

SOLIDWORKS CAM Standard



Overview:

This course teaches you how to use the SOLIDWORKS CAM Standard software to generate, modify and post process 2.5 axis milling toolpaths used for the machining of SOLIDWORKS part files.



Duration:

InClass: 2.5 Days (Full Time) 8:30am- 4:30pm

Distance Learning: 7 Days (Part Time)



Pre-requisites:

- Complete / attend Introduction to Technical Drawing course or similar experience
- Complete the SOLIDWORKS Essentials
- Computer literacy skills
- Understanding & reading technical drawings

SOLIDWORKS CAM Basics and user interface

- What is SOLIDWORKS CAM
- SOLIDWORKS CAM user interface
- Process overview
- Case study
- Feature Tree

Automatic Feature Recognition (AFR) and Operation Modification

- Working with features, operations and toolpaths
- Automatic Feature Recognition
- Modifying operations
- Modifying parameters

Interactive Feature Recognition (IFR)

- Interactive Feature Creation
- AFR and IFR feature creation
- IFR 2.5 Axis feature and operation creation
- IFR 2.5 axis feature selection

Merging features and operations

- Machining similar features
- Combine operations
- Combine selected operations



Avoid and contain areas

- Adding avoid and contain areas
- Case study

Pattern features and mirror toolpaths

- Patterning
- Create linear, circular and sketch driven patterns
- Mirror toolpaths
- Case study

Advanced features and operations

- Advanced feature creation
- Engrave feature creation
- Curve feature creation
- Multi-stepped hole machining
- Tap and thread mill
- Corner round and chamfer machining
- Multi surface feature creation

Customizing the Technology Database

- SOLIDWORKS CAM Technology Database (TechDB)
- User defined tool creation
- TechDB add machine
- TechDB add tool
- TechDB Create and apply strategy

