



Inventor<sup>®</sup> Self-paced Learning Modules  
**Autodesk Inventor<sup>®</sup>**  
Module Content  
Content and Learning Outcomes

**Module Introduction 1 - Using Module to Learn Inventor**

1. Describe the modules philosophy and explain how to read and use them to master Inventor.

**Module Introduction 2 - Configuring Your Inventor Software**

2. Configure the Inventor software to prepare it for the Autodesk Inventor course.

**Module 1 - Projects**

3. Describe an Inventor project.
4. Create the Inventor project that will be used to organize and manage the files that are created while completing the Inventor Modules

**Module 2 - Inventor's User Interface**

5. Describe template files, part files, sketches and 3D solid models.
6. Describe and configure Inventor's user interface, its menus and use of the mouse.
7. Apply the commands NEW, SAVE and CLOSE to create a new part file using a template, name it, save it and close it..

**Module 3 - Viewing the Sketch and Model**

8. Describe how to change the viewing position of a 3D model or 2D sketch by zooming, panning and orbiting.
9. Apply the command OPEN to open existing files.
10. Apply the commands ZOOM, ORBIT, ROTATE, ZOOM ALL, PAN, HOME and ISOMETRIC or use the wheel on the mouse to change the viewing position of the 3D model or 2D sketch.

**Module 4 - Sketching Lines**

11. Describe the Cartesian Coordinate System, parametric solid modeling, the base sketch and geometrical constraints.
12. Describe snapping onto grids, lines, endpoints and midpoints.
13. Describe and apply the command PROJECT GEOMETRY to project the Center Point onto the base sketch.
14. Following Inventors 2D sketching rules, describe and apply the LINE command to draw the base sketch of simple solid models.

**Module 5 - Extruding - Part 1**

15. Describe dimensional constraints, linear dimensions, driving and driven dimensions.
16. Apply the GENERAL DIMENSION command to insert the necessary linear dimensions to fully constrain base sketches.
17. Describe and apply the EXTRUDE command to extrude base sketches to create the 3D base solid model.

**Module 6 - Competency Test No. 1**

18. Within a one hour time limit, complete a written exam and a lab exercise.

**Module 7 - Extruding - Part 2**

19. Describe how to construct a solid model using multiple sketches, linear dimensions plus joining and cutting extrusions.

20. Describe and apply the 2DSKETCH command to create 2D sketches on a solid model. Extrude the sketches to either join them to or cut them from the solid model.

**Module 8 - Multiview Drawings**

21. Describe multiview drawings, the glass box principle, the three standard views, object lines and hidden lines.

22. From a 3D pictorial of an object, draw a multiview drawing using the three standard views.

**Module 9 - Visualizing 3D Models**

23. Using a multiview drawing, visualize and sketch an isometric drawing of the object on isometric grid paper.

**Module 10 - 2D Sketching Planes**

24. Describe the three predefined 2D sketching planes and the view of the model that they are on.

25. Construct solid models by drawing the base sketch on either the front or right side instead of the default top view.

26. Describe a consumed and an unconsumed sketch.

**Module 11 - Competency Test No. 2**

27. Within a one hour time limit, complete a written exam and a lab exercise.

**Module 12 - Circles**

28. Describe construction objects and their use in a 2D sketch.

29. Describe drawing circles and offsets in a 2D sketch.

30. Apply the commands CENTER POINT CIRCLE and OFFSET to complete 2D sketches.

**Module 13 - Arcs**

31. Describe the geometry of an arc and how they are drawn in Inventor.

32. Describe how to snap to midpoints of lines.

33. Apply the command CENTER POINT ARC to draw arcs in 2D sketches.

**Module 14 - Revolving**

34. Describe a centerline object and explain how it inserted and used in a 2D Sketch.

35. Describe how a base sketch is revolved with and without the use of a centerline to create a solid model.

36. Apply the REVOLVE command to create a solid model from a base sketch.

**Module 15 - Fillets and Chamfers**

37. Describe fillets and chamfers and how they are used in model mode.
38. Apply the FILLET and CHAMFER commands to create fillets and chamfers on solid models.

**Module 16 - Competency Test No. 3**

39. Within a two hour time limit, complete a written exam and a lab exercises.

**Module 17 - Angles**

40. Describe drawing inclined lines, aligned and angular dimensions, loops, trimming and extending.
41. Apply the command GENERAL DIMENSIONS to insert aligned and angular dimensions on a sketch.
42. Apply the commands TRIM and EXTEND to trim and extend objects in a sketch.

**Module 18 - Editing Geometry**

43. Describe how to select objects using windows and crossing windows.
44. Describe and apply the commands THREE POINT ARC, TWO POINT RECTANGLE and MOVE.

**Module 19 - Work Features**

45. Describe and apply the commands POLYGON, TANGENT CIRCLE, and THREAD.
46. Describe work features including a work point, work axis, and work planes and explain how they are used in model construction.
47. Describe and apply the commands WORK POINT, WORK AXIS, and WORK PLANE.

**Module 20 - Modifying Solid Models**

48. Describe how to edit the dimensions and part features of an existing solid model using the Browser Bar to aid you.
49. Describe how to suppress features in the solid model.
50. Describe and apply the MEASURE command to measure lengths, loops, angles, or areas of a solid model in either two dimensions or three dimensions.
51. Describe how to set the material and change the color of faces of the solid model.
52. Describe how to obtain the physical properties of a solid model.

**Module 21 - Competency Test No. 4**

53. Within a two hour time limit, complete a written exam and the lab exercise.

**Module 22 - Assemblies**

54. Describe an assembly and explain the difference between a top-down and a bottom-up assembly.
55. Describe and apply the SLICE GRAPHICS command.
56. Describe the PLACE COMPONENT and PLACE CONSTRAINT commands and apply them to assemble a series of parts to create an assembled model.

**Module 23 - Presentation Files**

57. Describe a presentation file, an exploded assembly and an animation.
58. Describe and apply the commands CREATE VIEW, PRECISE VIEW ROTATION, TWEAK COMPONENT and ANIMATE to create, tweak, and play the animation of an exploded assembly.

**Module 24 - 2D Drawings - Part 1**

59. Describe 2D drawing files, drawing sheets and drawing sizes.
60. Describe and apply the BASE VIEW, PROJECTED VIEW, SECTION VIEW and NEW SHEET commands to create multiview and isometric views of a solid model on a drawing sheet.

**Module 25 - 2D Drawings - Part 2**

61. Describe basic dimensioning, standards and styles and centerlines.
62. Describe and apply the STYLE EDITOR command to copy and edit standard styles to create your own dimension and text styles.
63. Describe and apply the RETRIEVE DIMENSION, CENTERLINES and GENERAL DIMENSION commands to place model and drawing dimensions on a model view.
64. Describe how styles are exported out and imported into drawing files.

**Module 26 - Competency Test No.5**

65. Within a six hour time limit, complete a written exam and the lab exercise..