Introduction to computer aided manufacturing (CAM)
Lesson 1

Tony Schmitz
University of Tennessee, Knoxville/ORNL

This program is funded by the Department of Defense (DoD) Industrial Base Analysis and Sustainment (IBAS) Program from the Office of Industrial Policy. DoD has collaborated with ORNL and IACMI to establish America’s Cutting Edge (ACE), a national initiative for machine tool technology development and advancement.
Machining operations

- chip formation by shearing with a sharp cutting edge
  - cutting force, power
  - temperature increase, tool wear
  - tool material, coatings, lubricant/coolant
- turning: straight turning, profiling, facing, drilling
  - parameters: depth of cut, feed per revolution, spindle speed, feed, cutting speed
- drilling: spot drilling, deep hole drilling
  - tools: spot/center drill, twist drill
  - parameters: drill diameter, feed per revolution, spindle speed, feed, cutting speed
- milling: face, peripheral, chamfer, up/down (conventional/climb)
  - milling cutters: face mill, square (flat nose) end mill, bull nose end mill, ball nose end mill
  - parameters: spindle speed, feed per tooth, feed, cutting speed, axial depth (stepdown), radial depth (stepover)
Computer numerically controlled (CNC) machining

- the part is designed using computer software to provide a digital model of the desired geometry – computer aided design (CAD)

- the CAD model is used in computer aided manufacturing (CAM) software to generate the instructions, or toolpath, for the CNC machine to produce the part

- the CNC part program, that includes the toolpath and other machine instructions, is uploaded to the CNC controller on the machine tool
- the part is machined and inspected
Key CAM software considerations

- the instructions are provided using M and G codes – computer code that is interpreted by the machine tool controller

The purpose of CAM is to use your part geometry and selected tools (face mill, end mill, drill, etc.) to produce that geometry from the stock model (rectangular block, forging, casting, additively manufactured preform).

CAM output is a part program. The exact format of the program depends on your machine’s controller (Fanuc, Siemens, Haas). It must be post-processed for the selected controller (“post the program to machine x”).
Key CAM software considerations

- process definitions in CAM includes planes for: rapid motions, feed motions, stock top, feature locations in CAD model

- machining parameters
  - axial depth of cut, stepdown
  - radial depth of cut, stepover
  - spindle speed
  - feed per tooth

- feed in motions
- feed out motions
Key CAM software considerations

▪ process definitions in CAM includes tool selection

▪ when the machine-spindle-holder-tool is selected, a **dynamic system** is defined
▪ the cutting force causes vibration because the tool is not rigid, which affects the machining process
▪ this dynamic system must be considered when selecting machining parameters

▪ More on this!
Your first task is to download Fusion 360

https://www.autodesk.com/campaigns/education/fusion-360
Lesson 1: Open the sample project

- Download the Intro to 2D Machining file (available online, instructions in video and listed below).
- Make sure that you’ve saved the example part file to a new folder so that you can modify it using the remaining videos.

What we cover in this lesson

- Information about this course.
- Lesson objectives.
- Opening the sample Project.
- Creating a New Project folder.
- Saving the Project.

Open the sample design file

1. Click Show Data Panel to open the Data Panel.
2. From the CAM Samples project, Tutorials folder, open the Intro to 2D Machining design file.

Next lesson

Setup

🕒 6 mins 28 seconds
Answer the following multiple choice questions in the online quiz.
1. The tutorial series is for: a) turning; b) additive; c) 2D milling; d) 3D milling.
2. The workspace for the sample part was set to: a) Design; b) Manufacture; c) Milling; d) Drawing.