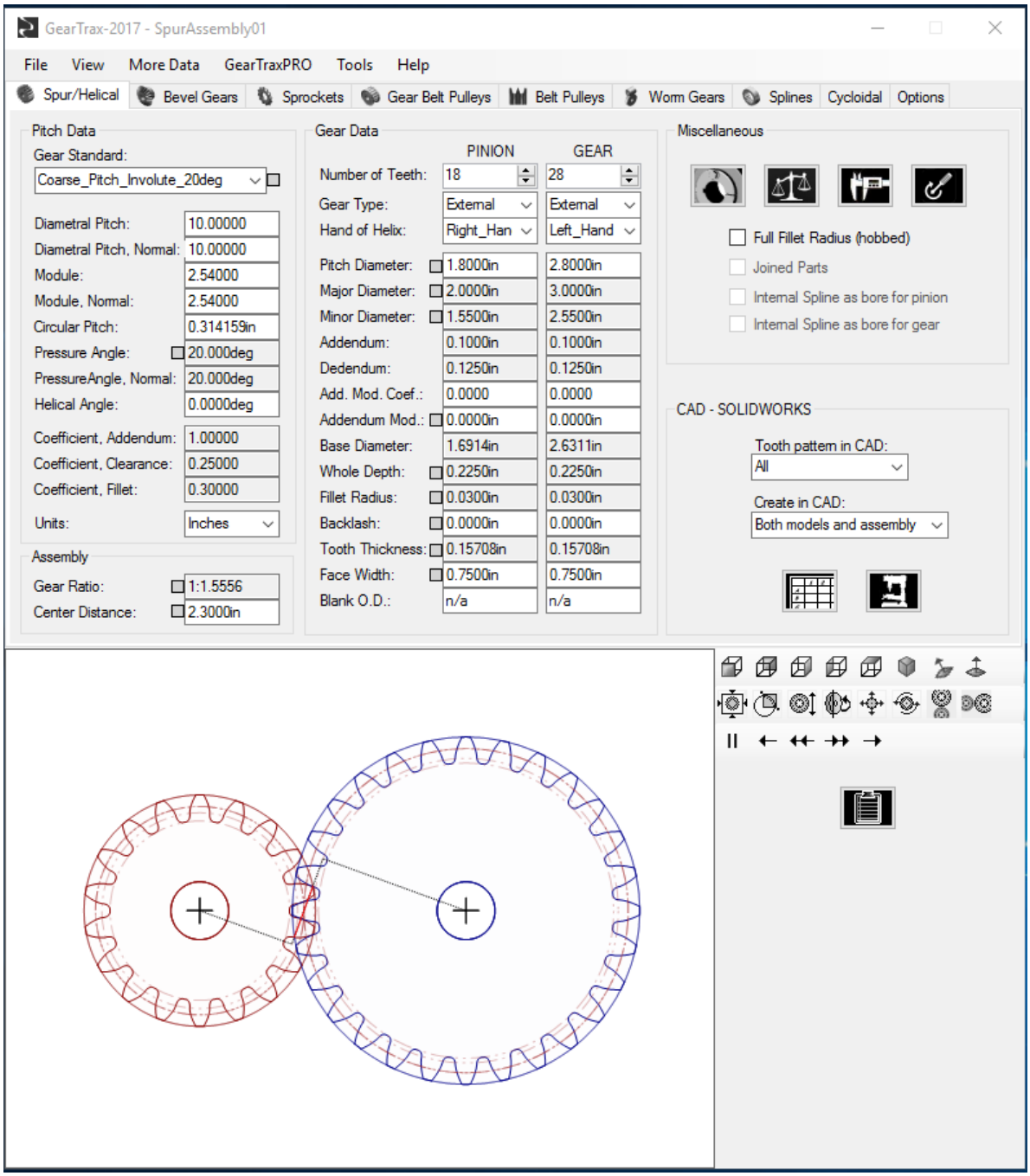
GearTrax 2017 Quickstart guide

Introduction:

GearTrax is a software that will automatically create parts in SolidWorks or other CAD software with gear tooth geometries that match your desired specifications.

Upon opening GearTrax, the following screen will be displayed.



Go to the tab that most directly corresponds with the part you need to make. This example shows the Spur/Helical tab. Generally, the Pinion refers to the smaller gear and the Gear refers to the larger of the two gears.

Terminology:

For the purpose of use in ME301, it is most important to know the following terms:

Diametral Pitch: # of teeth per inch of pitch diameter

Circular Pitch: # of teeth divided by pitch circumference

Major Diameter: diameter to outside edges of teeth

Minor Diameter: diameter to inside edges of teeth

Pitch Diameter: the effective diameter, approximately halfway between Minor and Major diameters

Center Distance: distance between the central axes of gear pairs. Can be calculated by averaging the two pitch diameters.

Gear Ratio: ratio of pinion pitch diameter to gear pitch diameter

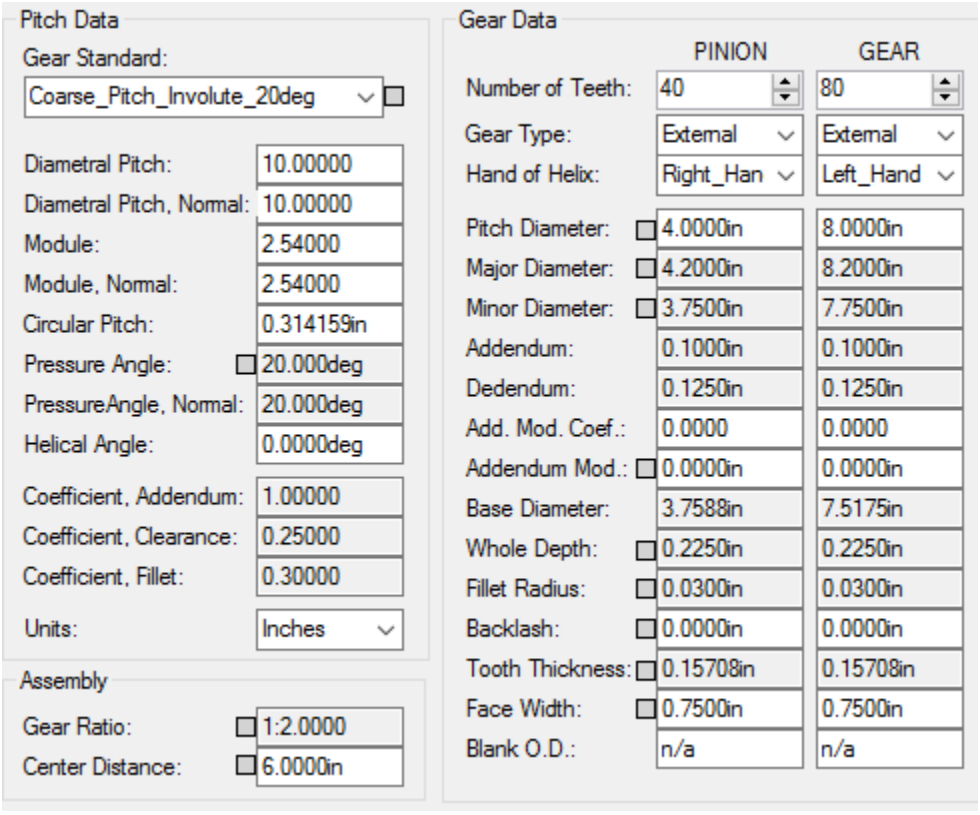
Module: pitch circumference divided by # of teeth; commonly used in metric gears. Module is the inverse of circular pitch.

The above definitions and additional details can be found at the link below. Section 7.4 has terminology for spur gears. https://www.cs.cmu.edu/~rapidproto/mechanisms/chpt7.html#HDR116A

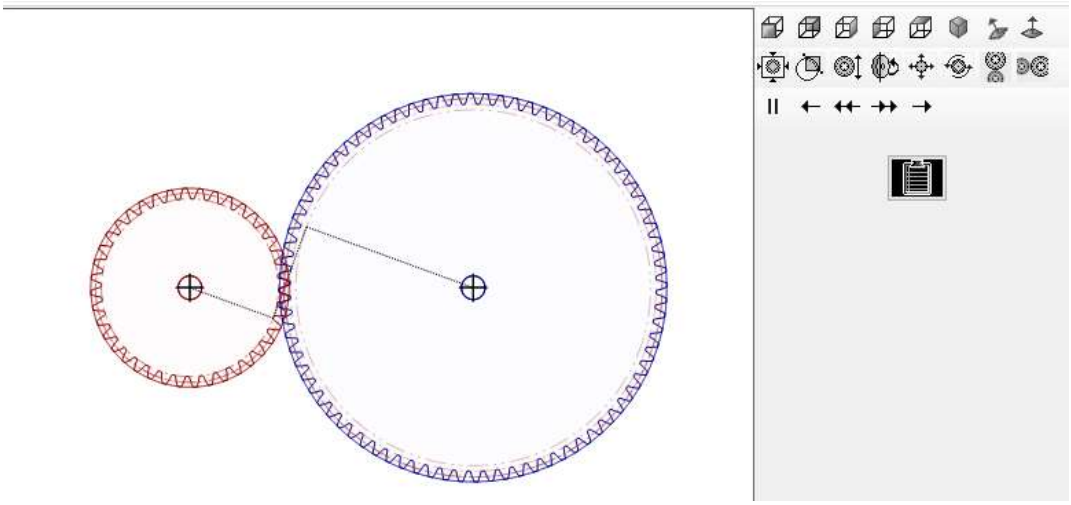
Section 13-2 in Shigley is also a helpful resource for gear nomenclature

Creating the Gear(s):

Specify desired parameters for the gear you wish to make.

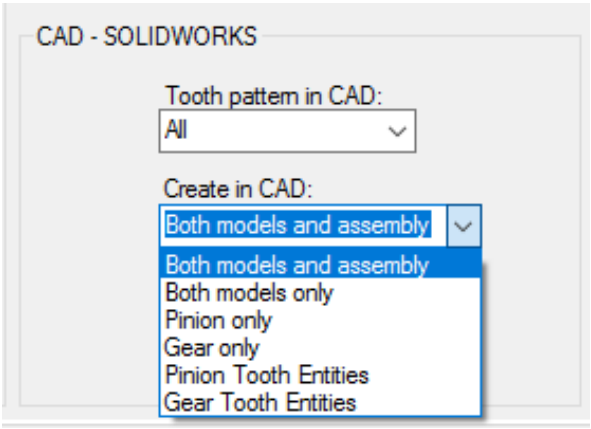


At the bottom of the screen a preview of the gears is displayed. The icons to the right can be used to move the view to check your work. Click on an icon corresponding to action you wish to perform then click and drag in the display area to move the view.



Exporting to Solidworks:

When you are finished with your gears and wish to export it to SolidWorks, select from the dropdown below what you want to export. The "models" correspond to a part in Solidworks and entities produce only the gear teeth in a part file. Note that the tooth entities are specific to each gear and that a sketch must already be open. The profile that is created in the sketch is the area where material should be removed.



Finally, select the "export to CAD" button  and watch the magic happen.