tutorial 2	16-16
Self-Evaluation Test	16-20
Review Questions	16-21
Exercise 1	16-22

Chapter 17: Miscellaneous Tools

Copying the Sketches	17-2
Scaling the Sketches	17-3
Finding the Center of Gravity	17-3

Extracting the iFeature	17-4
Inserting the iFeature	17-6
Creating iMates	17-7
Applying iMates in the Assembly Environment	17-9
Interactively place with iMates	17-9
Automatically generate iMates on place	17-9
Viewing the iProperties	17-9
Creating User-Defined Drawing Sheets	17-11
Importing AutoCAD Blocks into Inventor	17-18
Tutorial 1	17-21
Tutorial 2	17-26
Self-Evaluation Test	17-29
Review Questions	17-30
Exercise 1	17-32

Chapter 18: Introduction to Inventor Fusion

The Inventor Fusion Environment	18-2
The Ribbon	18-5
Marking Menu	18-6
Sketching in Inventor Fusion	18-6
Modeling in Inventor Fusion	18-7
Creating Extruded Features	18-8
Creating Revolved Features	18-9
Creating Sweep Features	18-9
Creating Primitives	18-10
Creating Holes	18-13
Creating Fillets	18-14
Creating Chamfers	18-14
Creating Draft Features	18-16
Creating Shell Features	18-17
Editing in Inventor Fusion	18-17
Moving Features	18-17
Editing Features Using the Press/Pull Tool	18-18
Editing Edges	18-19
Creating Mirror Features	18-19
Creating Pattern	18-20
Associativity in Inventor Fusion	18-22
Dissociativity in Inventor Fusion	18-23
Dimensioning in Inventor Fusion	18-24
Tutorial 1	18-24
Tutorial 2	18-29
Self-Evaluation Test	18-40
Review Questions	18-41
Exercise 1	18-41