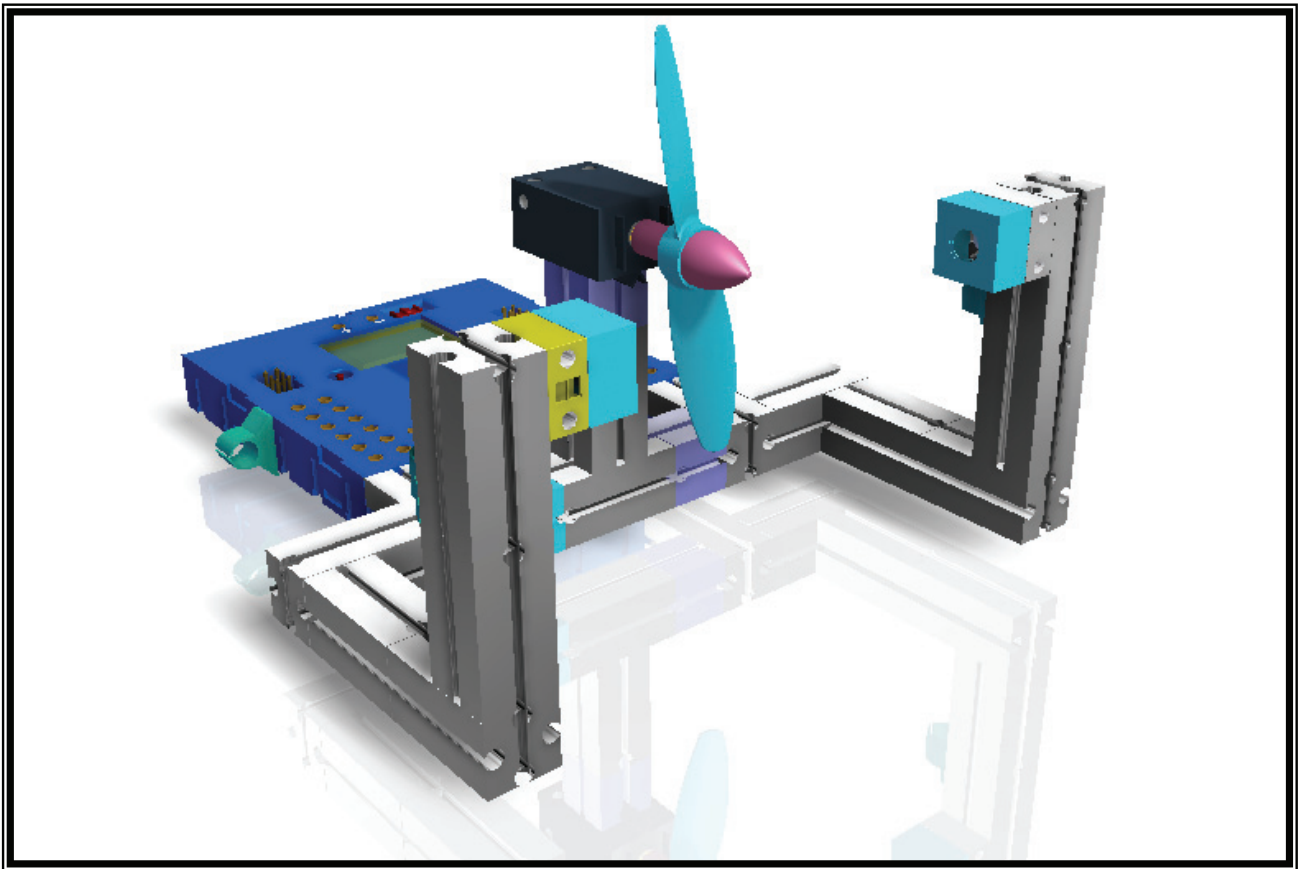


Tools for Design

with FischerTechnik:

AutoCAD® 2012 and Autodesk® Inventor® 2012

2D Drawing ❖ 3D Modeling ❖ Hand Sketching



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Table of Contents

Preface	i
Acknowledgments	ii
Introduction	
Getting Started	
Introduction	Intro-2
Development of Computer Aided Design	Intro-2
Why use AutoCAD [®] 2012?	Intro-5
Why use Autodesk [®] Inventor [®] 2012?	Intro-6
Tutorial Style Lessons	Intro-7
Section I – AutoCAD[®]	
Chapter 1	
Fundamentals of AutoCAD[®]	
Getting started with AutoCAD [®] 2012	1-2
AutoCAD [®] 2012 Screen Layout	1-3
Application Menu	1-4
Workspace Switching	1-4
Quick Access Toolbar	1-4
AutoCAD Menu Bar	1-4
Graphics Window	1-4
Graphics Cursor or Crosshairs	1-5
Command prompt Area	1-5
Cursor Coordinates	1-5
Status Toolbar	1-5
Ribbon Tabs and Panels	1-5
Draw and Modify Toolbar Panel	1-6
Layers Control Toolbar Panel	1-6
Annotation Toolbar Panel	1-6
Layout/Model Toolbar	1-6
Viewing Toolbar	1-6
Mouse Buttons	1-7
[Esc] – Canceling commands	1-7
On-Line Help	1-8
Leaving AutoCAD [®] 2012	1-9
Creating a CAD file folder	1-9

Computer Aided Design/Drafting	1-10
Starting Up AutoCAD® 2012	1-10
Drawing Units display Setup	1-12
Drawing Area Setup	1-13
Drawing Lines with the <i>LINE</i> command	1-15
Visual reference	1-17
<i>GRID</i> ON	1-18
<i>SNAP Mode</i> ON	1-19
Using the <i>ERASER</i>	1-20
Repeat the last command	1-21
The CAD Database and the User Coordinate System	1-22
Changing to the 2D UCS icon Display	1-23
Cartesian and Polar Coordinate Systems	1-24
Absolute and Relative coordinates	1-24
Defining Positions	1-25
The <i>Guide Plate</i>	1-26
Creating <i>Circles</i>	1-30
Save the CAD file	1-32
Close the Current Drawing	1-33
The <i>Spacer</i> design	1-34
Start a New Drawing	1-34
Drawing Units Setup	1-35
Drawing Area Setup	1-36
Using the Line Command	1-38
Using the <i>ERASE</i> command	1-40
Using the Arc Command	1-40
Using the Circle command	1-42
Saving the CAD Design	1-43
Exit AutoCAD	1-43
Review Questions	1-44
Exercises	1-45

Chapter 2

Basic Object Construction and Dynamic Input Using AutoCAD®

Introduction	2-2
Starting Up AutoCAD® 2012	2-3
Dynamic Input	2-3
The <i>RockerArm</i> Design	2-6
Activate the Startup option	2-7
Drawing Units Display Setup	2-8
<i>GRID</i> and <i>SNAP</i> intervals Setup	2-9
Drawing Area Setup	2-10
Referencing the World Coordinate System	2-11
Creating <i>Circles</i>	2-12

<i>Object Snap</i> toolbar	2-13
Using the <i>LINE</i> command	2-14
Creating a <i>TTR circles</i>	2-16
Using the <i>TRIM</i> Command	2-18
Using the <i>Polygon</i> command	2-20
Creating a Concentric <i>Circle</i>	2-22
Using the <i>QuickCal</i> Calculator to Measure Distance and Angle	2-23
Saving the CAD file	2-27
Exit AutoCAD	2-27
Review Questions	2-28
Exercises	2-29

Chapter 3

Geometric Construction and Editing Tools Using AutoCAD®

Geometric Constructions	3-2
Starting Up AutoCAD® 2012	3-3
Geometric Construction - CAD Method	3-4
• Bisection of a Line or Arc	3-4
• Bisection of An Angle	3-7
• Transfer of An Angle	3-9
• Dividing a Given Line into A Number of Equal Parts	3-13
• Circle Through Three Points	3-15
• Line Tangent To A circle from A Given Point	3-16
• Circle of a Given Radius Tangent To Two Given Lines	3-17
The <i>Gasket</i> design	3-20
Drawing Units Display Setup	3-21
<i>GRID</i> and <i>SNAP</i> intervals Setup	3-22
Using the <i>LINE</i> command	3-23
<i>Object Snap Toolbar</i>	3-25
Using the <i>EXTEND</i> Command	3-28
Using the <i>TRIM</i> Command	3-29
Creating a <i>TTR circle</i>	3-30
Using the <i>FILLET</i> command	3-31
Converting objects into a <i>Polyline</i>	3-32
Using the <i>OFFSET</i> command	3-33
Using the Area Inquiry Tool to Measure Area and Perimeter	3-34
Using the <i>EXPLODE</i> command	3-36
Create another <i>FILLET</i>	3-36
Saving the CAD file	3-37
Exit AutoCAD	3-37
Review Questions	3-38
Exercises	3-39

Chapter 4

Orthographic Views in Multiview Drawings Using AutoCAD®

Introduction	4-2
The LOCATOR Design	4-2
The <i>Locator</i> part	4-3
Starting Up AutoCAD® 2012	4-3
<i>Layers</i> setup	4-4
Drawing <i>Construction Lines</i>	4-5
Using the <i>OFFSET</i> command	4-5
Set Layer <i>Object</i> as the current layer	4-7
Using the <i>Running Object Snaps</i>	4-7
Creating <i>Object lines</i>	4-9
Turn <i>OFF</i> the construction Lines	4-10
Adding more objects in the Front View	4-10
AutoCAD's AutoSnap™ and AutoTrack™ features	4-11
Adding more objects in the Top View	4-13
Drawing using the <i>Miter Line</i> method	4-17
More <i>Layers setup</i>	4-19
Top View to Side View Projection	4-20
Completing the Front View	4-22
Object Information Using the List command	4-24
Object Information Using the Properties command	4-25
Review Questions	4-26
Exercises	4-27

Chapter 5

Basic Dimensioning and Notes Using AutoCAD®

Dimensioning Tools in AutoCAD®	5-2
The P-Bracket Design	5-2
Starting Up AutoCAD® 2012	5-3
<i>Layers</i> setup	5-4
The P-Bracket Design	5-5
Drawing Construction Lines	5-6
Using the <i>OFFSET</i> command	5-6
Set Layer <i>Object_Lines</i> as the <i>current layer</i>	5-8
Creating Object lines	5-8
Creating Hidden lines	5-9
Creating Center lines	5-10
Turn off the Construction lines	5-10
Using the <i>FILLET</i> command	5-11
Saving the Completed CAD Design	5-12
Accessing the Dimensioning Commands	5-13
The <i>Dimension toolbar</i>	5-14

Using <i>Dimension Style Manager</i>	5-14
Dimensions Nomenclature and Basics	5-15
Using the <i>Center Mark</i> command	5-18
Adding <i>Linear</i> Dimensions	5-19
Adding <i>Angular</i> Dimensions	5-20
Adding <i>Radius</i> and <i>Diameter</i> Dimensions	5-21
Using the <i>SINGLE LINE TEXT</i> command	5-22
Adding Special Characters	5-23
Saving the design	5-24
A Special Note on Layers containing Dimensions	5-24
Review Questions	5-25
Exercises	5-26

Chapter 6

Pictorials and Sketching

Engineering Drawings, Pictorials and Sketching	6-2
Isometric Sketching	6-7
Isometric Sketching Exercises	6-9
Oblique Sketching	6-10
Oblique Sketching Exercises	6-11
Perspective Sketching	6-12
One-point Perspective Sketching	6-13
Two-point Perspective Sketching	6-14
Perspective Sketching Exercises	6-15
Questions	6-16
Exercises	6-17

Section II – Autodesk® Inventor®

Chapter 7

Parametric Modeling Fundamentals – Autodesk® Inventor®

Getting Started with Autodesk® Inventor®	7-2
The Screen layout and Getting Started toolbar	7-3
The New File dialog box and Units Setup	7-4
Autodesk® Inventor® Screen Layout	7-5
Application Menu	7-6
Quick Access Toolbar	7-6
Ribbon Tabs	7-6
Online Help Panel	7-6
2D Sketch Toolbar	7-7
Graphics Window	7-7

Graphics Cursor or Crosshairs	7-7
Message and Status bar Area	7-8
Mouse Buttons	7-8
[Esc] – Canceling commands	7-9
On-Line Help	7-9
Data Management using Inventor Project files	7-10
Setup of a New Inventor Project	7-12
The Content of the Inventor Project File	7-14
Leaving Autodesk® Inventor®	7-14
Feature-based Parametric Modeling	7-15
The Adjuster design	7-16
Starting Autodesk® Inventor®	7-16
Autodesk® Inventor® Screen Layout	7-18
Creating Rough Sketches	7-19
Step 1: Creating a rough sketch	7-20
Graphics Cursors	7-20
Geometric Constraint Symbols	7-21
Step 2: Apply/modify constraints and dimensions	7-22
Dynamic Viewing Functions – <i>Zoom</i> and <i>Pan</i>	7-25
Modifying the dimensions of the sketch	7-25
Step 3: Completing the Base Solid Feature	7-26
Isometric View	7-27
Dynamic Rotation of the 3-D block – 3D Orbit	7-28
Dynamic Viewing – Quick Keys	7-29
Viewing Tools – Standard Toolbar	7-30
Display Modes	7-35
Orthographic vs. Perspective	7-35
Disable the Heads-Up Display Option	7-36
Sketch plane – It is an XY CRT, but an XYZ World	7-37
Step 4-1: Adding an extruded feature	7-39
Step 4-2: Adding a cut feature	7-43
Save the Model	7-46
Questions	7-47
Exercises	7-48

Chapter 8

Constructive Solid Geometry Concepts – Autodesk® Inventor®

Introduction	8-2
Binary Tree	8-3
The Locator Design	8-4
Modeling Strategy – CSG Binary Tree	8-5
Starting Autodesk Inventor	8-6
<i>GRID</i> and <i>SNAP</i> intervals Setup	8-7
Base Feature	8-8

Model Dimensions Format	8-10
Modifying the dimensions of the sketch	8-10
Repositioning Dimensions	8-11
Using the Measure Tools	8-12
Completing the Base Solid Feature	8-15
Creating the next solid feature	8-16
Creating a CUT Feature	8-20
Creating a PLACED FEATURE	8-23
Creating a Rectangular Cut Feature	8-25
Questions	8-27
Exercises	8-28

Chapter 9

Model History Tree – Autodesk® Inventor®

Introduction	9-2
The <i>Saddle Bracket</i> Design	9-3
Starting Autodesk Inventor	9-3
Modeling Strategy	9-4
The <i>Autodesk Inventor Browser</i>	9-5
Creating the Base Feature	9-5
Adding the Second Solid Feature	9-8
Creating a 2D sketch	9-9
Renaming the Part Features	9-11
Adjusting the Width of the Base Feature	9-12
Adding a Placed Feature	9-13
Creating a Rectangular Cut Feature	9-15
History-based Part Modifications	9-16
A Design change	9-17
Assigning and Calculating the Associated Physical Properties	9-20
Questions	9-22
Exercises	9-23

Chapter 10

Parametric Constraints Fundamentals Using Autodesk® Inventor®

CONSTRAINTS and RELATIONS	10-2
Create a Simple Triangular Plate Design	10-2
Fully Constrained Geometry	10-3
Starting Autodesk Inventor	10-3
Displaying existing constraints	10-4
Applying Geometric/Dimensional Constraints	10-6
Over-constraining and Driven Dimensions	10-10
Deleting existing constraints	10-11

Using the Auto Dimension command	10-12
Constraint Settings	10-16
Parametric relations	10-17
Dimensional Values and Dimensional Variables	10-19
Parametric Equations	10-20
Viewing the Established Parameters and Relations	10-21
Saving the Model File	10-22
Using the Measure Tools	10-23
Questions	10-27
Exercises	10-28

Chapter 11

Geometric Construction Tools Using Autodesk® Inventor®

Introduction	11-2
The Gasket Design	11-2
Modeling Strategy	11-3
Starting Autodesk Inventor	11-4
Creating a 2D Sketch	11-5
Editing the Sketch by Dragging the Sketched Entities	11-7
Adding Additional Constraints	11-9
Using the <i>Trim</i> and <i>Extend</i> Commands	11-10
The <i>Auto Dimension</i> command	11-12
Creating Fillets and Completing the Sketch	11-14
Fully Constrained Geometry	11-15
Profile Sketch	11-17
Redefining the Sketch and the Profile	11-18
Create an OFFSET Cut Feature	11-22
Questions	11-25
Exercises	11-26

Chapter 12

Parent/Child Relationships and the BORN Technique Using Autodesk® Inventor®

Introduction	12-2
The BORN Technique	12-2
The U-Bracket Design	12-3
Sketch plane setting	12-4
Applying the BORN Technique	12-5
Creating the 2-D Sketch of the Base feature	12-7
Create the First Extrude Feature	12-11
The Implied Parent/Child Relationships	12-12
Creating the Second Solid Feature	12-12

Creating the First Cut feature	12-16
Creating the Second Cut feature	12-17
Examining the Parent/Child Relationships	12-19
Modify a Parent Dimension	12-20
A Design Change	12-21
Feature Suppression	12-22
A different approach to the CENTER_DRILL feature	12-23
Suppress the Rect_Cut feature	12-25
Creating a Circular Cut Feature	12-26
A Flexible Design Approach	12-28
Creating and Editing Material Properties	12-29
Using the New Material	12-31
Questions	12-33
Exercises	12-34

Chapter 13

Part Drawings and 3D Annotations Using Autodesk® Inventor®

Drawings from Parts and Associative Functionality	13-2
Starting Autodesk Inventor	13-3
Drawing Mode – 2D Paper Space	13-3
Drawing Sheet Format	13-5
Using the Pre-defined Drawing Sheet Formats	13-7
Deleting, Activating, and Editing a Drawing Sheet	13-9
Adding a Base View	13-10
Creating Projected Views	13-11
Adjusting the View Scale	13-12
Repositioning Views	13-13
Displaying Feature Dimensions	13-14
Repositioning and Hiding Feature Dimensions	13-16
Adding Additional Dimensions – Reference Dimensions	13-18
Adding Center Marks and Center Lines	13-19
Completing the Drawing Sheet	13-22
Associative Functionality – Modifying Feature Dimensions	13-23
3D Annotations in Isometric Views	13-26
Questions	13-34
Exercises	13-35

Chapter 14

Symmetrical Features in Designs Using Autodesk® Inventor®

Introduction	14-2
A Revolved Design: PULLEY	14-2
Modeling Strategy – A Revolved Design	14-3

Starting Autodesk Inventor	14-4
Setup the display of the Sketch Plane	14-4
Creating the 2-D Sketch for the Revolved feature	14-5
Creating the Revolved Feature	14-9
Mirroring Features	14-10
Creating A Pattern Leader Using Construction Geometry	14-12
Circular Pattern	14-17
Examining the design parameters	14-19
Drawing Mode – Defining New Border and Title Block	14-19
Creating a Drawing Template	14-22
Creating Views	14-23
Retrieve Dimensions – Features option	14-26
Associative Functionality – A Design Change	14-28
Adding Center lines to the Pattern feature	14-30
Completing the Drawing	14-31
Saving the Pulley Design as an AutoCAD 2D Drawing	14-34
Opening the Saved Drawing in AutoCAD	14-36
Questions	14-38
Exercises	14-39

AutoCAD® and Autodesk® Inventor®

Chapter 15

Design Reuse Using AutoCAD® and Autodesk® Inventor®

Introduction	15-2
The Geneva Wheel Design	15-3
Internet Downloading the Geneva-Wheel DWG file	15-3
Opening AutoCAD DWG file in Inventor	15-4
Using the Measuring Tools	15-5
Opening the AutoCAD DWG Layout	15-7
2D Design Reuse	15-9
Completing the Imported Sketch	15-13
Creating the First Solid Feature	15-15
Creating a Mirrored Feature	15-16
Circular Pattern	15-17
Completing the Geneva Wheel design	15-18
Additional Parts	15-19
Export an Inventor 2D Sketch as AutoCAD Drawing	15-22
Design Reuse – Sketch Insert option	15-23
Questions	15-29
Exercises	15-30

Chapter 16

Assembly Modeling: Putting It All Together Using Autodesk[®] Inventor[®]

Introduction	16-2
Assembly Modeling Methodology	16-3
The Shaft Support Assembly	16-4
Additional <i>Parts</i>	16-4
(1) Collar	16-4
(2) Bearing	16-5
(3) Base-Plate	16-5
(4) Cap-Screw	16-6
Starting Autodesk Inventor	16-7
Placing the First Component	16-8
Placing the Second Component	16-9
Degrees of Freedom and Constraints	16-10
Assembly Constraints	16-11
Apply the First Assembly Constraint	16-13
Apply a Second Mate Assembly Constraint	16-15
Constrained Move	16-16
Apply a Flush Constraint	16-17
Placing the Third Component	16-19
Applying an Insert Constraint	16-19
Assemble the Cap-Screws	16-21
Exploded View of the Assembly	16-22
Editing the Components	16-24
Adaptive Design Approach	16-25
Delete and Re-apply Assembly Constraints	16-29
Setup a Drawing of the Assembly Model	16-31
Creating a Parts List	16-33
Editing the Parts List	16-34
Changing the Material Type	16-36
Completing the Assembly Drawing	16-38
Bill of Materials	16-39
(a) BOM from Parts List	16-39
(b) BOM from Assembly Model	16-40
Questions	16-41
Exercises	16-42

Chapter 17

Design Analysis Using Autodesk® Inventor® Stress Analysis Module

Introduction	17-2
Problem Statement	17-4
Preliminary Analysis	17-4
• Maximum Displacement	17-5
Finite Element Analysis Procedure	17-6
Create the <i>Autodesk Inventor</i> Part	17-7
Creating the 2D Sketch for the Plate	17-8
Assigning the Material Properties	17-10
Switch to the <i>Stress Analysis</i> Module	17-11
Create an FEA Simulation	17-12
Applying Constraints and Load	17-14
Creating a Mesh and Run the Solver	17-16
Refinement of the FEA Mesh – Global Element Size	17-18
Refinement of the FEA Mesh – Local Element Size	17-20
Comparison of Results	17-23
Creating an HTML Report	17-24
Geometric Considerations of Finite Elements	17-25
Questions	17-26
Exercises	17-27

Chapter 18

Assembly Model with FischerTechnik Robo TX Using Autodesk® Inventor®

Introduction	18-2
The Hand Dryer Assembly	18-2
Starting Autodesk Inventor	18-3
Placing the First Component	18-4
Placing the Second Component	18-5
Degrees of Freedom Display	18-6
Adjusting the Component's Orientation	18-7
Apply the Assembly Constraints	18-8
Assemble the next Component	18-11
Adding the 15mm Block to the Assembly	18-12
Assemble the Propeller Motor	18-19
Assemble the Propeller	18-22
Creating a Sub-Assembly	18-25
Adding the L-shape Sub-Assembly to the main assembly	18-27

Adding the Phototransistor and Lens-Tip-Light Components	18-29
Simple Animation with the Drive Constraint Command	18-30
Conclusion	18-34
Questions	18-35
Exercises	18-36

Index