

SOLIDWORKS CAM Standard

Length: 2.5 Days

Prerequisites: Experience with the SOLIDWORKS design software. Experience with the Windows® operating system.

Description: This course teaches 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.

Course Syllabus

Introduction

- About This Course
- Windows
- Use of Color
- More SOLIDWORKS Training Resources

Lesson 1 - SOLIDWORKS CAM Basics and User Interface

- What is SOLIDWORKS CAM?
- SOLIDWORKS CAM Add-in
- SOLIDWORKS CAM User Interface
- Process Overview

Lesson 2 - Automatic Feature Recognition (AFR) and Operation Modification

- Working with Features, Operations and Toolpaths
- Automatic Feature Recognition
- Feature Strategy

Lesson 3 - Interactive Feature Recognition (IFR)

- Interactive Feature Creation

Lesson 4 - Interactive Operations

- Interactive 2.5 Axis Mill Operations
- Save Operation Plan

Lesson 5 - Merging Features and Operations

- Machining Similar Features
- Create Group

- Combine Operations

Lesson 6 - Avoid and Contain Areas

- Adding Avoid and Contain Areas

Lesson 7 - Pattern Features and Mirror Toolpaths

- Patterning

Lesson 8 – Advanced Features and Operations

- Advanced Feature Creation

Lesson 9 – Customizing the Technology Database

- SOLIDWORKS CAM Technology Database (TechDB)
- User Designed Tool

Appendix A – Review Section

- Waterjet, Plasma and Laser Machining

Appendix B – Tolerance Based Machining

- Tolerance Based Machining Overview





**CADIMENSIONS IS A SOLIDWORKS
CERTIFIED TRAINING CENTER**

CADIMENSIONS TRAINING CATALOG

