



Inventor® Self-paced Learning Modules

Autodesk Inventor® Advanced

Module Content

Content and Learning Outcomes

Module Introduction 1 - Using Modules to Learn Inventor

1. Describe the modules philosophy and explain how to read and use them to master Inventor.
2. List the files required to complete the Inventor Advanced Modules.

Module Introduction 2 - Configuring Your Inventor Software

3. Configure your Inventor software to prepare you for the Autodesk Inventor Advanced course.

Module 1 - Projects - Part 2

4. Describe and create the Inventor Advanced project that you will be using to organize and manage the files that you will be using and creating in this course.

Module 2 - Feature Patterns

5. Describe a rectangular and a circular feature pattern.
6. Apply the commands RECTANGULAR PATTERN and CIRCULAR PATTERN to speed the creation of solid models that contain rectangular and/or circular arrays.

Module 3 - Hole Features

7. Describe center points and drilled, counterbored, countersunk and threaded. parametric hole features.
8. Describe and apply the commands CENTER POINT and HOLES to locate and create drilled, counterbored, countersunk and threaded parametric hole features in a part.

Module 4 - Mirroring Features

9. Describe symmetry and mirroring.
10. Apply the command MIRROR to mirror symmetrical features to speed the creation of parts that have symmetrical features.

Module 5 - Pattern and Mirror Sketches

11. Apply the RECTANGULAR PATTERN, CIRCULAR PATTERN and MIRROR sketching commands in 2D sketches to create parts with rectangular patterns, circular patterns and mirrors for parts that have symmetrical features.

Module 6 - Competency Test No. 1

12. Within a one hour time limit, complete a written exam.

Module 7 - Rib and Web Features

13. Describe a rib and a web.
14. Apply the command RIB to an open profile sketch to create rib and web features on a part.

Module 8 - Coils and Threads

15. Describe a coil and a thread.
16. Apply the command COIL to create coils.
17. Apply the command COIL and SPLIT to create real life threads either cut into or added onto the part.

Module 9 - Shell

18. Describe a shell.
19. Apply the command SHELL to create a shell in a solid model and remove the faces as required.
20. Apply the RIB command to create a rib and web network.

Module 10 - Engraving and Embossing Text

21. Describe embossed and engraved text and apply the EMBOSS and TEXT commands to emboss and engrave text onto parts.
22. Describe a shared sketch and use them to speed the creation of parts.

Module 11 - Competency Test No. 2

23. Within a one hour time limit, complete a written exam.

Module 12 - Sweep Features

24. Describe a sweep and an open and closed profile and path.
25. Apply the command SWEEP to a closed profile and path sketch to create a part or solid features on a part.

Module 13 - Loft Features

26. Describe a loft.
27. Apply the command LOFT to blend or transition a solid that connects two or more planer sections or part faces of different shapes.

Module 14 - 3D Sketch

28. Describe the 3D sketch environment.
29. Apply the INCLUDE GEOMETRY and the 3D SKETCH commands to create 3D sketch paths.

Module 15 - Shading and Shadows

30. Describe the three shaded modes that the current part can be displayed.
31. Describe the two different camera display modes to view the current part.
32. Describe the three different shadow display modes to view the current part.

Module 16 - Competency Test No. 3

33. Within a one hour time limit, complete a written exam.

Module 17 - Sheet Metal - Part 1

34. Describe sheet metal parts, templates and the Sheet Metal environment.
35. Describe and apply the commands SHEET METAL STYLE, TWO POINT RECTANGLE, FACE and FLANGE to create simple sheet metal parts.

Module 18 - Sheet Metal - Part 2

36. Describe the terms reliefs, corners and hems and their use in sheet metal design.
37. Apply the BEND, CORNER SEAM and HEM commands to create sheet metal parts.

Module 19 - Sheet Metal - Part 3

38. Describe folding and flat pattern layout in sheet metal design.
39. Apply the FOLD command to create sheet metal parts starting with a flat pattern and folding it.
40. Apply the FLAT PATTERN command to create a flat pattern layout from a sheet metal part.

Module 20 - Sheet Metal - Part 4

41. Describe a cut.
42. Apply the CUT command to create a cut in a sheet metal part.
43. Apply the CORNER ROUND and CORNER CHAMFER commands to create a fillets and chamfers on a sheet metal part.

Module 21 - Competency Test No. 4

44. Within a one hour time limit, complete a written exam.

Module 22 - Weldments - Part 1

45. Describe a weldment and how they are created.
46. Describe how weldment assemblies are prepared for the welds to be applied.
47. Apply the commands FILLET, CHAMFER, EXTRUDE and SWEEP to prepare a weldment for future welding.

Module 23 - Weldments - Part 2

48. Describe fillet and grove welds.
49. Apply the commands FILLET WELD and GROVE WELD to create welds on an weldment assembly.

Module 24 - Weldments - Part 3

50. Describe how a weldment is machined after the welds have been created.

Module 25 - Engineer's Notebook

51. Describe the Engineer's Notebook and how it is used for in an Inventor file.
52. Create notes, comments and views in the Engineer's Notebook.

Module 26 - Competency Test No. 5

53. Within a one hour time limit, complete a written exam.