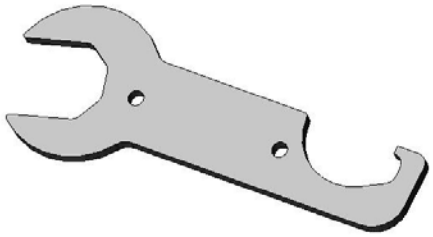
Due by e-mail by the start of class on 23 April – instructor@engineering.ucsb.edu – **With the subject line ME158-HW3-<your last name>.**

Part 1 – Importing Solidworks Geometry into MasterCAM

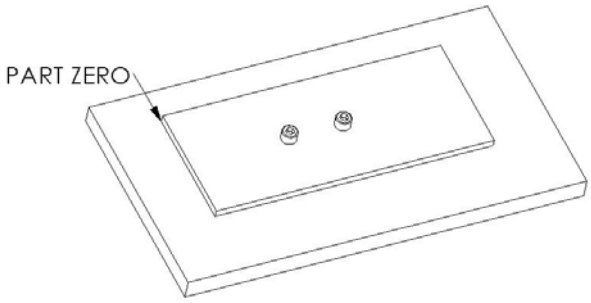
This week you will use MasterCAM to write a program that we will make in the shop. The part is a wrench that can be used to repair the headset and bottom bracket on a bicycle.



You will machine the part out of a rectangular piece of 3/16" thick aluminum bar. It will be clamped to a tooling plate on the mill, so that all you need to machine is the perimeter of the wrench.

Details:

Stock size:	7.2" x 3" x .187"
Cutting tool:	3/8" diameter 2-flute end mill, programmed as tool #1
Part zero:	Upper left hand corner of the stock, **bottom surface of the stock (top surface of tooling plate.**
Cutting direction:	Clockwise (climb milling)
Cutting speed:	2000 RPM
Feed rate:	Plunge 3 ipm, horizontal 10 ipm
Postprocessor:	**Dynapath System 10/20 Conversational 4X Mill**



Instructions:

1. Import Solidworks file of wrench into Mastercam from the course web site
2. Set up the stock and cutting tool
3. Program a profile mill with one roughing and one finish pass
4. Verify that the part will machine correctly
5. Save the Mastercam and code files with your initials + "lab1" (example: dpblab1.mcx, dblab1.nc). E-mail them to instructor@engineering.ucsb.edu

Files to save:

1. The .emcx file which contains your Mastercam model (e-mail this file to instructor@engineering.ucsb.edu).
2. The .nc file that contains the code for making your part on the Tree mill.
3. Save the files on your x-drive or IC lab shared directory so that you can get the files from the computer in the shop. You can also save them to a memory stick.

Part 2 – Starting the Class Project
Due at the beginning of class 23 April: a printout of your drawing of the part that you design for your project.

This week you will **begin** working on your final project for the class. For the project you will design a part that can be made on the Tree mill in the student shop and complete all of the steps needed to make the part: create a drawing, manufacturing planning, creating a manufacturing model in Master CAM and post-processing the model. We will make lab time to machine as many of the project parts as possible. Priority will go to parts for ME153 and ME189 projects.

The assignment this week is to create a Solidworks model of the part that you would like to make, and then make a drawing of the model. Creating a solid model is often easier than making a clear drawing of the part. There is a lot of important manufacturing information in a drawing that is not shown in most models such as tolerances, surface finishes and geometrical constraints.

You may change your project idea during the next several weeks – this is a first draft.

Professional-quality drawings are expected – even if it takes a few iterations. The drawings should include:

- Several different views to clearly show the geometry of the part. Show hidden lines in top and side views, hide them in the isometric views. At a minimum a drawing should have a top and side views, and usually a front view as well. Section and auxiliary views are often useful for complex parts. It is very helpful to include an isometric view without any dimensions just to help the reader understand the part quickly.
- All necessary dimensions, but no duplicates. The tolerance shown in a dimension indicates the accuracy with which that feature is to be machines. Generally three decimal points indicates .005" tolerance, and two indicates .015" or .020". The actual tolerance for your drawing is defined in your title block. Ordinate dimensioning is often clearest for parts that will be made on a CNC mill. Critical dimensions are often shown with exact tolerances.
- A title block that includes the part name, the name of the person who created the drawing and the date on which it was created / revised, acceptable tolerances for the dimensions, the material from which the part is being machined and the necessary surface finish.
- Notes explaining information about the part that cannot be shown with dimensions.

When large format plotters are available designers often use C, D and E size sheets (17"x22", 22"x34" and 34"x44") in order to show all of the views and notes on a single sheet. We are limited to A size sheets with our printers, so sometimes several sheets are needed.

The part should me machinable on the Tree's 27" x 14" x 6" machining envelope, and it should have some interesting features – pockets, bosses, lighting holes, complex surfaces, webs etc. A plate with a hole in it won't do. I would really like to see parts that you really need for class projects, work or hobbies. Besides being a bit challenging, the only restriction is that the parts can't be weapons-related or have to do with anything illegal.