Advance Lathe Mill Multiaxis Design & Toolpaths using
Mastercam

Course Curriculum (Duration: 100 Hrs.)

Chapter 1: Before you start using Mastercam
   a. Coordinate Systemes
   b. Conversions, Measurements, and Tools
   c. Machining Processes
   d. Lathe, Drilling, Reaming, Tapping and Milling
   e. Cutting Tools and Tool Holders
   f. Cutting Speeds, Feeds, and Revolutions per Minute (RPM)
   g. Chip Formation, Load, and Material Removal Rates
   h. Work Holding and Setup
   i. Introduction to CNC Part Programming

Chapter 2: Basics of CAD/CAM & Introduction to Mastercam
   a. Introduction to CAD/CAM
   b. Advantages of CAD/CAM over conventional methods
   c. Mastercam Software Installation
   d. Mastercam Modules
   e. GUI
   f. Getting Help
   g. File Management
   h. File Conversions

Chapter 3: Lathe Design
   a. Set the plane to diameter
   b. Create lines on diameter
   c. Create a groove
   d. Create a revolved solid
   e. Create dimensions
   f. Use radius blends
   g. Create chamfers
   h. Assignments

Chapter 4: Lathe Setup
   a. Select a Lathe machine definition
   b. Set the Lathe machine group properties (tools, materials, stock, chuck, jaws, etc.)
   c. Set up a Lathe toolbar configuration
   d. Explore the Lathe toolbar icons
   e. Face, rough, finish, groove, and thread a part
   f. Cut the part off
   g. Verify all toolpaths
   h. Post process the operations to create the NC program (G-code)
   i. Assignments

Chapter 5: Lathe Toolpaths
   a. Review Mastercam chaining as used for lathe machining
   b. Set up stock and chuck jaws
   c. Create geometry as necessary to program operations on a solid model
   d. Create a custom tool, tool holder, and tool library Flip stock
   e. Create and use a template and merge part geometry with a template
   f. Apply both standard and quick lathe toolpath operations
   g. Assignments
   h. Lathe Applications
Chapter 6: Milling Design
   a. Set the plane to diameter
   b. Create lines on diameter
   c. Create a groove
   d. Create a revolved solid
   e. Create dimensions
   f. Use radius blends
   g. Create chamfers
   h. 3D Modeling of various Parts & Assemblies
   i. Assignments

Chapter 7: Mill Toolpaths
   a. Chaining Overview
   b. Toolpaths Overview
   c. Alt Mount
   d. Power Mount
   e. Tool Manager
   f. Autowinder
   g. Dash
   h. 30 Degree Clamp
   i. Receiver
   j. Assignments

Chapter 8: Mill Applications
   a. Programming the brake for a limited production run.
   b. Programming the brake for a big production run with dedicated fixturing.
   c. Assignments

Chapter 9: Multiaxis (Curve Drill and Circle Mill)
   a. 4th Axis Rotary Machining:
      Axis substitution
      • Creating geometry based on a cylinder
      • Marking at 90, 30, and 45 degree positions
      • Drill point on angle
      • Adding the 4th axis mill
      • Backplot and verify the toolpath

      Axis Positioning
      • Loading a Parasolid model
      • Part Fixture
      • Drill rotary
      • Multiple offsets
      • Positioning

Chapter 10: Rotary Surface Machining
   a. Axial machining
   b. Radial machining
   c. Axis substitution
   d. Axis Positioning
   e. Shifter
   f. Horizontal Overview

Chapter 11: 5 Axis Toolpath
   a. Overview
   b. Toolpath vector from a Point
   c. Toolpath vector from a Line
   d. Toolpath vector from a Chain
   e. Toolpath vector from a Plane
   f. Toolpath vector from a Surface
   g. Toolpath vector from Curves and Lines
   h. Trimming a shield
   i. Drilling and Circle Mill
   j. Final Project

www.ifsacademy.org