

CAM Exercises

Introduction

The goal of all CAM software providers is to speed the process of transitioning a concept from Art to Part. Regardless of the software package you are using there are some basic things that they all must do. Throughout this text I give examples using ESPRIT ([DP Technologies](#)) but the concepts and the steps apply to all of the CAM packages I've had occasion to use and I expect they will continue to be important to programmers machine tool operators and manufacturing engineers for many years into the future.

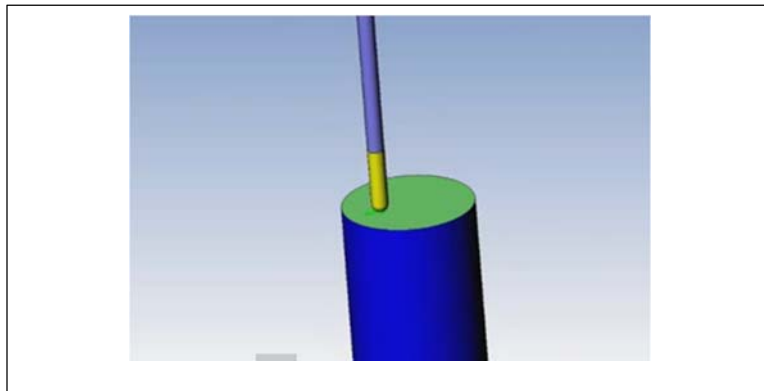
To effectively use CAM software users need an understanding of machining processes, machine tool operation and setup, as well as an understanding of the designer's intent. This guide is intended to give a brief introduction to each of those areas.

CAM Exercise 1 Creating NC Code

The code that is ultimately run on the machine tool is called NC-code, G-code, or G&M-code.

This NC-code consists of a list of commands read by a machine tool's controller "telling" the machine what to do.

In this exercises you will start with a CAM file that faces the top of a round workpiece and engraves the text John Smith on the top.



You will then have a chance to edit the engraving text, and finally to create the NC-Code that can be loaded into the machine tool's controller.

Read this introduction before completing the exercise which is available on the WPI LearnCNC Platform.

Step 1 Before you Start

The first step in the exercise has you create a working folder where you will download an Esprit file (.esp) and a post processor file (.pst).

I recommend that you create your working folder on a network drive that you will have access to regardless of which computer you are using to complete the exercise(es).

<https://youtu.be/WBvCQwIEQWU>

Once you download the files (from the links provided within the WPI LearnCNC Platform to the working folder you created you will learn how to set the default file locations within Esprit. (Setting the Default ESPRIT File location will only be remembered on a per user and per computer basis)

<https://youtu.be/H50LxFULdFs>

Step 2 Simulate the Cutting Operations

No matter what CAM software you are using there will some method to simulate the machining operations you have programmed. This simulation

will tell you what the computer thinks the program will do.

The second step in the exercise has you open the simulation toolbar and the properties tab

As you “play” the simulation you will observe the display of the PVs in the properties tab. You will be able to adjust the speed of the simulation and pause at any time. You can even single step through the simulation forward or backward.

Simulating in the CAM software is one of the 6 key safety steps in the Safe Operation Checklist which we will use while completing the machining labs.

https://youtu.be/KO_DEf_lFow

Step 3 an Zoom and Rotate Commands

While you simulate you will be asked to experiment with the pan, zoom, and rotate commands in Esprit. These tools can be very useful while creating programs and in debugging during simulations.

<https://youtu.be/H1xPz478fY0>

Step 4 Open the Project Manager

In step four you will open the Esprit Project manager and position it on the screen.

The Project Manager provides a wide variety of functions related to machining operations, tools, and features.

To display the Project Manager:

- Press the F2 key
- On the View menu, select Project Manager

The Project Manager is comprised of separate managers:



CAD Features: This tab is only visible when a CAD file is opened that was saved using the ESPRIT FX add-in. Displays the CAD design feature tree of a solid model.

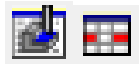


Features: Lists every work coordinate and feature and the operations placed on those

features. Create, edit, and associate features to work coordinates, copy, rename, and delete features. You can also simulate cutting operations by work coordinate or by feature.



Tools: Lists every tool in the current session. You can create, edit, rename, copy, and delete tools and have access to tool libraries.



Operations: Lists every cutting operation in the current session. Use the Operation Manager to edit, rename, copy, delete, re-order, sort, and simulate operations.

The Project Manager window can be dragged to a new location or resized to accommodate your preferences. Click, hold, and drag with the pointer on the bar at the top of the Project Manager:

<https://youtu.be/7WMO0dlCfKI>

Step 5 Specify the Work Coordinates

The work coordinates, also referred to as the workpiece's origin, are one of the most important things that the programmer controls when creating

a CNC Program. With a typical 3 axis CNC Milling Machine the Spindle will move up and down along a Z axis while the workpiece is fixed to table that moves in the X and Y directions.

When the programmer assigns a work offset, or work origin, to the program it allows them to write the program without knowing where on the table the operator will fix the workpiece and in fact without even knowing specifically which machine tool will be used.

<https://youtu.be/LNzKBvR4Gno>

Step 6 Edit the Operation

Once you have set the work coordinates you will edit the Text to Engrave text in the operation changing “John Smith” to your name and then simulate again.

Step 7 Preparing to Create NC-code

The code you will create is primarily a list of commands telling the machine tool where to position the tip of the tool and how fast to move from point to point, how fast to spin the spindle and when to turn the it on and off, and when to turn on and off other auxiliary functions of the machine tool. These types of commands are typically preceded with a G or M followed by a number indicating what the command is and then any coordinates or other information the machine tool needs to know in order to complete the operation.

For example:

```
"G54 G01 X1. Y1. Z1. F35 M08"
```

Tells the machine tool to position the tip of the cutting tool:

- using the reference coordinate system stored in the 54 memory register (G54) and
- moving in a straight line (G01)

- at a speed of 35 inches per minute (F35)
- from wherever it happens to be
- to the position X=1 inch, Y=1 inch, and Z=1 inch (X1. Y1. Z1.) and
- once arriving at (1,1,1) to turn on the coolant pump (M08).

In addition to these types of commands the NC-code can also include information that is useful to the operator of the machine too but is ignored by the machine itself.

In this step of the exercise you will input some of this type of information. You will Open the Machine Setup Dialog and et the **Program Name** to be “your username” and the **Program Number** to 18018.

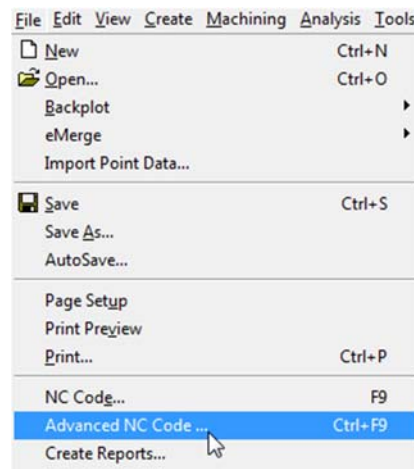


<https://youtu.be/dJVig3uP8>

Step 8 Create the NC-Code

The process of creating NC-code from the information the CAM software has stored in its internal format is called post processing. Different machine tools sometimes use different G and M codes so CAM software programs will typically use lists of conversion parameters that are saved in what are called "post" files.

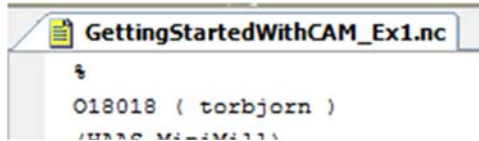
In this step of the exercise you will tell ESPRIT which post file to use by opening the Advanced NC Dialog and then you will create NC code that can be sent to a machine tool.



<https://youtu.be/Qmb6o9wXEVA>

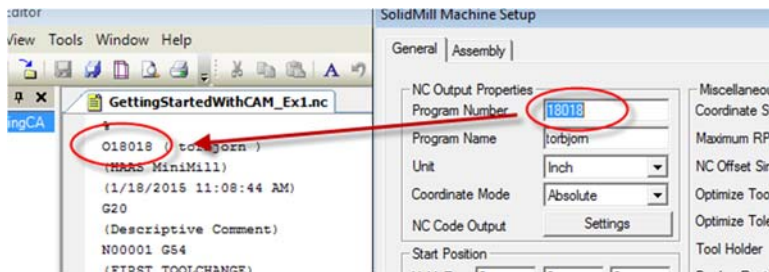
Step 9 Examine the code

The NC-code file created is saved as a text file with the .nc extension and will open in the default NC-code editor you will note that the file begins and ends with the symbol "%"



The % at the beginning and end of the file indicates to the machine tool where the NC-code begins and ends within the file. If the symbols are missing the file will not load correctly.

You should also note that the first actual line of code starts with the letter "O", followed by the program number, followed by the Program Name. The program number cannot exceed 5 digits and should not start with a "9xxxxx"



Exercise 6 Complete the CAM 1 Exercise

ME1800 Labs [View Course](#)
ME1800 Lab Exercises

ME1800 Labs

- ✓ **CAM 1: Getting Started**
 - Module Introduction and Pre-Test
 - Getting Started With CAM - Creating NC Code**
 - Module Review and Post-Test
- **CAM 2: Standard Milling Operations and**