MICROCAD
TRAINING & CONSULTING

INV104: Inventor - CAM Milling Fundamentals

Course Length 2 Full Days, or 4 Sessions

Schedule

2 Full Days
9:00am - 4:00pm ET

Morning - 4 Sessions
9:00am - 12:00pm ET

Afternoon - 4 Sessions
1:00pm - 4:00pm ET

Evening - 4 Sessions 5:00pm - 8:00pm ET

Course Price $995 per person (open enroll)
*Dedicated 1-on-1 group sessions are recommended
(Max class size is 5 students)

Designed for

This course is designed for Inventor users who wish to expand their skills with CAM applications and will focus on generating 2D & 3D milling, and drilling toolpaths, once models are created. It will not cover how to use Inventor modeling tools.

Prerequisites

Users should have prior knowledge of 3D model creation/modification, as well as an understanding of the CNC milling process. It is recommended that students are familiar with a recent version of Inventor, as taught in the INV101: Inventor Fundamentals course, or equivalent.

What you get

Students will get classroom access to the software and Autodesk Authorized Training courseware (these can be purchased in addition to the training) and the knowledge to get started with Inventor and Inventor CAM.

Notes

The course length is a guideline. Course topics and duration may be modified by the instructor based upon the knowledge and skill level of the students.

All courses will be taught on the most current release, depending on availability of courseware.

Course Plan

The Inventor CAM Milling Fundamentals course focuses on instructing users on how to use the Inventor CAM add-on to create milling toolpaths. The course includes a review of the overall Inventor interface. Using hands-on practice examples, you will learn the key skills and knowledge required to take the 3D model, set it up in the CAM environment, and assign the 2D and 3D milling toolpaths needed to generate the CNC code for milling machines.

Topics Covered

- Navigate the Inventor software interface to locate and execute commands.
- Use the model orientation commands to pan, zoom, rotate, and review existing Inventor models.
- Assign visual styles to models.
- Locate, modify, and create tools in the Tool Library.
- Set up machining operations using Inventor CAM.
- Create 2D Milling and 3D Milling toolpaths using the Inventor CAM interface in support of 3 axis CNC Mills.
- Create Drilling toolpaths using the Inventor CAM interface in support of 3 axis CNC Mills.
- Use the Simulation option to visualize toolpaths.
- Import a tool library.
- Create a toolpath template.
- Post process an Inventor CAM setup to output the CNC code required to machine a part modeled in Inventor.

For more information, please contact our main office:

MicroCAD Training & Consulting
440 Arsenal Street
Watertown, MA 02472

Phone: 888-355-0081 Fax: 617-923-7006
mtcinfo@microcad3d.com
www.microcad3d.com