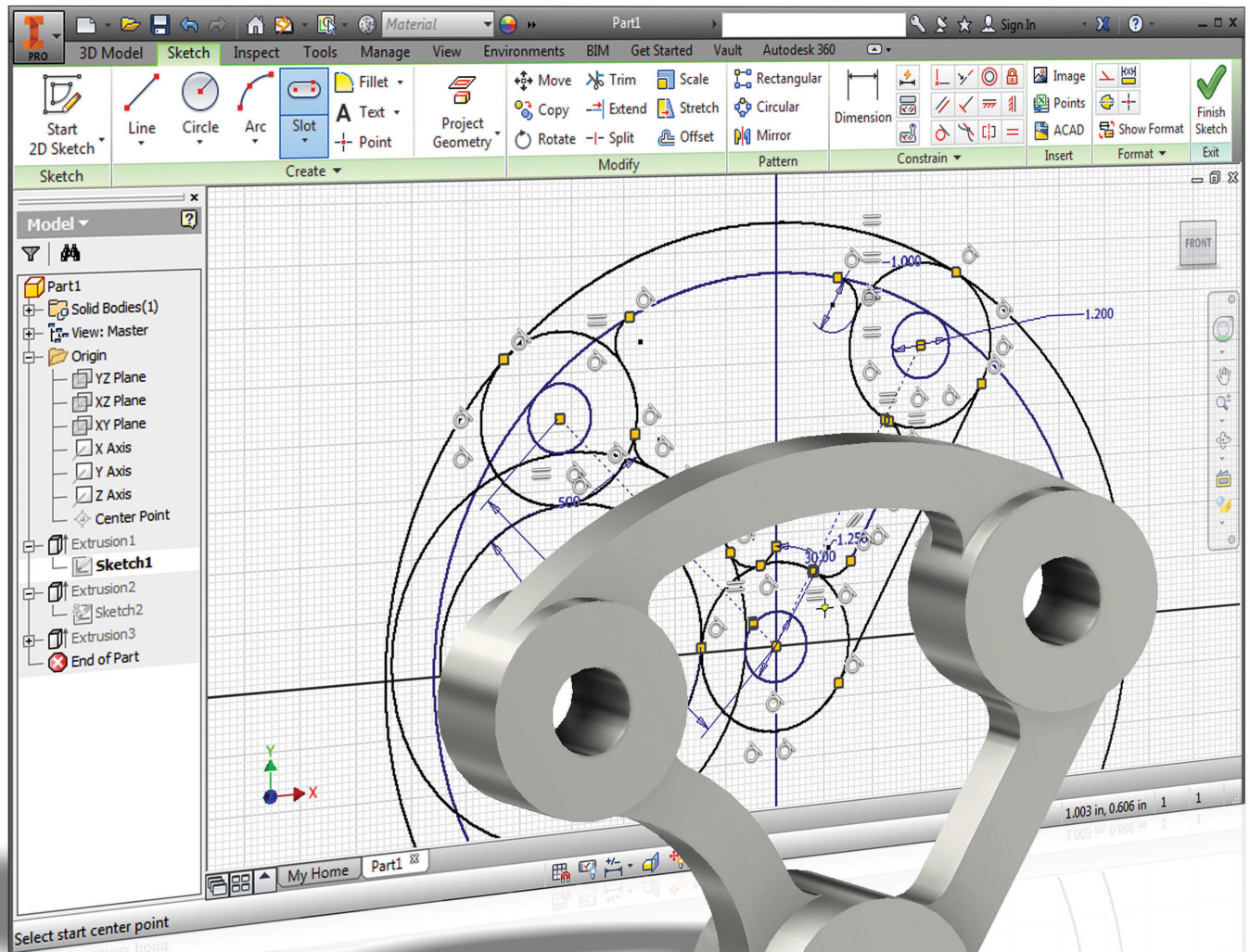


Parametric Modeling with Autodesk® Inventor® 2015



Randy H. Shih

Visit the following websites to learn more about this book:



[amazon.com](https://www.amazon.com)

[Google books](https://books.google.com)

[BARNES & NOBLE](https://www.barnesandnoble.com)

Table of Contents

Preface	i
Acknowledgments	ii
Table of Contents	iii
Autodesk Inventor Certified User Examination Overview	xiii
Chapter 1	
Getting Started	
Introduction	1-3
Development of Computer Geometric Modeling	1-3
Feature-Based Parametric Modeling	1-7
Getting Started with Autodesk Inventor	1-8
The Screen Layout and Getting Started Toolbar	1-9
The New File Dialog Box and Units Setup	1-10
Default Autodesk Inventor Screen Layout	1-11
Application Menu	1-12
Quick Access Toolbar	1-12
Ribbon Tabs and Tool Panels	1-12
Online Help Panel	1-12
Create Toolbar	1-13
Graphics Window	1-13
Message and Status Bar	1-13
Mouse Buttons	1-14
[Esc] - Canceling Commands	1-14
Autodesk Inventor Help System	1-15
Data Management using Inventor Project files	1-16
Setup of a New Inventor Project	1-17
The Content of an Inventor Project File	1-20
Leaving Autodesk Inventor	1-20
Chapter 2	
Parametric Modeling Fundamentals	
Introduction	2-3
The Adjuster Design	2-4
Starting Autodesk Inventor	2-4
The Default Autodesk Inventor Screen Layout	2-6
Sketch Plane – It is an XY Monitor, but an XYZ World	2-7
Creating Rough Sketches	2-9
Step 1: Creating a Rough Sketch	2-10
Graphics Cursors	2-10
Geometric Constraint Symbols	2-11
Step 2: Apply/Modify Constraints and Dimensions	2-12
Dynamic Viewing Functions – <i>Zoom</i> and <i>Pan</i>	2-15

Modifying the Dimensions of the Sketch	2-15
Step 3: Completing the Base Solid Feature	2-16
Isometric View	2-17
Dynamic Rotation of the 3-D block - 3D Orbit	2-18
Dynamic Viewing - Quick Keys	2-20
Viewing Tools – Standard Toolbar	2-21
Display Modes	2-25
Orthographic vs. Perspective	2-25
Disable the Heads-Up Display Option	2-26
Step 4-1: Adding an Extruded Feature	2-27
Step 4-2: Adding a Cut Feature	2-32
Step 4-3: Adding another Cut Feature	2-35
Save the Model	2-38
Review Questions	2-39
Exercises	2-40

Chapter 3

Constructive Solid Geometry Concepts

Introduction	3-3
Binary Tree	3-4
The Locator Design	3-5
Modeling Strategy - CSG Binary Tree	3-6
Starting Autodesk Inventor	3-7
Base Feature	3-8
<i>GRID</i> intervals Setup	3-9
Model Dimensions Format	3-12
Modifying the Dimensions of the Sketch	3-12
Repositioning Dimensions	3-13
Using the Measure Tools	3-14
Completing the Base Solid Feature	3-17
Creating the Next Solid Feature	3-18
Creating a CUT Feature	3-22
Creating a PLACED FEATURE	3-25
Creating a Rectangular Cut Feature	3-27
Save the Model	3-29
Review Questions	3-30
Exercises	3-31

Chapter 4

Model History Tree

Introduction	4-3
The <i>Saddle Bracket</i> Design	4-4
Starting Autodesk Inventor	4-4
Modeling Strategy	4-5
The <i>Autodesk Inventor Browser</i>	4-6

Creating the Base Feature	4-6
Adding the Second Solid Feature	4-9
Creating a 2D Sketch	4-10
Renaming the Part Features	4-12
Adjusting the Width of the Base Feature	4-13
Adding a Placed Feature	4-14
Creating a Rectangular Cut Feature	4-16
History-Based Part Modifications	4-17
A Design Change	4-18
Assigning and Calculating the Associated Physical Properties	4-21
Review Questions	4-23
Exercises	4-24

Chapter 5

Parametric Constraints Fundamentals

CONSTRAINTS and RELATIONS	5-3
Create a Simple Triangular Plate Design	5-3
Fully Constrained Geometry	5-4
Starting Autodesk Inventor	5-4
Displaying Existing Constraints	5-5
Applying Geometric/Dimensional Constraints	5-7
Over-Constraining and Driven Dimensions	5-11
Deleting Existing Constraints	5-12
Using the Auto Dimension Command	5-13
Constraint and Sketch Settings	5-18
Parametric Relations	5-19
Dimensional Values and Dimensional Variables	5-21
Parametric Equations	5-22
Viewing the Established Parameters and Relations	5-24
Saving the Model File	5-25
Using the Measure Tools	5-26
Review Questions	5-30
Exercises	5-31

Chapter 6

Geometric Construction Tools

Introduction	6-3
The Gasket Design	6-3
Modeling Strategy	6-4
Starting Autodesk Inventor	6-5
Create a 2D Sketch	6-6
Edit the Sketch by Dragging the Sketched Entities	6-8
Add Additional Constraints	6-10
Use the <i>Trim</i> and <i>Extend</i> Commands	6-11
The <i>Auto Dimension</i> Command	6-13

Create Fillets and Completing the Sketch	6-15
Fully Constrained Geometry	6-16
Profile Sketch	6-18
Redefine the Sketch and Profile	6-19
Create an OFFSET Cut Feature	6-23
Review Questions	6-26
Exercises	6-27

Chapter 7

Parent/Child Relationships and the BORN Technique

Introduction	7-3
The BORN Technique	7-3
The U-Bracket Design	7-4
Sketch Plane Settings	7-5
Apply the BORN Technique	7-6
Create the 2-D Sketch of the Base Feature	7-8
Create the First Extrude Feature	7-11
The Implied Parent/Child Relationships	7-12
Create the Second Solid Feature	7-12
Create the First Cut Feature	7-16
The Second Cut Feature	7-17
Examine the Parent/Child Relationships	7-19
Modify a Parent Dimension	7-20
A Design Change	7-21
Feature Suppression	7-22
A Different Approach to the CENTER_DRILL Feature	7-23
Suppress the Rect_Cut Feature	7-25
Create a Circular Cut Feature	7-26
A Flexible Design Approach	7-28
View and Edit Material Properties	7-29
Review Questions	7-31
Exercises	7-32

Chapter 8

Part Drawings and Associative Functionality

Drawings from Parts and Associative Functionality	8-3
Starting Autodesk Inventor	8-4
Drawing Mode - 2D Paper Space	8-4
Drawing Sheet Format	8-6
Using the Pre-Defined Drawing Sheet Formats	8-8
Delete, Activate, and Edit Drawing Sheets	8-10
Add a Base View	8-11
Create Projected Views	8-12
Adjust the View Scale	8-13
Repositioning Views	8-14

Display Feature Dimensions	8-15
Repositioning and Hiding Feature Dimensions	8-17
Add Additional Dimensions – Reference Dimensions	8-19
Add Center Marks and Center Lines	8-20
Complete the Drawing Sheet	8-23
Associative Functionality – Modifying Feature Dimensions	8-24
3D Annotations in Isometric Views	8-27
Review Questions	8-35
Exercises	8-36

Chapter 9

Datum Features and Auxiliary Views

Work Features	9-3
Auxiliary Views in 2D Drawings	9-3
The <i>Rod-Guide</i> Design	9-3
Modeling Strategy	9-4
Starting Autodesk Inventor	9-5
Apply the BORN Technique	9-5
Creating the Base Feature	9-7
Create an Angled Work Plane	9-9
Create a 2D Sketch on the Work Plane	9-10
Use the Projected Geometry Option	9-10
Complete the Solid Feature	9-14
Create an Offset Work Plane	9-15
Create another Cut Feature Using the Work Plane	9-16
Start a New 2D Drawing	9-18
Add a Base View	9-19
Create an Auxiliary View	9-20
Display Feature Dimensions	9-22
Adjust the View Scale	9-24
Retrieving Dimensions in the Auxiliary View	9-25
Add Center Marks and Center Lines	9-28
Complete the Title Block with iProperties	9-31
Edit the Isometric view	9-33
Review Questions	9-34
Exercises	9-35

Chapter 10

Symmetrical Features in Designs

Introduction	10-3
A Revolved Design: PULLEY	10-3
Modeling Strategy - A Revolved Design	10-4
Starting Autodesk Inventor	10-5
Setup the Display of the Sketch Plane	10-5
Create the 2-D Sketch for the Revolved Feature	10-6

Create the Revolved Feature	10-10
Mirroring Features	10-11
Create a Pattern Leader Using Construction Geometry	10-13
Circular Pattern	10-18
Examine the Design Parameters	10-20
Drawing Mode – Defining New Border and Title Block	10-20
Create a Drawing Template	10-24
Create the Necessary Views	10-25
Retrieve Dimensions – Features Option	10-28
Associative Functionality – A Design Change	10-30
Add Center Lines to the Pattern Feature	10-32
Complete the Drawing	10-33
Review Questions	10-36
Exercises	10-37

Chapter 11

Advanced 3D Construction Tools

Introduction	11-3
A Thin-Walled Design: <i>Dryer Housing</i>	11-3
Modeling Strategy	11-4
Starting Autodesk Inventor	11-5
Set Up the Display of the Sketch Plane	11-5
Create the 2-D Sketch for the Base Feature	11-6
Create a Revolved Feature	11-9
Create Offset Work Planes	11-10
Start 2D Sketches on the Work Planes	11-11
Create a Lofted Feature	11-14
Create an Extruded Feature	11-16
Complete the Extruded Feature	11-18
Create 3D Rounds and Fillets	11-19
Create a Shell Feature	11-20
Create a Pattern Leader	11-21
Create a Rectangular Pattern	11-24
Create a Swept Feature	11-26
Define a 2D Sweep Path	11-26
Define the Sweep Section	11-28
Complete the Swept Feature	11-30
Review Questions	11-32
Exercises	11-33

Chapter 12

Sheet Metal Designs

Sheet Metal Processes	12-3
Sheet Metal Modeling	12-5
K-Factor	12-6

The Actuator Bracket Design	12-7
Starting Autodesk Inventor	12-8
Sheet Metal Defaults	12-9
Create the Base Face Feature of the Design	12-12
Using the Flange Command	12-15
Mirroring Features	12-19
Create a Cut Feature	12-20
Create a Fold Feature	12-22
Create the Associated Flat Pattern	12-25
Confirm the Flattened Length	12-26
Create a 2D Sheet Metal Drawing	12-27
Review Questions	12-34
Exercises	12-35

Chapter 13

Assembly Modeling - Putting It All Together

Introduction	13-3
Assembly Modeling Methodology	13-4
The Shaft Support Assembly	13-5
Additional <i>Parts</i>	13-5
(1) Collar	13-5
(2) Bearing	13-6
(3) Base-Plate	13-6
(4) Cap-Screw	13-7
Starting Autodesk Inventor	13-8
Placing the First Component	13-9
Placing the Second Component	13-10
Degrees of Freedom and Constraints	13-11
Assembly Constraints	13-12
Apply the First Assembly Constraint	13-15
Apply a Second Mate Assembly Constraint	13-16
Constrained Move	13-17
Apply a Flush Constraint	13-18
Placing the Third Component	13-20
Applying an Insert Constraint	13-21
Assemble the Cap-Screws	13-22
Exploded View of the Assembly	13-23
Editing the Components	13-25
Adaptive Design Approach	13-26
Delete and Re-apply Assembly Constraints	13-30
Setup a Drawing of the Assembly Model	13-32
Creating a Parts List	13-34
Edit the Parts List	13-35
Change the Material Type	13-37
Add the Balloon Callouts	13-39
Complete the Title Block Using the iProperties option	13-39

Bill of Materials	13-41
(a) BOM from Parts List	13-41
(b) BOM from Assembly Model	13-42
Review Questions	13-43
Exercises	13-44

Chapter 14

Content Center and Basic Motion Analysis

Introduction	14-3
The Crank-Slider Assembly	14-4
Create the Required Parts	14-4
Starting Autodesk Inventor	14-6
Placing the First Component	14-7
Placing the Second Component	14-8
Apply the Assembly Constraints	14-9
Apply a Second MATE Constraint	14-10
Constrained Move	14-11
Place the Third Component	14-11
Assemble the CS-Rod Part	14-14
Make a Copy of the PIN Part	14-15
Assemble the CS-Slider Part	14-16
Add an Angle Constraint to Fully Constrain the Assembly	14-21
Interference Analysis	14-23
Basic Motion Analysis	14-24
3D Grip Editing the CS-Slider Part	14-27
Review Questions	14-31
Exercises	14-32

Chapter 15

2D Design Reuse, Collision and Contact

Introduction	15-3
The Geneva CAM Assembly	15-4
Internet Download the Geneva-Wheel DWG File	15-4
Opening AutoCAD DWG File in Inventor	15-5
Use the Measuring Tools	15-6
Switch to the AutoCAD DWG Layout	15-8
2D Design Reuse	15-10
Complete the Imported Sketch	15-14
Create the First Solid Feature	15-16
Create a Mirrored Feature	15-17
Circular Pattern	15-18

Complete the Geneva Wheel Design	15-19
Additional Parts	15-20
Start a New Assembly	15-22
Placing the Second Component	15-23
The Assembly Joint Command	15-24
Create a Joint Connection	15-25
Constrained Move	15-26
Placing a Copy of the Geneva-Driver Part	15-26
Create a Second Joint Connection	15-27
Assemble the Geneva-Pin Part	15-28
Repositioning the Pieces	15-30
Animation with Drive Tool	15-31
Use the Inventor Contact Solver	15-33
Constrained Move with Contact Solver	15-35
Review Questions	15-36
Exercises	15-37

Chapter 16

Introduction to Stress Analysis

Introduction	16-2
Problem Statement	16-4
Preliminary Analysis	16-4
• Maximum Normal Stress	16-5
• Maximum Displacement	16-5
Finite Element Analysis Procedure	16-6
Create the <i>Autodesk Inventor</i> Part	16-7
Create the 2D Sketch for the Plate	16-7
Assigning the Material Properties	16-10
Switch to the <i>Stress Analysis</i> Module	16-11
Create an FEA Simulation	16-12
Apply Constraints and Loads	16-14
Create a Mesh and Run the Solver	16-16
Refinement of the FEA Mesh – Global Element Size	16-18
Refinement of the FEA Mesh – Local Element Size	16-20
Comparison of Results	16-23
Create an HTML Report	16-24
Geometric Considerations of Finite Elements	16-25
Conclusion	16-26
Summary of Modeling Considerations	16-26
Review Questions	16-27
Exercises	16-28

Appendix

Index

Notes:

Autodesk Inventor Certified User Examination Overview

The Autodesk Inventor Certified User examination is a performance-based exam. The examination is comprised of approximately 30 questions to be completed in two hours. The test items will require you to use the Autodesk Inventor software to perform specific tasks and then answer questions about the tasks.

Performance-based testing is defined as *Testing by Doing*. This means you actually perform the given task then answer the questions regarding the task. Performance-based testing is widely accepted as a better way of insuring the users have the skills needed, rather than just recalling information.

The Autodesk Inventor Certified User examination is designed to test specific performance tasks in the following 9 sections:

Section 1: User Interface

Objectives: Primary Environments, UI Navigation/Interaction, Graphics Window Display, Navigation Control.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Screen Layout	1-8, 1-10
Inventor Overview	1-8
New File	1-10
Application Menu	1-11
Mouse Buttons	1-14
Inventor Help System	1-15
Dynamic Viewing – Quick Keys	2-20
Viewing Tools	2-21
2D/3D Navigation Wheels	2-23
Display Modes	2-25
Browser	4-6

Section 2: File Management

Objectives: Project Files.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Data Management	1-16
New Inventor Project	1-17
Content of the Inventor Project File	1-20
Activate an Inventor Project File	2-5

Section 3: Sketches

Objectives: Creating 2D Sketches, Draw Tools, Sketch Constraints, Pattern Sketches, Modify Sketches, Format Sketches, Sketch Doctor, Shared Sketches, Sketch Parameters.

Certification Examination Performance Task	Covered in this book on Chapter – Page
2D Sketch Tool	2-8
Geometric Constraint Symbols	2-11
Apply/Modify Constraints and Dimensions	2-12
Modifying Dimensions	2-15
Profile	2-30
<i>GRID</i> and <i>SNAP</i> Intervals Setup	3-9
Two Point Rectangle	3-10
Modifying Dimensions	3-12
Repositioning Dimensions	3-13
Center Point Circle	3-19
Create Dimensions	3-12
Show Dimensions	4-13
Concentric	4-14
Edit Sketch	4-18
Look At	4-18
Horizontal Constraint	5-5
Geometric Constraints Tool	5-7
Fix Constraint	5-8
General Dimension Tool	5-9
Vertical Constraint	5-9
Undo	5-9
Show All Constraints	5-10
Over Constraining	5-11
Driven Dimensions	5-11
Delete Constraints	5-12
Auto Dimension	5-13
Tangent Constraint	5-15
Coincident Constraint	5-17
Constraint Settings	5-18
Parametric Equations	5-22
Parameters	5-24
Editing by Dragging	6-8
Tangent Constraint	6-10
Extend	6-11
Trim	6-12
Auto Dimension	6-13
Fillet, 2D	6-15
Fix Constraint	6-16

Edit Sketch	6-19
Offset command	6-23
Trim	7-9
Auto Dimension	7-11
Implied Parent/Child Relationships	7-12
Center Point Arc	7-13
Show Dimensions	7-20
2D Fillet	9-7
Project Geometry	9-10
Reference Geometry	9-10
Linear Diameter	10-7
Construction Geometry	10-13
Construction Style	10-15
Fillet, 2-D	11-8
Project Geometry	11-13
Edit Component	14-27

Section 4: Parts

Objectives: Creating parts, Work Features, Pattern Features, Part Properties.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Extrude Command	2-16
Adding an Extruded Feature	2-27
Cut Feature	2-32
Extrude Join	3-20
Sketched Feature	3-25
Placed Feature	3-25
Hole Tool	3-25
Cut Feature	3-28
Browser	4-6
Base Feature	4-6
Symmetric Option	4-8
Local Update	4-13
Creating a Placed Feature	4-14
Hole Tool	4-14
To Next Option	4-16
Profile Sketch	6-18
Edit Sketch	6-20
Edit Feature	6-20
Symmetric Option	7-15
Feature Suppression	7-22
Material Properties	7-29
Angled Work Plane	9-9
Offset Work Plane	9-15

Revolve Command	10-10
Mirror Command	10-11
Mirror Plane	10-11
Circular Pattern	10-18
Revolve Feature	11-9
Offset Work Plane	11-10
Loft Command	11-14
Fillet, 3-D	11-19
Shell Command	11-20
Rectangular Pattern	11-24
Sweep Path	11-26
Sweep Section	11-28
Swept Feature	11-30
Mirror Features	12-19
Threads and Coil	13-7

Section 5: Assemblies

Objectives: Creating Assemblies, Viewing Assemblies, Animation Assemblies, Adaptive Features, Parts, and Subassemblies.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Assembly Modeling Methodology	13-4
Assembly Module	13-8
Place Component	13-9
Degrees of Freedom	13-11
Assembly Constraints	13-12
Mate Constraint	13-15
Constrained Move	13-17
Flush Constraint	13-18
Insert Constraint	13-21
Move Component	13-23
Create Component	13-26
Adaptive	13-28
Delete Constraints	13-30
Content Center	14-3
Assembly Module	14-6
Place Component	14-7
Constraint Command	14-9
Mate Constraint	14-9
Degrees of Freedom	14-10
Content Center – Fastener	14-11
Constrained Move	14-11
Cylindrical Pins	14-12

Angle Constraint	14-21
Edit Angle	14-23
Analyze Interference	14-23
Drive Constraint	14-24
Collision Detection	14-27
3D Grips	14-28
Place Component	15-22
Joint Command	15-24
Rotational	15-25
Degrees of Freedom	15-26
Rigid	15-28
Flip Component	15-29
Contact Solver	15-33
Contact Set	15-33
Constrained Move	15-35

Section 6: Presentations

Objectives: Creating Presentations.

- Refer to the Autodesk Inventor Help system on this topic.

Section 7: Drawings

Objectives: Create drawings.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Drawing Mode	8-4
Drawing Sheet Format	8-6
Styles and Standard Editor	8-6
Third Angle of Projection	8-6
Drawing Resources	8-8
Sheet Formats	8-8
Delete Sheet	8-10
Activate Sheet	8-10
Edit Sheet	8-10
Base View	8-11
Projected View	8-12
Edit View	8-13
Repositioning Views	8-14
Annotation Tab	8-15
Retrieve Dimensions	8-15
Reference Dimensions	8-19
Center Mark	8-20
Centerline Bisector	8-20
Text	8-23

Delete View	8-27
3D Annotations	8-27
Center Lines	8-31
Leader Text	8-32
Format Text	8-33
Diameter Symbol	8-33
Auxiliary View	9-3
Drawing Mode	9-18
Edit Sheet	9-18
Base View	9-19
Auxiliary View	9-20
Drawing Annotation Tool	9-22
Retrieve Dimensions	9-23
Edit View	9-24
Select Dimensions	9-25
Arrowheads Inside	9-26
Center Mark	9-28
Centerline Bisector	9-28
Text	9-31
Define New Border	10-21
Define New Title Block	10-22
Drawing Template	10-24
Section View	10-25
Hidden Line Removed	10-26
Retrieve Dimensions	10-28
Centered Pattern	10-32
Arrowheads Inside	10-33
Single Dimension Line	10-33
Centerline Bisector	10-34
Drawing Mode	12-27
Base View	12-27
Projected View	12-29
Sheet Metal View	12-29
Flat Pattern View	12-30
Drawing Annotation Tool	12-31
Center Mark	12-31
Retrieve Dimensions	12-32
Assembly Drawing	13-32
Parts List	13-34
Edit Parts List	13-35
iProperties	13-37
Balloon	13-39
Bill of Materials	13-41
Export Bill of Materials	13-41

Section 8: Sheet Metal

Objectives: Creating Sheet Metal Parts, Modify Sheet Metal Parts, Flat Pattern.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Sheet Metal template file	12-8
Sheet Metal Defaults	12-9
Edit Sheet Metal Rule	12-9
Edit Unfolding Rule	12-10
Face command	12-13
Unfold Options	12-13
Bend Option	12-13
Flange command	12-15
Fold Feature command	12-23
Flat Pattern command	12-25

Section 9: Visualization

Objectives: Create Rendered Images, Animate an Assembly.

Certification Examination Performance Task	Covered in this book on Chapter – Page
Drive Constraint	14-24
Minimum Button	14-24
Forward Button	14-24
Reverse Button	14-25
Record	14-25
Increment Option	14-26
Repetitions Option	14-26
Collision Detection	14-27
Joint Command	15-24
Rotational	15-25
Rigid	15-28
Flip Component	15-29
Drive Tool	15-31
Contact Solver	15-33
Contact Set	15-33

- Every effort has been made to cover the exam objectives included in the Autodesk Inventor Certified User Examination. However, the format and topics covered by the examination are constantly changing. Students planning to take the Certified User Examination are advised to visit the Autodesk website and obtain information regarding the format and details about the Autodesk Inventor Certified User Examination.