

Creating Toolpaths for a CNC Lathe

This course provides information on the setup and creation of toolpaths for a CNC lathe.

Overview

CNC machines come in an almost endless array of configurations for various applications. So far, we have only talked about CNC Mills. More specifically vertical milling centres. In this course we turn our attention to the CNC Lathe. We identify the difference in a lathe's coordinate system, tools and how to create lathe specific toolpaths.

Course Learning Objectives

- Explore toolpaths for CNC lathes.
- Identify a lathe coordinate system.
- Choose lathe specific tools.
- Create turning toolpaths.

Chapter 1: Setting Up and Cutting a Profile

In this chapter you will learn how to set up a CNC Lathe coordinate system, stock and begin cutting some basic geometry. The fundamental difference in how a lathe coordinate system is set and how the stock is held are critical to the success of turning toolpaths. Once you understand these concepts, we can begin exploring more complicated geometry.

After completing this chapter, you will be able to:

- Identify a lathe coordinate system.
- Practice setting up a CAM program for CNC Turning.
- Create a lathe toolpath

Lesson 1: Creating a CAM setup for Turning

The process of creating a CAM setup has many of the same options for a lathe as it does a mill. However, we must completely understand some basic parameters that apply only to a lathe. In this lesson you will focus on understanding coordinate system orientation and location, stock holding and stock creation.

After completing this lesson, you will be able to:

- Recognise unique setup options for a lathe.
- Practice creating a CAM setup.
- Identify a lathe coordinate system.

L1-01 – CNC Lathe coordinate systems 6:00

- Modify preview technology.
- Create a CAM turning setup.



L1-02 – Setting up stock for a CNC lathe 3:15

- Modify default stock parameters.
- Create a program number and comment.

L1-03 – Identifying on screen stock visualization 3:00

- Demonstrate how to show and hide stock.
- Identify right-click menu options.

Lesson 2: Understanding lathe tooling

Lathe tools are fundamentally different than Mill tools. While it is true that some mill tools utilize inserts and some lathes have live tooling that can use endmills, there is a fundamental difference in lathe tools that we need to identify. In this lesson you will look at various tools to understand their shapes, designations and locations in the machine. Finally, you will identify how to create custom tools in Fusion 360.

After completing this lesson, you will be able to:

- Identify lathe specific tooling.
- Recognize tool holding devices.
- Create a digital version of a lathe tool.

L2-01 – Identifying specific lathe tools 5:30

- Demonstrate how to navigate a tool library.
- Identify lathe specific tools.

L2-02 – Lathe tool location and orientation 6:30

- Identify options for tool orientation.
- Review tool parameters.

L2-03 – Creating a lathe tool in Fusion 360 6:00

- Create a threading tool in a tool library.
 - Demonstrate how to adjust tool creation parameters.

Lesson 3: Creating our first lathe toolpath

The process of creating a toolpath for turning is the same as creating a toolpath for milling, but there are different options and controls we need to understand. In this lesson you will use the toolpath “turning profile” to explore everything from programming a basic roughing pass to controlling a tool turret B axis and finishing the inside of a part. Turning profile is a versatile toolpath that will be our first step in cutting revolved geometry.

After completing this lesson, you will be able to:

- Recall various toolpath options for turning.
- Identify B axis tool control.
- Create a turning profile toolpath.

L3-01 – Creating a basic external roughing turning toolpath 10:00

- Create a profile toolpath.

- Identify toolpath parameters.

L3-02 – Exploring external finishing options for turning profile 5:30

- Modify the tool used for a profile toolpath.
- Modify toolpath parameters.
- Create a facing toolpath.
- Use rest machining on a profile toolpath.

L3-03 – Using turning profile on internal geometry 6:45

- Create an internal turning profile toolpath.
- Identify toolpath warning.
- Modify tool orientation in a toolpath.

L3-04 – Getting a profile toolpath to drop into a radial groove 3:45

- Modify profile toolpath settings to rough a radial groove.
- Identify toolpath options.

L3-05 – Profile leads and transitions 5:30

- Modify profile toolpath leads and transition settings.
- Modify the safe z distance in a setup.

Chapter 2: Internal and External Grooving Toolpaths

In Chapter 2 you will dive in deeper to turning toolpath functionality by exploring internal and external grooving. With Fusion 360 we can identify and cut a single position groove as well as handle roughing and finishing multiple grooves in multiple orientations.

After completing this chapter, you will be able to:

- Identify groove geometry.
- Create roughing and finishing groove toolpaths

Lesson 1: Creating grooves for external threading

Cutting threads on a lathe can be very rewarding when they work. A common practice when cutting threads manually or on a CNC machine is to create a groove at the end so the tool can stop moving without continuing to cut. In this lesson we are going to explore how to cut that geometry with the single groove toolpath.

After completing this lesson, you will be able to:

- Create a single groove toolpath.

L1-01 – Using the single groove toolpath in Fusion 360 7:00

- Create a single groove toolpath.
- Modify tool parameters.
- Use simulate to verify a toolpath.

Lesson 2: Roughing multiple grooves

The grooving toolpath in Fusion 360 has amazing functionality with relatively little guidance from the user. In this lesson you will identify and use this tool to rough multiple external and internal grooves on the same part.

After completing this lesson, you will be able to:

- Explain how to use turning groove for internal and external geometry.
- Create a roughing toolpath for external grooves.

L2-01 – Turning Groove for external roughing 6:15

- Create a groove toolpath for roughing.
- Create a custom grooving tool.

L2-02 – Turning Groove for internal roughing 7:30

- Create an internal profile toolpath.
- Create an internal groove toolpath.
- Modify parameters to fix toolpath warnings.

Lesson 3: Finishing multiple grooves

The grooving toolpath in Fusion 360 can both rough and finish geometry. We are learning this in 2 steps to identify the various options, but then we explore how to apply this efficiently to our geometry. In this lesson you will identify how to duplicate and edit toolpaths as well as finish groove geometry both internally and externally.

After completing this lesson, you will be able to:

- Explain roughing vs finishing.
- Create a duplicate toolpath.
- Modify toolpath parameters.

L3-01 – Duplicated toolpaths to reuse parameters and selections 7:00

- Demonstrate how to duplicate a toolpath.
- Identify the difference between derived and duplicated operations.
- Show how to rename an operation.

L3-02 – Creating an operation to rough and finish 3:15

- Modify toolpath parameters to rough and finish.
- Demonstrate how to suppress and delete an operation.

L3-03 – Finishing internal grooves 3:30

- Modify toolpath parameters to rough and finish.
- Modify tool parameters.
- Use Simulate to validate toolpaths.

L3-04 – Addressing collisions in a file 2:00

- Modify toolpath parameters to avoid collisions.

Lesson 4: Finishing a part with rounded grooves

Grooves on a part are traditionally thought of as having square corners for things like o-rings or circlips, but that is not a requirement. In this lesson you will look at parts that have grooves with tapered walls and rounded corners to explore how the approach might change.

After completing this lesson, you will be able to:

- Explain grooving for rounded geometry.
- Create a parting toolpath.
- Create machine-specific NC code.

L4-01 – Creating a CAM setup 3:45

- Create a setup for turning.
- Practice setting up stock and a coordinate system.

L4-02 – Roughing and finishing a part 8:15

- Create a facing toolpath.
- Create roughing and finishing toolpaths.
- Use Simulate to validate material removal.

L4-03 – Parting and simulating 7:00

- Modify tool parameters.
- Create a parting toolpath.
- Modify profile roughing toolpath extensions.

L4-04 – Outputting g-code 6:00

- Create machine-specific NC code.

Chapter 3: Threading and Chamfering for Turning

In this chapter you will turn our attention to more finishing toolpaths to break edges and create threads. There are a few different ways to create threads on turned parts and you will start by looking at cutting external threads, tapping internal threads along the axis of revolution as well as identifying live turning tools to drill and tap on a C axis.

After completing this chapter, you will be able to:

- Identify the need for C axis tooling.
- Create chamfer toolpaths.
- Practice cutting threads on turned parts

Lesson 1: Breaking external edges with chamfer toolpaths

In the instance where a part is designed, and your job is to produce code to cut physical parts you might have some prep work to do. Some designs have sharp corners while others have been deburred digitally. In this lesson you will explore some direct modelling tools to add and remove chamfers and how we can use turning toolpaths to break sharp edges.

After completing this lesson, you will be able to:

- Demonstrate how to add and remove chamfers from a model.
- Create a chamfer toolpath.

L1-01 – Removing or adding chamfers to a model 3:45

- Use Capture Design History.
- Use Delete to remove a feature.
- Create a chamfer on a model.

L1-02 – Cutting chamfers with turning toolpaths 6:15

- Create a chamfer toolpath.
- Modify chamfer toolpath parameters.

L1-03 – Roughing and finishing a simple part 6:30

- Create a facing toolpath.
- Create a profile for roughing and finishing.
- Create a single groove toolpath.

Lesson 2: Cutting external threads

Cutting external threads with Fusion 360 is a straightforward process, but we do need to have some basic information handy. In this lesson you will explore how to remove physical threads from a model as well as define thread depth and pitch to accurately cut threads.

After completing this lesson, you will be able to:

- Apply direct modelling techniques to remove threads
- Create an external threading toolpath.

L2-01 – Model prep to remove threads 5:00

- Create a sketch and extrude to remove a modelled thread.
- Use direct modelling tools to remove geometry.

L2-02 – Create an external thread cutting toolpath 8:30

- Create an external threading toolpath.
- Analyse a toolpath simulation.
- Modify geometry using press pull.
- Modify tool geometry.

L2-03 – Create more external thread cutting toolpaths 2:45

- Create an external threading toolpath.
- Analyse a toolpath simulation.

Lesson 3: Drilling and Tapping internal threads

Internal threads can also be cut but often they are better suited to a drilling and tapping operation. In this lesson you will learn how to drill and tap holes that are on the rotation axis of our part.

After completing this lesson, you will be able to:

- Create drilling and tapping toolpaths.

L3-01 – Drilling on the axis of revolution 6:00

- Create a peck drilling toolpath.
- Identify drilling cycle options.

L3-02 – Creating a rigid tapping toolpath 4:30

- Create a tapping cycle toolpath.

Lesson 4: Drilling and Milling off the axis of revolution

We have already talked about B axis control on some machines that allows a lathe tool to rotate in order to better cut geometry. In this lesson you will focus on another common function of CNC lathes, C axis cutting. C axis control and the addition of live tooling allows us to mill pockets and drill and tap holes that are on the face of a revolved part.

After completing this lesson, you will be able to:

- Recognize C axis operations.
- Create an off axis drilling and tapping toolpath.

L4-01 – Creating a C-axis drilling and tapping toolpath 7:00

- Create a drilling toolpath off the axis of revolution.
- Create a C-axis tapping toolpath.

L4-02 – Creating a C axis milling toolpath with live tooling 5:45

- Create a 2D adaptive toolpath with live tooling.

L4-03 – Creating a toolpath pattern 2:15

- Create a circular pattern of a toolpath

L4-04 – How to post code with C axis live tooling 6:00

- Use Post Process to create code.
- Demonstrate how to find and download new post configurations.

Chapter 4: CNC Lathe Stock Handling

In this chapter you will learn more about various options that CNC Lathes can have. You will learn about parting to cut geometry away from the stock as well as some various options for stock transfer using multiple chucks. CNC lathes often can move another chuck in to grab the part, cut it away from stock and then machine the back side.

After completing this chapter, you will be able to:

- Explain part handling.
- Create a parting toolpath.
- Summarize stock transfer

Lesson 1: Programming a part that needs stock transfers

Stock transfer can be done manually or with a machine that supports sub-spindle operations. Before we explore this lets practice some of the toolpaths we have learned and program a part that needs stock transfer.

After completing this lesson, you will be able to:

- Explain the need for stock transfer.
- Create toolpaths to rough and finish a part from a single direction.
- Analyse and identify collisions.

L1-01 – Roughing and finishing the outside of an adapter 7:30

- Explain the process of importing a tool library.
- Create a face toolpath.
- Create a profile toolpath.
- Create a single groove toolpath.
- Modify toolpath parameters to avoid stock collisions.

L1-02 – Roughing and finishing the inside of an adapter 8:45

- Create drilling and boring toolpaths.
- Modify radii parameters to avoid crossing the rotation axis.

Lesson 2: Understanding Stock Transfers

Some CNC Lathes come with multiple spindles that allow you to move stock and parts around. This functionality is known as stock transfer. In Fusion 360 we have 2 operations specific to stock transfer that allow us to move and access both sides of a part. In this lesson you will learn how to implement stock transfer.

After completing this lesson, you will be able to:

- Explain stock transfer.
- Create a stock transfer operation.

L2-01 – Manual stock flips 4:30

- Create a new setup.
- Create a multi-pass facing operation.

L2-02 – Using multiple lathe spindles 4:45

- Create a secondary spindle chuck operation.
- Create a secondary spindle return operation.

Lesson 3: Parting and Stock Transfer

When a lathe part has been completely cut it is time to remove it from the remaining stock. In Fusion 360 this can mean dropping into the base of the lathe, using a part catcher or invoking a stock transfer. Additionally, the parting toolpath allows us to chamfer and fillet the back edge of a part helping us create something that is complete without additional toolpaths. In this lesson you will create a parting operation and explore all its options.

After completing this lesson, you will be able to:

- Discover various options of a turning part operation.
- Create a parting toolpath with stock transfer.

L3-01 – Parting without edge clean up 2:30

- Create a parting toolpath.
- Apply the part catcher option.

L3-02 – Parting with edge clean up 3:00

- Modify a parting toolpath with the edge break option.
- Modify radii to adjust the tool start location.

L3-03 – Parting with stock transfer 3:30

- Modify a parting toolpath to include a stock transfer.
- Create stock transfer and chuck operations.

Lesson 4: Finishing the back of a part

After a part has been transferred the back side still needs to be cut. In this lesson you will continue learning about options inside of a profile toolpath to utilize boring bars in tight areas.

After completing this lesson, you will be able to:

- Discover various options of a turning part operation.
- Create a profile toolpath at a 45-degree angle.

L4-01 – Using profile to finish a part 11:45

- Modify the tool angle in a profile operation.
- Use Simulate to validate a toolpath.

Lesson 5: Programming a part that uses a lathe and a mill

We have explored a situation where a turned part needs to be flipped and machined from 2 directions, but what about using multiple machines? In this lesson you will take a look at a simple part that has turning and milling operations and some workflow options to help us utilize the shape of our part before and after each setup.

After completing this lesson, you will be able to:

- Discover various options for defining stock.
- Create a body to represent stock.
- Review multiple toolpaths that utilise rest machining.

L5-01 – Creating basic turning toolpaths 5:00

- Create a new setup.
- Create roughing and finishing toolpaths.
- Create a comment in the design.

L5-02 – Preparing for a milling setup 6:00

- Create a new milling setup.
- Use direct modelling tools to create a stock body.
- Use a body for stock in a new setup.

L5-03 – Create milling toolpaths 7:00

- Create contour toolpaths for open pockets.
- Use Simulate to ensure all stock is removed.

L5-04 – Using stock for rest machining 8:30

- Analyse multiple toolpath options for rest machining.
- Use save options in simulate to create an STL file.
- Modify the scale of an STL file.
- Create a distributed design.

Chapter 5: Course Assessment and Peer Review Assignment

Challenge Assignment

4x Challenge Assignments

Assignment 1: Create a CAM setup from a supplied file. Create a new turning setup. Stock should be oversized and include enough material on the back side to be held in a chuck. Create roughing and finishing toolpaths for the outside and inside of the part and then post the NC code.

Assignment 2: as above but with a different part

Assignment 3: as above but a different part with threading and chamfer techniques.

Assignment 4: as 1 & 2 above but with a different part and chamfer, parting with edge clean up.