

Introduction to CAD, CAM & CNC Machining

This course provides an introductory look into the 3D modelling, toolpath programming and practical use of CNC machining.

Overview

This course introduces you to the foundational knowledge in computer aided design, manufacture and the practical use of CNC machines. In this course we begin with the basics in Fusion 360 CAD by learning how to properly sketch and model 3D parts. Before we program any toolpaths, we will explore CNC machining basics to ensure we have the ground level foundational knowledge needed to effectively define toolpaths. Finally, we explore the basics of setting up a CAM program and defining toolpaths to cut simple geometry. This is the same basic process that gets repeated for the design and manufacture of any part and is a critical step in learning and understanding the process.

Course Learning Objectives

- Explain the CAD design process as applied to prismatic parts.
- Summarize the workflow of digital manufacturing.
- Demonstrate knowledge and skills in basic Fusion 360 CAM
Recall foundational knowledge of practical CNC machining.

Chapter 1: (Fusion 360 Foundational Design Concepts)

Lesson 1: Exploring the Fusion 360 Interface

In this lesson, you will learn how to navigate Fusion 360 and identify various access points to tools and technology.

After completing this lesson, you will be able to:

- Identify areas of the Fusion 360 UI

L1-01 – The Fusion 360 UI 9:30

- Identify areas of the fusion 360 UI
- Use the right click marking menu

L1-02 – What is a workspace 2:30

- Demonstrate how to change a workspace

L1-03 – Fusion 360 on the web 9:30

- Create a project and folder
- Show how to upload a design
- Use Browser CAD

Lesson 2: The basics of sketching in Fusion 360

Fusion 360 is a powerful CAD/CAM and CAE software. Before we can even think about using some of Fusion's advanced tools, we need to understand how to make geometry. If you are brand new to Fusion 360 or coming from another CAD software, you will learn the best practices for creating sketch geometry as the basis for any design.

After completing this lesson, you will be able to:

- Create a sketch.
- Apply dimensions and constraints to sketches.
- Create a reference plane. Modify a sketch.

L2-01 – How to create a sketch 10:00

- Create a sketch
- Use Constraints and Dimensions to define a sketch

L2-02 – Editing a sketch 4:30

- Modify a sketch dimension
- Use dimensions and constraints to define a sketch

L2-03 – Sketch Parameters 3:00

- Use sketch parameters

L2-04 – Toggling the visibility of sketches and their dimensions

- Demonstrate how to hide a sketch
- Show sketch dimensions

Lesson 3: The basics of geometry creation in Fusion 360

With a solid foundation in how to sketch with Fusion 360 we can take that exciting step of turning our work into a solid model. There are many ways to utilize sketch geometry to create a solid like extrude, revolve and loft. In this lesson you will explore extrudes and revolves to create common geometry for CNC mills and lathes.

After completing this lesson, you will be able to:

- Demonstrate how to turn a sketch into a solid.
- Use extrude and revolve.
- Identify options during feature creation.

L3-01 – Extruding a sketch into 3D 2:30

- Use Extrude
- Change a parameter

L3-02 – Spinning a sketch to make a solid 3:00

- Use Revolve



L3-03 – Exploring surface creation 4:30

- Use direct modelling tools to remove geometry
- Create a revolved surface
- Use Patch

Lesson 4: The basics of geometry modification in Fusion 360

We know how to sketch and how to turn that work into a solid, but there is another aspect of modelling we need to explore, modification. After a solid is created we can modify it further with tools like fillet, draft and shell. In this lesson you will explore some of the common tools we need to produce models for production.

After completing this lesson, you will be able to:

- Modify geometry with features.
- Use extrude to remove material.
- Demonstrate how to fillet a sharp edge.

L4-01 – Understanding modification and direct editing tools 5:30

- Modify hole locations
- Use Delete to remove chamfers
- Use Press Pull to change a fillet radius

L4-02 – Using draft and shell features 4:00

- Use Draft
- Use Shell
- Modify a body by removing a fillet

Quizzes

1 question per each video (in video questions)

Lesson 5: How do we properly use bodies and components

Another piece of the modelling puzzle is the classification and use of bodies vs. components. The approach to this can be difficult to understand at first so you will start by learning how to make or convert between bodies and components. In this lesson you will explore the creation and usage of bodies and components as they apply to digital design and manufacturing.

After completing this lesson, you will be able to:

- Identify a body and a component.
- Explain the benefits of using a component.
- Use component colour cycling.

L5-01 – Creating bodies and components 6:00

- Create a body and component
- Use component colour cycling

- Demonstrate how to convert a body to a component

Lesson 6: Creating 2D drawings from a 3D model

Being able to read and create details drawings is an important part of design and manufacturing. This might mean creating a model from a detailed drawing to handle an engineering change order or creating the detailed drawing to pass down the line for production or quoting. Detail drawings contain critical information about geometric references, surface finishes and datum locations. In this lesson you will learn how to create detailed drawings in Fusion 360.

After completing this lesson, you will be able to:

- Identify aspects of the drawing workspace.
- Explain the model to drawing workflow.
- Use GD&T symbols on a drawing.

L6-01 – Creating a simple detailed drawing 6:30

- Use the drawing workspace to create a simple drawing
- Identify areas of the drawing workspace
- Use Dimensions

L6-02 – How to add symbols and notes 5:30

- Demonstrate how to add notes
- Use Control Frame to add a GD&T note

L6-03 – Drawings for Assemblies 6:30

- Demonstrate how to create a parts list
- Modify component properties

L6-04 – Customizing a drawing 6:30

- Demonstrate how to make a custom title block
- Use a custom title block for a new drawing

Practice Exercises

3x Practice Exercises

- 1) Create a body by extruding a sketch.
- 2) Practice: Use direct modelling tools to remove and modify geometry.
- 3) Modify a design and create a detailed drawing

Challenge Assignments

1 Challenge Assignment – Foundation Design

Chapter 2: CNC and Machining Basics

Lesson 8: What is Stock

If you are not familiar with machining, then you might not know exactly what stock is. Stock is the material we start with, and we need to define this in CAM so we can visualize and program where to remove material. In this lesson we are going to talk about some basics for standard stock sizes and how that can potentially drive the size of a final design.

After completing this lesson, you will be able to:

- Identify standard stock sizes.
- Explain the benefits of stock size vs. finished part size.

L1-01 – Common materials

- Identify different types of material.
- Define stock.
- Summarize differences in material shapes.

Lesson 9: Figuring out how to hold stock

It is easy to dive right into CAM and begin creating toolpaths but understanding how to hold the stock as well as avoid collisions between work holding and the tool is imperative. In this lesson we will look at some common methods for holding material for CNC milling.

After completing this lesson, you will be able to:

- Identify types of work holding devices.
- Explain how to install and remove stock from a vice.
- Describe the reason to use parallels in a vice.

L2-01 – Using T slots and clamps

- Identify methods of holding stock on a machine.
- Compare various methods for holding stock.

L2-02 – Basics of Vices

- Describe the various parts of a vice.
- Identify the height of stock and use of parallels.

L2-03 – Vice Squareness

- Describe the importance of vice squareness.
- Identify the tools needed to properly align a vice to a machine.

L2-04 – CNC Machine Safety

- Recall the tools required to use and install a vice.
- Review safe work practices prior to machine operation.

Lesson 10: Work and Tool Coordinate Systems

Coordinate systems in CNC milling and turning are fundamental topics that must be understood before programming any kind of tool motion. Knowing which axis the tool is in, the direction of travel and location of zero must be known in order to control the tool motion accurately. Let's take a look at the coordinate systems on a standard vertical mill as well as a +Z+C lathe.

After completing this lesson, you will be able to:

- Identify typical coordinate systems.
- Explain tool motion related to a coordinate system.

L3-01 – How typical CNC mills move

- Discover the work coordinate system of a mill.
- Identify axis orientation on a CNC mill.
- Explain positive vs. negative directions.

L3-02 – How typical CNC lathes move

- Discover the work coordinate system of a lathe.
- Identify the differences between a lathe and mill.
- Explain tool direction.

Lesson 11: Types of tools and their motions

It might seem like a minor thing, but there are so many tools on the market today that we need to understand and identify several parameters that will affect the downstream CAM toolpaths. We need to know the size and shape of a tool, if its centre cutting, general parameters about how fast it can spin and how much material it can remove. If it is used for roughing or finishing and if it needs coolant just to name a few. In addition to the parameters of a tool we also need to consider factors about the direction the tool is moving relative to its rotation and how deep of a cut it can make. As you can see there is a lot of information packed into this subject! Let us look at some common tool types and applications.

After completing this lesson, you will be able to:

- Identify a tool by shape.
- Distinguish a tool's use based on geometry.

L4-01 – Types of milling tools

- Describe roughing vs. finishing tools.
- Explain milling tools and their functions.
- Identify types of tool holders.
- Identify types of tools.

L4-02 – Types of lathe tools

- Describe the difference between mill and lathe tools.
- Identify tool shape and nomenclature.
- Identify types of tool holders.

Practice Exercises

3x Practice Exercises

- 1) Use direct modelling techniques to remove features.
- 2) Create a 3D model based on parameters.
- 3) Convert Bodies to Components.

Challenge Assignment

1 Challenge Assignment – Parametric Drawing for CNC Programming

Chapter 3: Setting up CAM Programs

Lesson 11: How to define a CAM setup in Fusion 360

A CAM setup contains valuable information about a CNC program. The size of the raw stock, the location of a coordinate system, which bodies are included to be cut, if there is a vice or fixture as well as general information about the program itself. Without this setup information Fusion 360 will be missing a vital part of the process. Let's explore the process of setting up a CAM program for CNC milling.

After completing this lesson, you will be able to:

- Define a coordinate system.
- Identify stock setup information.
- Explain the process of defining a CAM setup for milling.

L1-01 – Setting up stock 5:30

- Use Inspect to measure a part
- Show how to set stock in a setup

L1-02 – Locating and orienting a coordinate system 3:00

- Modify stock dimensions
- Demonstrate how to set a WCS

L1-03 – General setup parameters 4:30

- Modify a program number and comment

Lesson 12: Creating and Managing Tool Libraries

We have learned all about the different types of tools available, but now we need to understand how to create and manage digital versions of the physical tools we have. Fusion 360 allows us to store tool libraries locally or on the cloud, but we are responsible for ensuring the tools accuracy. In this lesson you will learn how to make and utilize cutting tool libraries.

After completing this lesson, you will be able to:

- Create a custom tool.
- Modify tool parameters.
- Create and manage a tool library.

L2-01 – Creating a custom tool 12:00

- Create a custom tool
- Create a cloud tool library
- Modify a copied tools parameter

L2-02 – Utilizing a tool library 6:00

- Use Import to load a tool library
- Modify tool order

Lesson 13: Machine configurations

Machine configurations are new to Fusion 360's preview functionality and the first step in creating a digital representation of a physical machine. We can define the speed at which a mill can move and rotate and how far it can travel. These and other parameters will help us validate any tool motions for potential issues on the physical machine. So, let's learn about the process of setting up a machine configuration.

After completing this lesson, you will be able to:

- Create a machine configuration.
- Explain why machine configurations are important.

L3-01 – Create a machine configuration 11:30

- Modify a machine configuration

Practice Exercises

3x Practice Exercises

- 1) Create a Milling CAM Setup 1.
- 2) Create a Milling CAM Setup 2.
- 3) Create a custom milling tool.

Challenge Assignment

1 Challenge Assignment – Setting up a CAM file

Chapter 4: Creating your first CNC program

Lesson 14: Cutting a Waterjet part

Cutting waterjet or laser parts can be a great introduction to CAM programming because it limits the number of parameters to adjust. There are many complex motion waterjet and lasers machines on the market, but you will limit the motions to only 2 axes. In this lesson you will program a basic 2 axis part.

After completing this lesson, you will be able to:

- Identify a waterjet tool.
- Create a waterjet toolpath.

L1-01 – Creating a CAM setup for waterjet 3:30

- Create a new CAM setup
- Modify a WCS

L1-02 – Filtering the sample tool library 2:30

- Use a filter to select a tool library sample tool

L1-03 – Selecting edges and faces for an operation 11:00

- Use face and edge selections for a toolpath
- Modify toolpath parameters

L1-04 – Simulating a toolpath 7:00

- Use Simulate to validate tool motion

Lesson 15: Introduction to 2.5 Axis Machining

CNC machines and toolpaths can vary in their motion and amount of control. The next step we want to take in the learning process is 2.5 axis machining. This type of machining is used when the tool needs to move up and down for positioning, but only move side to side when cutting geometry. We are going to explore this type of machining by learning about facing and contour operations.

After completing this lesson, you will be able to:

- Review a CNC Mill setup.
- Apply basic knowledge to create a facing toolpath. Identify various options when creating a toolpath.

L2-01 – Reviewing a CAM setup for a CNC Mill 5:00

- Modify the physical material of a part
- Edit a CAM setup

L2-02 – Creating a facing toolpath 10:00

- Modify tool parameters



- Demonstrate how to create a facing toolpath

L2-03 – Creating an external contour 11:00

- Show how to create a 2d contour toolpath

L2-04 – Modifying a Machine Configuration 7:30

- Edit a machine configuration
- Edit a toolpath

Lesson 16: Machining Open and Closed Pockets

A lot of machining jobs can be broken down into two main types of material removal. Open or closed pockets. The two types of geometry can have a different tool approach but the process of setting them up is nearly identical. You will take a look at examples of both open and closed pockets so we have an understanding on the setup and options for machining these types of geometry.

After completing this lesson, you will be able to:

- Identify open and closed pockets.
- Demonstrate how to set up a pocket toolpath.

L3-01 – Identifying types of pockets 2:45

- Demonstrate how to identify pockets
- Use Measure to find minimum internal radii

L3-02 – Roughing a closed pocket 4:30

- Use an adaptive motion toolpath
- Demonstrate how to rough a closed pocket

L3-03 – Roughing an open pocket 4:30

- Modify a contour toolpath
- Use Duplicate to copy a toolpath

L3-04 – Rest machining pockets 8:00

- Use 2D contour to finish a pocket
- Modify a toolpath parameter

Lesson 17: Drilling and tapping toolpaths

The last piece of the introductory puzzle is drilling and tapping holes. With the basics of facing, contouring and pocketing out of the way, the last operation you are likely to start with would be drilling and tapping. In this lesson we learn about drilling cycles and basic parameters needed to drill and tap holes.

After completing this lesson, you will be able to:

- Describe drilling cycle parameters.
- Create drilling and tapping toolpaths.

L4-01 – Spot drilling holes 4:00

- Modify a tool diameter
- Create a spot drilling toolpath

L4-02 – Creating a peck drilling toolpath 3:30

- Create a peck drilling toolpath

L4-03 – Posting G code for a basic CNC program 4:30

- Create an NC file

Practice Exercises

3x Practice Exercises

- 1) Create a waterjet program.
- 2) Create facing and contour toolpaths.
- 3) Create adaptive clearing and contour toolpaths.

Challenge Assignment

1 Challenge Assignment – Create a New Tool Library and add a tool to it, then create a CNC program

Chapter 5: Course Assessment and Peer Review Assignment

Challenge Assignment

2x Challenge Assignments

Create a dimensioned model drawing and format on a 2D drawing border

Created a shared CAM file which you will upload to the cloud for collaborative review