Course: NX/Solid Edge CAM Pro Turning Fundamentals

Duration: 2 days

Version: NX2008 SE CAM Pro 2022

At Course Completion

After successfully completing this course, you should be able to perform the following activities in NX/Solid Edge CAM Pro:

- Define part and blank geometry for lathe or turning operations.
- Creating lathe or turning facing, centerline drilling, roughing, finishing, and grooving operations.
- Place and control avoidance and containment parameters for your typical lathe or turning operations.

Prerequisites

Before taking this course, students must have.

- **Completed the NX Fundamentals course or the Solid Edge CAM Express Fundamentals course.**
- A good understanding of the basic manufacturing environment in NX/SE CAM Pro, including the GUI, Operation Navigator, and Manufacturing Assemblies.
- A good understanding of Parent Groups in the NX/SE CAM Pro application.
- Mechanical Manufacturing Experience
- Windows Experience

Course Content

Course consists of;

- 8 Video Lectures (PowerPoint's to support the instructors lecture).
- 326 Instructor lead video demonstrations.
- 22 practical activities to reinforce the lessons.
- Solution videos for each activity.

Topics:

Day 1

Lesson 1: Define part and blank geometry

Upon completion of this lesson, you will be able to:

- Define Part and blank boundaries by selecting their solid bodies.
- Define geometry objects.
- Define cut region and containment within an operation.
- Define blank geometry as tube stock.

Lesson 2: Retrieve and create tools

Upon completion of this lesson, you will be able to:

- Retrieve tools from the library.
- Create new tools.

Lesson 3: Face operations

Upon completion of this lesson, you will be able to:

- Machine the face of the part.
- Display the cut region.
- Define the limit of the cut region using an axial trim plane.
- Define avoidance parameters.
- Define additional face stock.
- Generate the tool path and visualize the in-process workpiece.

Lesson 4: Common options

Upon completion of this lesson, you will be able to:

- Define rough stock.
- Define grind stock.
- Define part boundary parameters.
- Define feed rates.
- Define From, Return, and Start points.
- Define Engages and Retracts.

Lesson 5: Centerline operations

Upon completion of this lesson, you will be able to:

- Create centerline spot drilling and drilling operations.
- Define drilling geometry both automatically and manually.
- Control chip removal by defining variable increments.

Day 2

Lesson 6: Rough operations

Upon completion of this lesson, you will be able to:

- •Create a Rough Turn operation.
- • Change the Cut Strategy.
- • Identify the appropriate Cut Depths.
- • Add and remove Cleanup passes.
- • Create additional Profiling passes (spring pass).
- • Add a Local Return Point to the Spring pass.
- • Add Post Commands to Local Return moves.
- • Apply Single and Multiple Ramping Cut Strategies.
- • Edit Part Boundary parameters from within an operation.
- • Use Containment to limit the cut area.
- • Use the different Reversal Mode options to remove material from the valleys of the part boundary found in the cut region.

Lesson 7: Finish operations

Upon completion of this lesson, you will be able to:

- • Create a Finish operation to cut the part OD.
- • Create a Finish operation to cut the part ID.
- • Use the Corner Control options to finish the convex corners.
- • Use the Fillet options to finish the concave corners.

Lesson 8: Groove operations

Upon completion of this lesson, you will be able to:

- • Identify the different groove methods.
- • Identify the options unique to grooving.
- • Specify Stepover and Cut Depth.
- • Manually specify the cut region.
- • Create GROOVE and GROOVE_FACE operations.

Note: The number of lessons covered on any given day could vary due to the progress of the student.