EN1740 Computer Aided Visualization and Design

Spring 2012

2/23/2012

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Last Time:

- Standards and practices governing engineering drawings

Tonight:

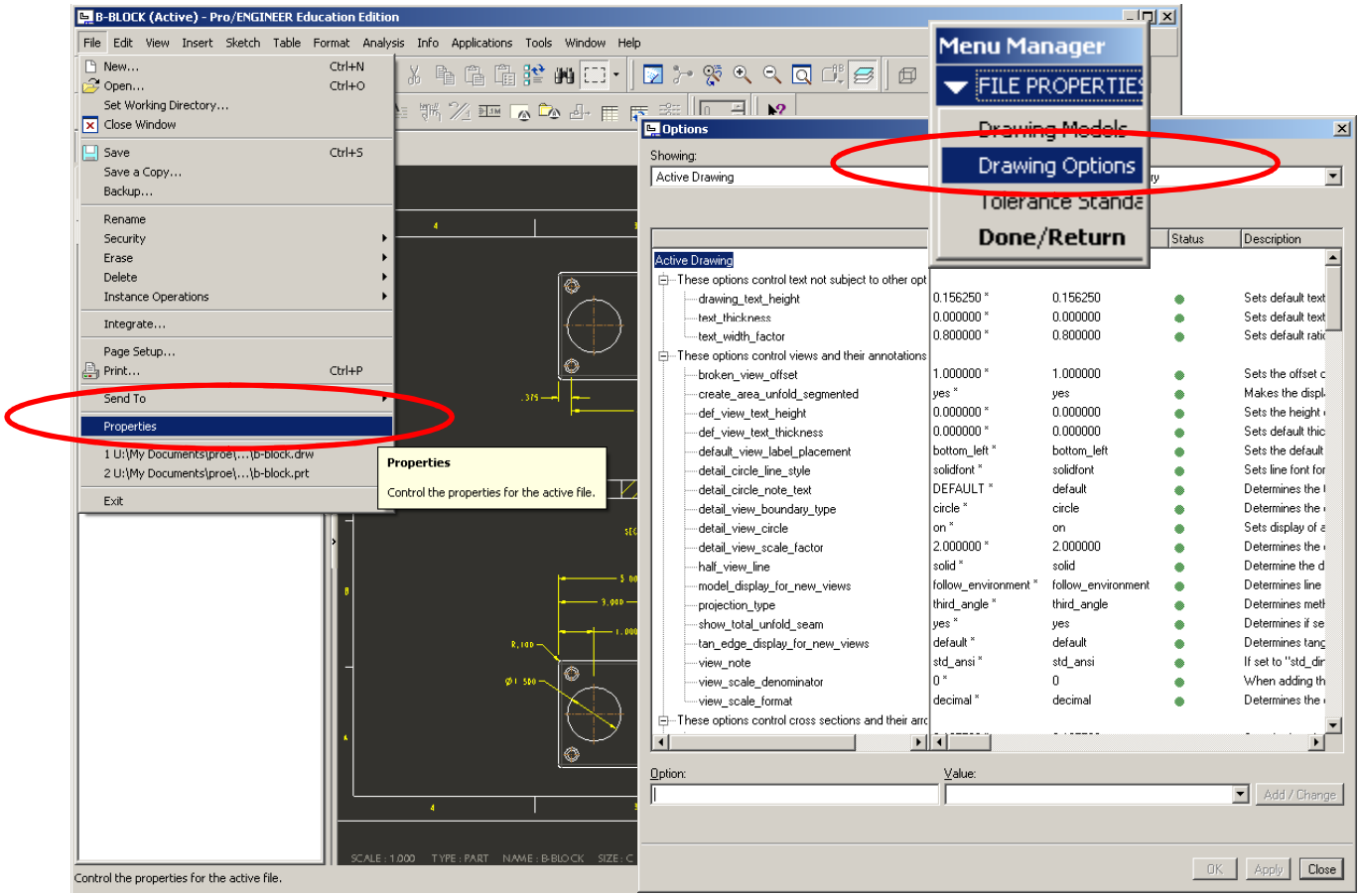
- How to get Pro/Engineer to stick to the standards
  - What can be automated
  - What we must format
- Advanced geometry creation
  - Sweeps
  - Helical sweeps
    - Threads
    - Springs

EXERCISE – Format Extension, Dimension, Leaders and Arrowheads

Arrowheads and Text Height is all that's left - These we can set Automatically!!

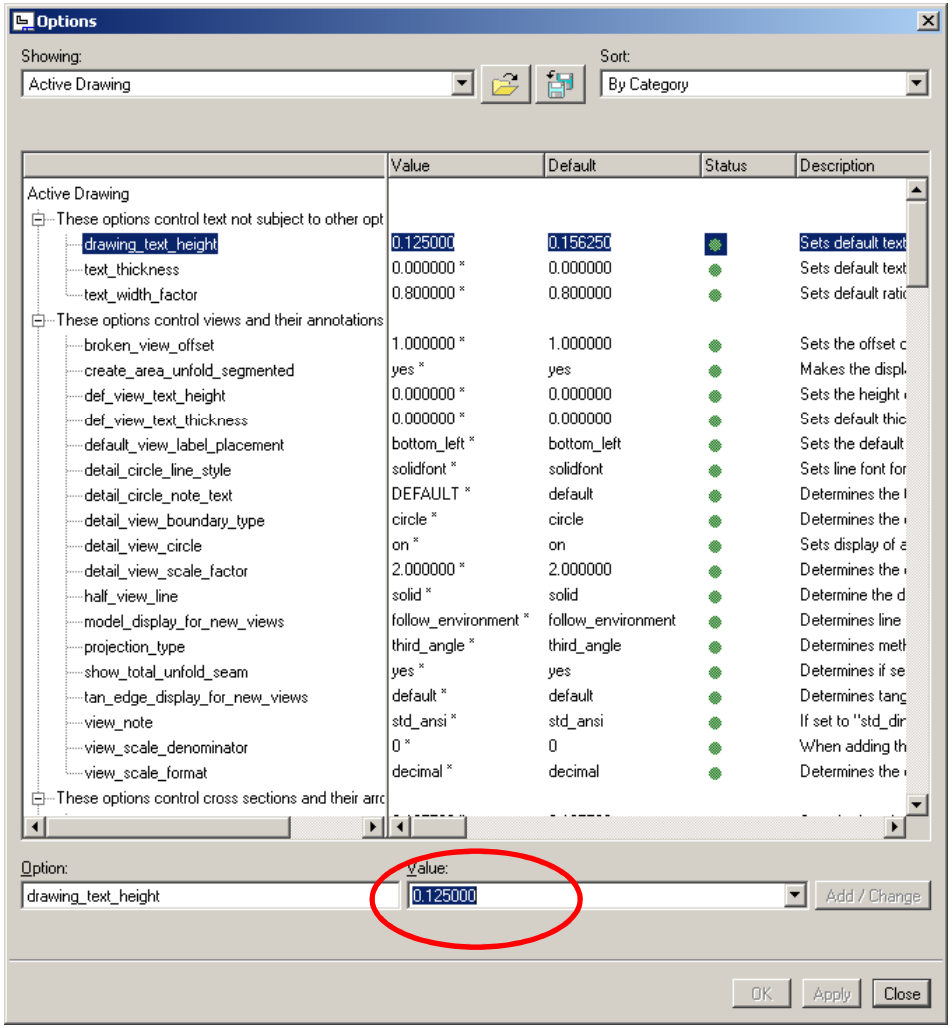
File > Properties >

Drawing Options



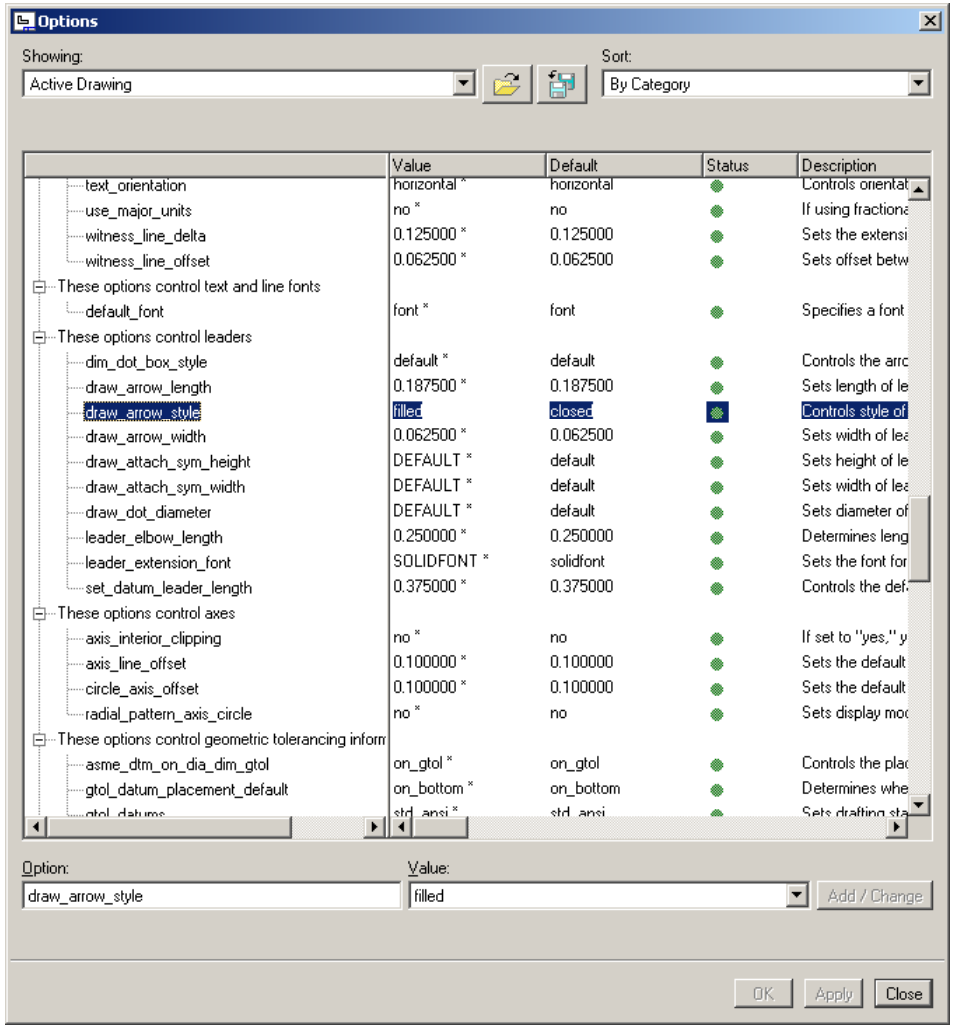
EXERCISE – Format Extension, Dimension, Leaders and Arrowheads

- Find Option “drawing_text_height”
- Change to .125in
- Click Add/Change
- Click Apply
- Click Close



EXERCISE – Format Extension, Dimension, Leaders and Arrowheads

- Find Option “drawing_arrow_style”
- Change to filled
- Click Add/Change
- Click Close

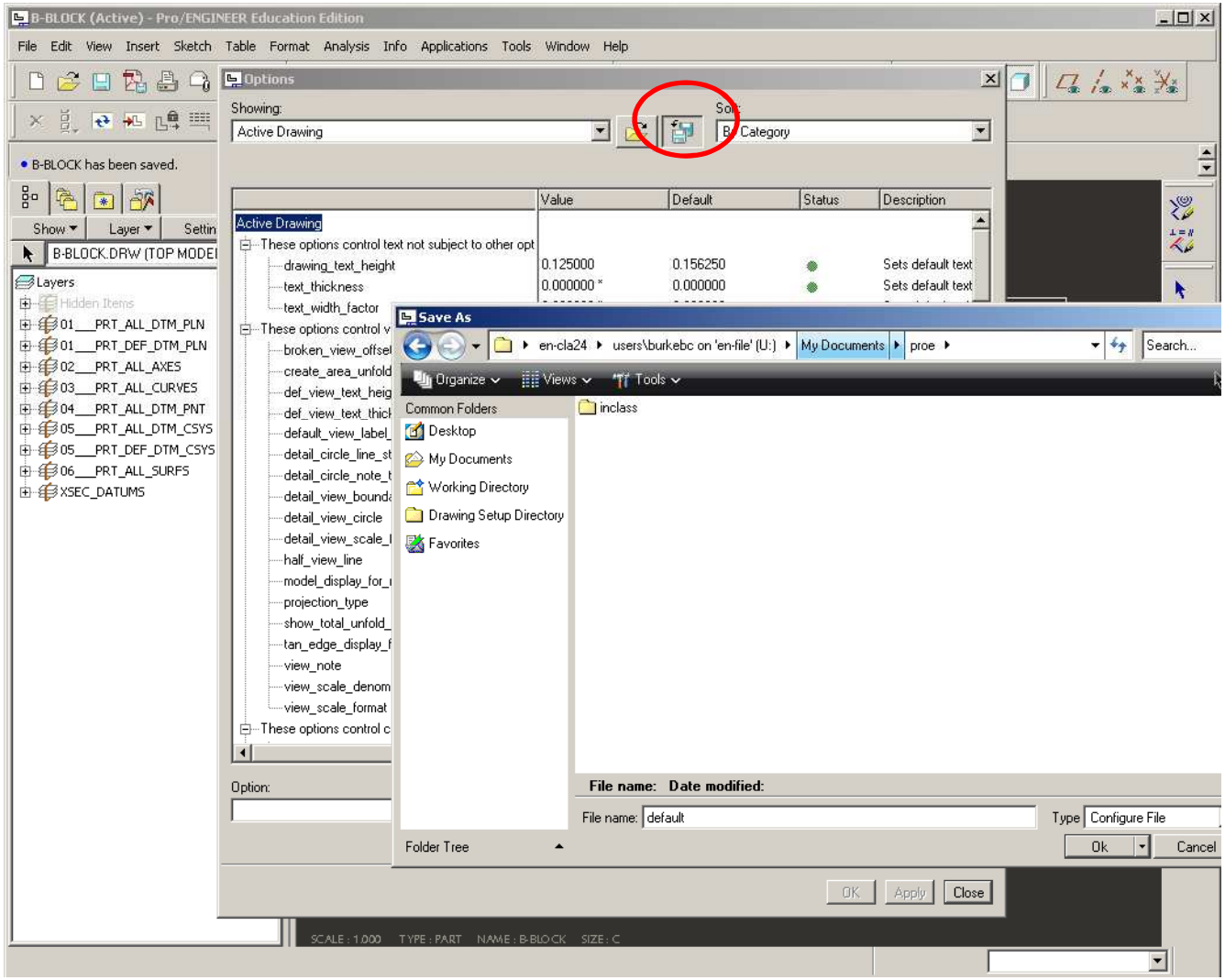


EXERCISE – Format Extension, Dimension, Leaders and Arrowheads

Let's set it and forget it.

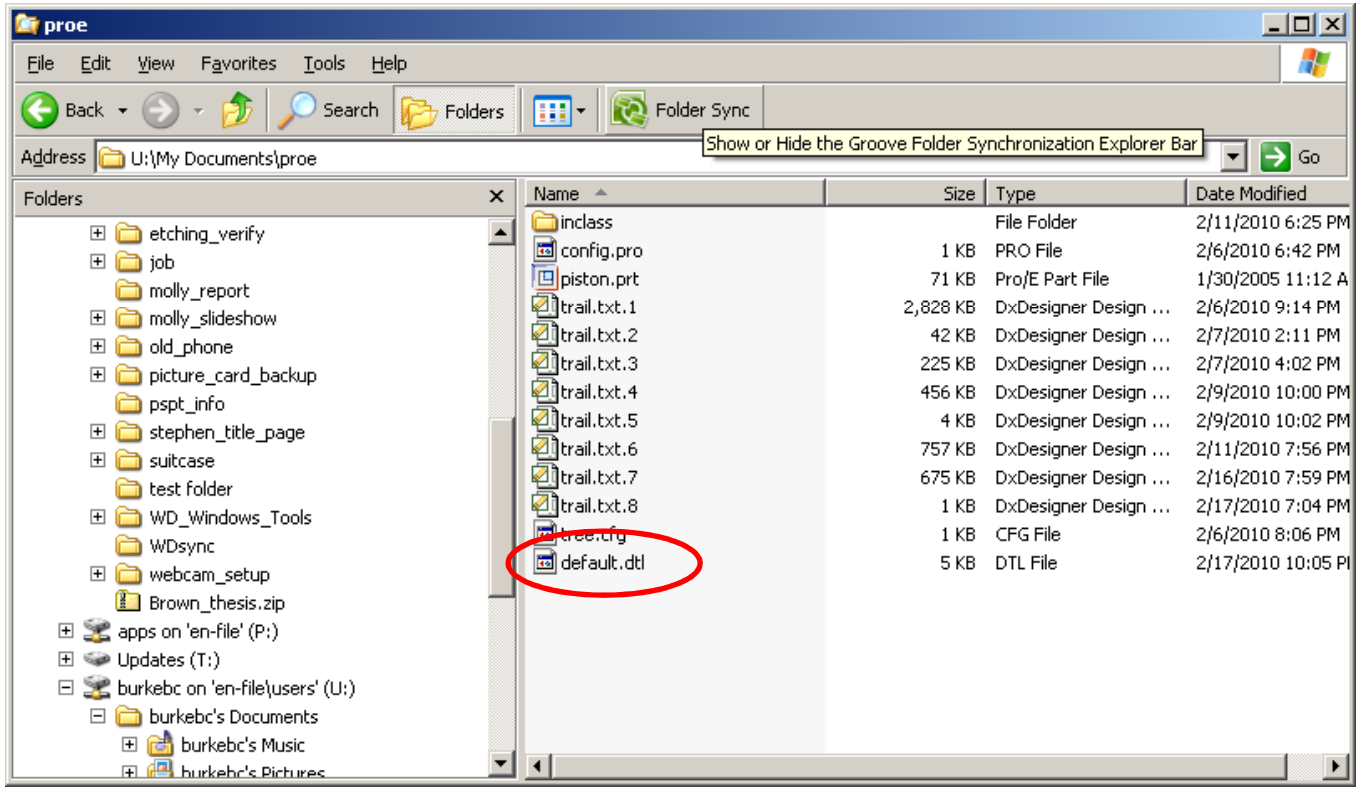
• Save to proe directory >

Filename "default"



EXERCISE – Format Extension, Dimension, Leaders and Arrowheads

Check to make sure this saved OK.

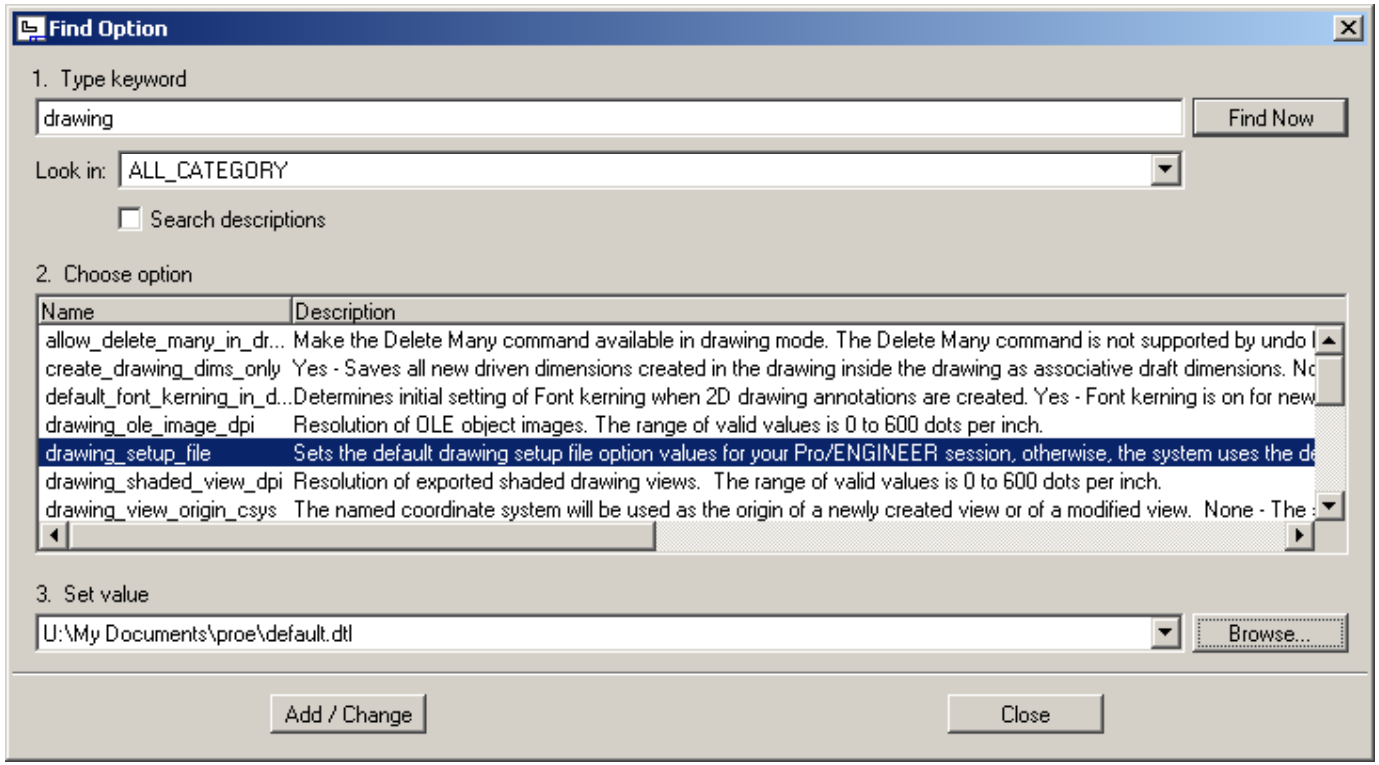


EXERCISE – Format Extension, Dimension, Leaders and Arrowheads

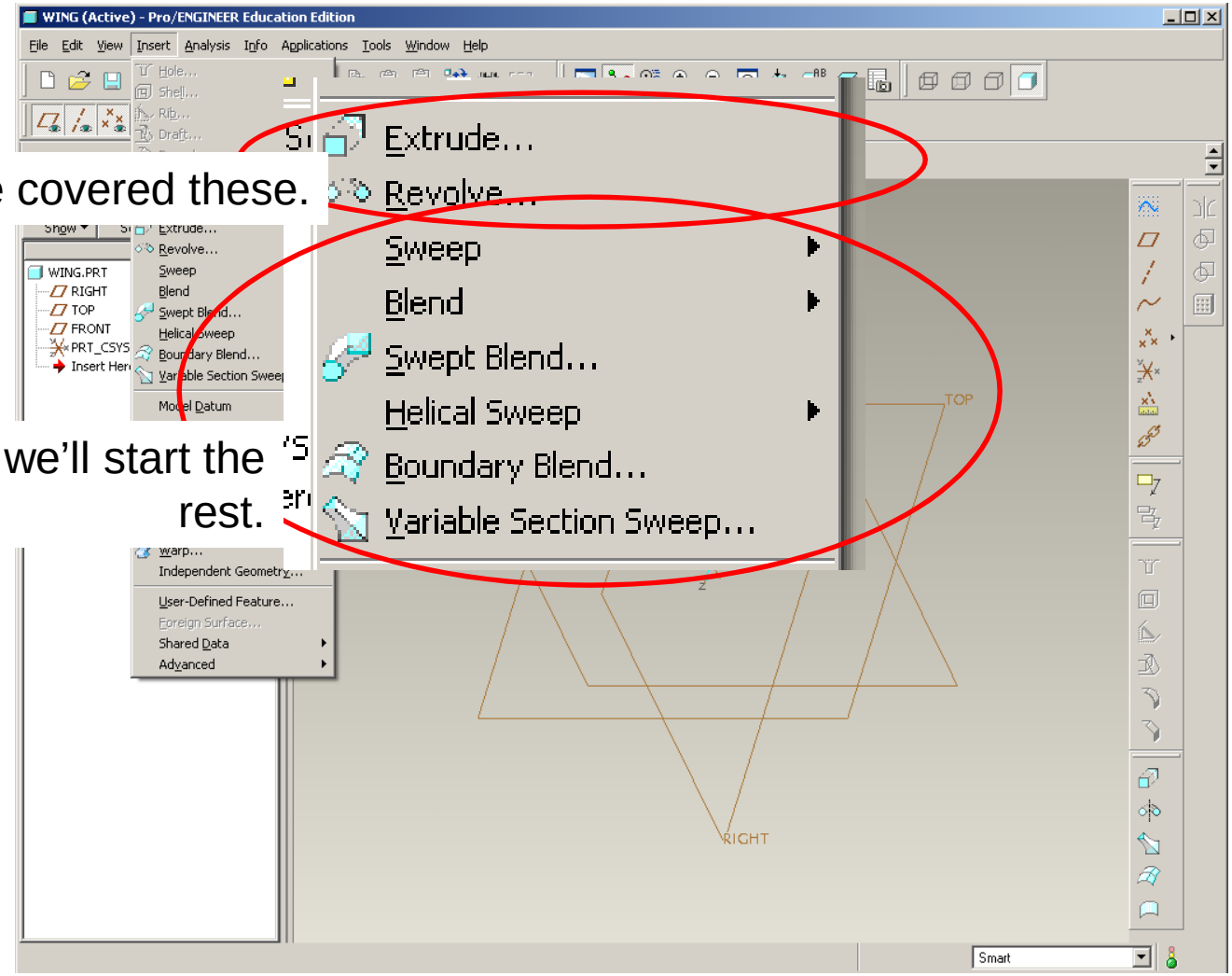
In order for Pro/E to use this drawing standards file every time, we need to add a config.pro option to specify where this file lies

Add drawing_setup_file option and specify path to default.dtl.

SAVE OPTIONS.



Geometry creation tools



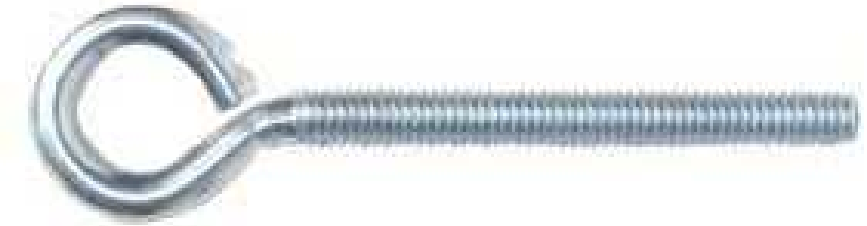
We've covered these.

Tonight we'll start the rest.

Advanced geometry

Stuff we can't make with Extrude or Revolve

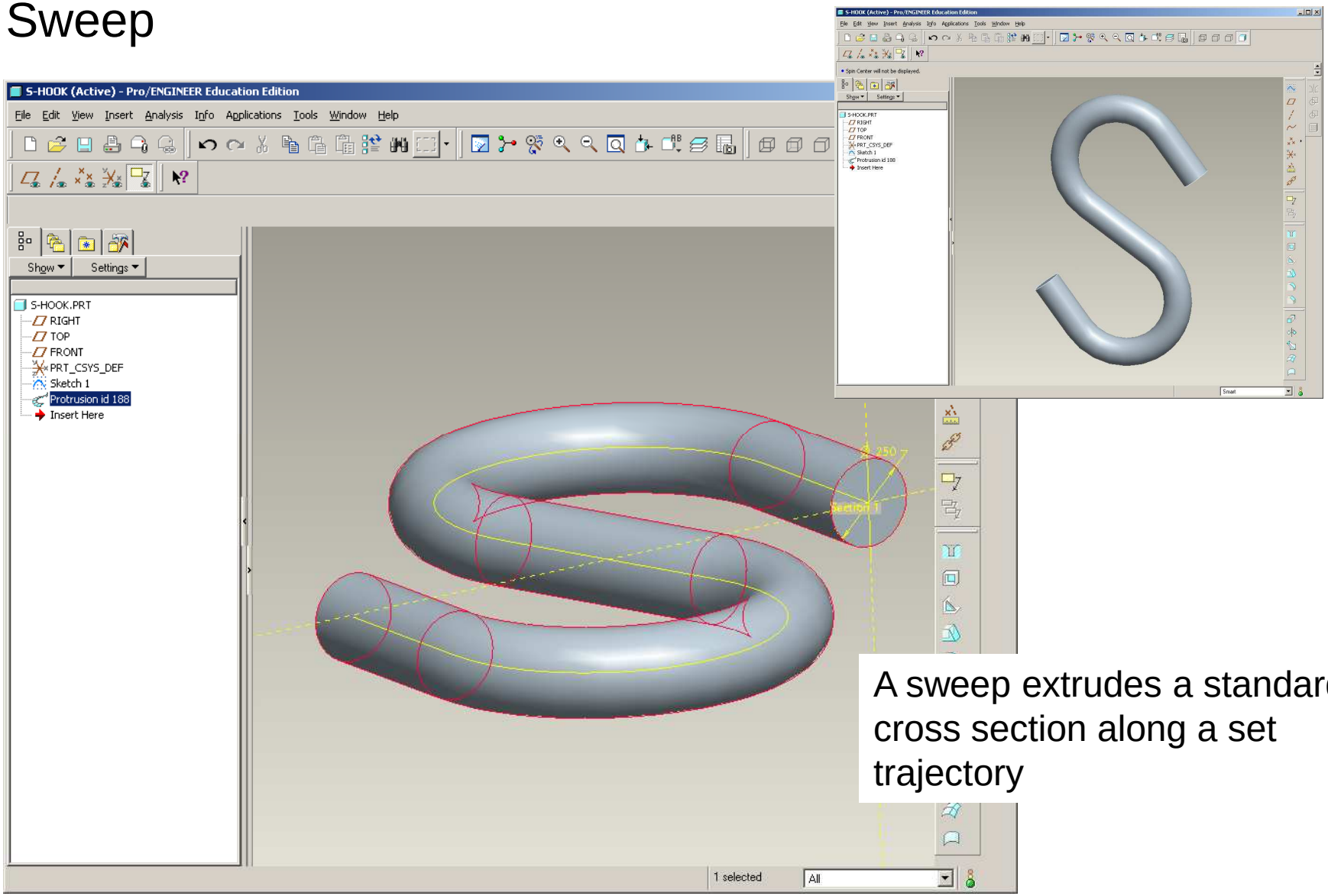
http://aluminumjonboat.com/



http://www.qcsupply.com/qcsupply/browse/productDetailWithPicker.jsp?productId=40995

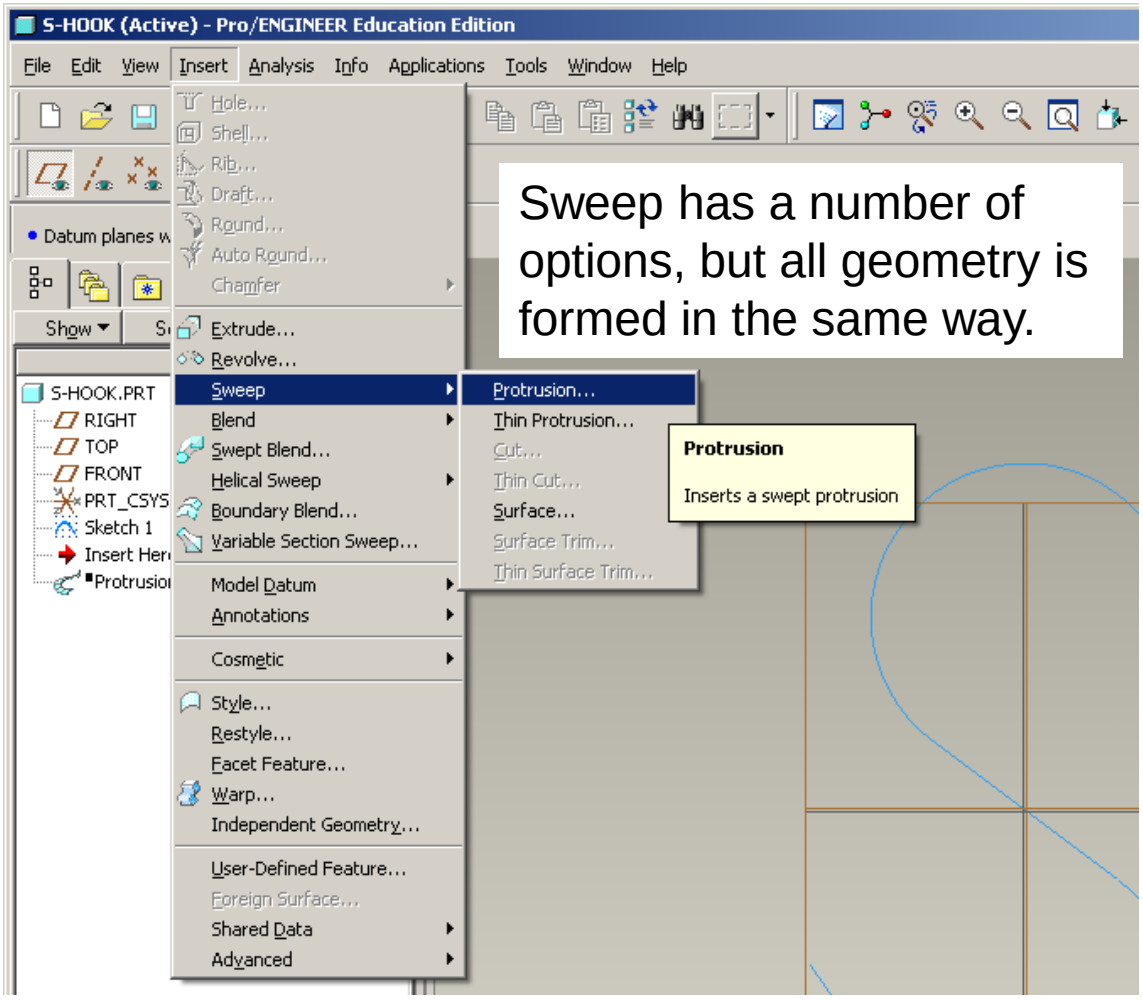
http://concrete-submarine-hull.blogspot.com/

Sweep



A sweep extrudes a standard cross section along a set trajectory

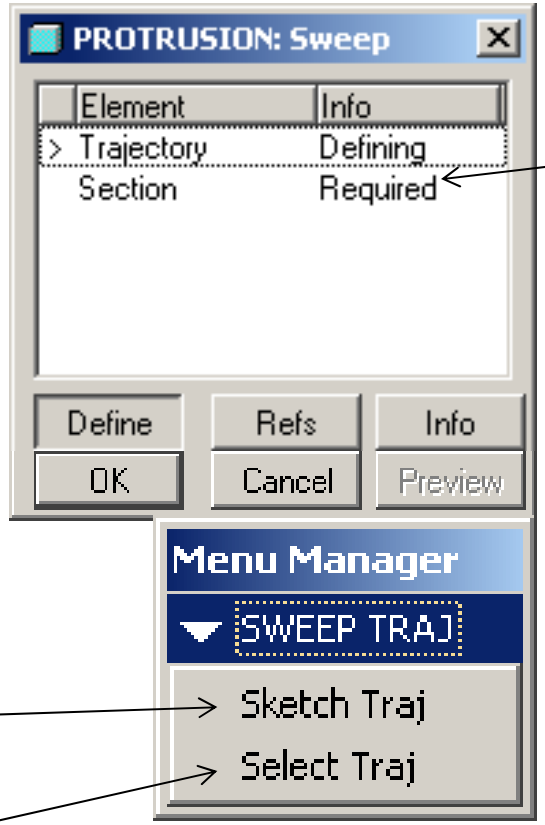
Sweep


S-HOOK (Active) - Pro/ENGINEER Education Edition
File Edit View Insert Analysis Info Applications Tools Window Help
Hole...
Shell...
Rib...
Draft...
Round...
Auto Round...
Chamfer
Extrude...
Revolve...
Sweep
Blend
Swept Blend...
Helical Sweep
Boundary Blend...
Variable Section Sweep...
Model Datum
Annotations
Cosmetic
Style...
Restyle...
Facet Feature...
Warp...
Independent Geometry...
User-Defined Feature...
Foreign Surface...
Shared Data
Advanced
Protrusion...
Thin Protrusion...
Cut...
Thin Cut...
Surface...
Surface Trim...
Thin Surface Trim...
Protrusion
Inserts a swept protrusion
S-HOOK.PRT
RIGHT
TOP
FRONT
PRT_CSYS
Sketch 1
Insert Here
Protrusion
Show
Sweep has a number of options, but all geometry is formed in the same way.

Sweep

Every Sweep feature needs a :

- Trajectory
- Section



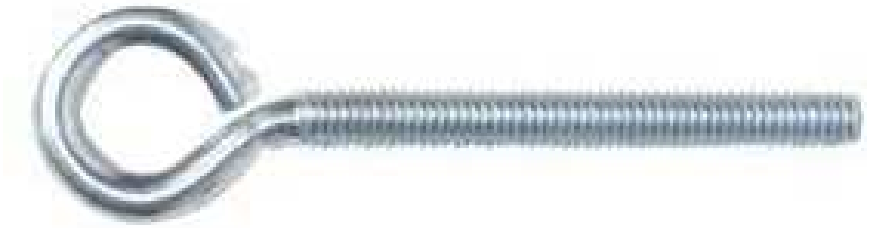
- Sections are created in the same way all sketches are created.
- Origin of section is "Startpoint"

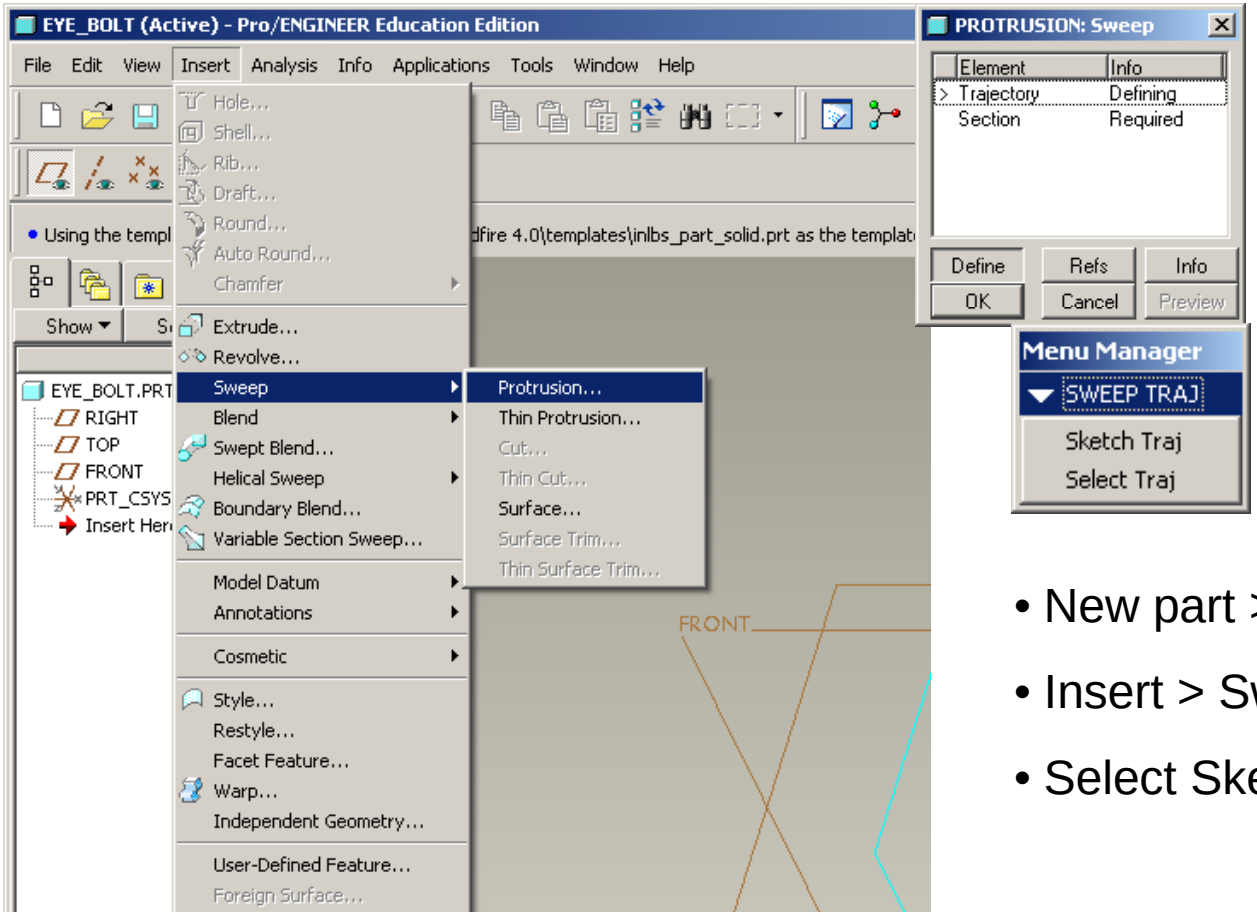
Trajectories can be either:

- Sketched as a part of the feature creation
- Selected from some other part of the model

EXERCISE – Sweep

Let's create an eye-bolt





- New part > name eye_bolt.prt
- Insert > Sweep > Protrusion
- Select Sketch Traj

EXERCISE – Sweep

First step when sketching a trajectory is to define sketch plane

This is a different way of doing something you're already used to

Step 1 – Select sketch plane

FRONT plane for this exercise

Step 2 – Accept or Flip viewing direction

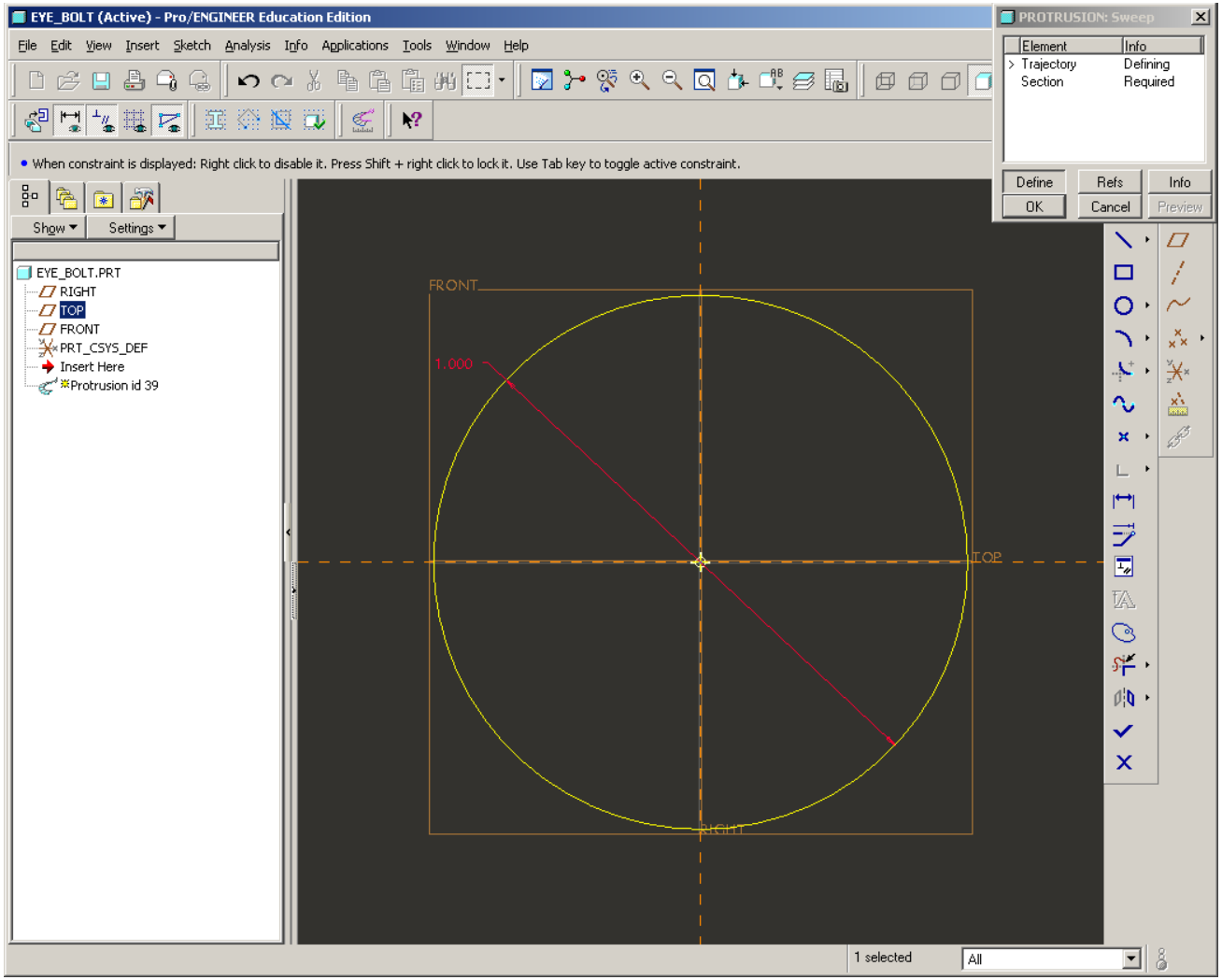
ACCEPT for this exercise

Step 3 – Select reference plane

Use TOP datum as Top reference for this exercise

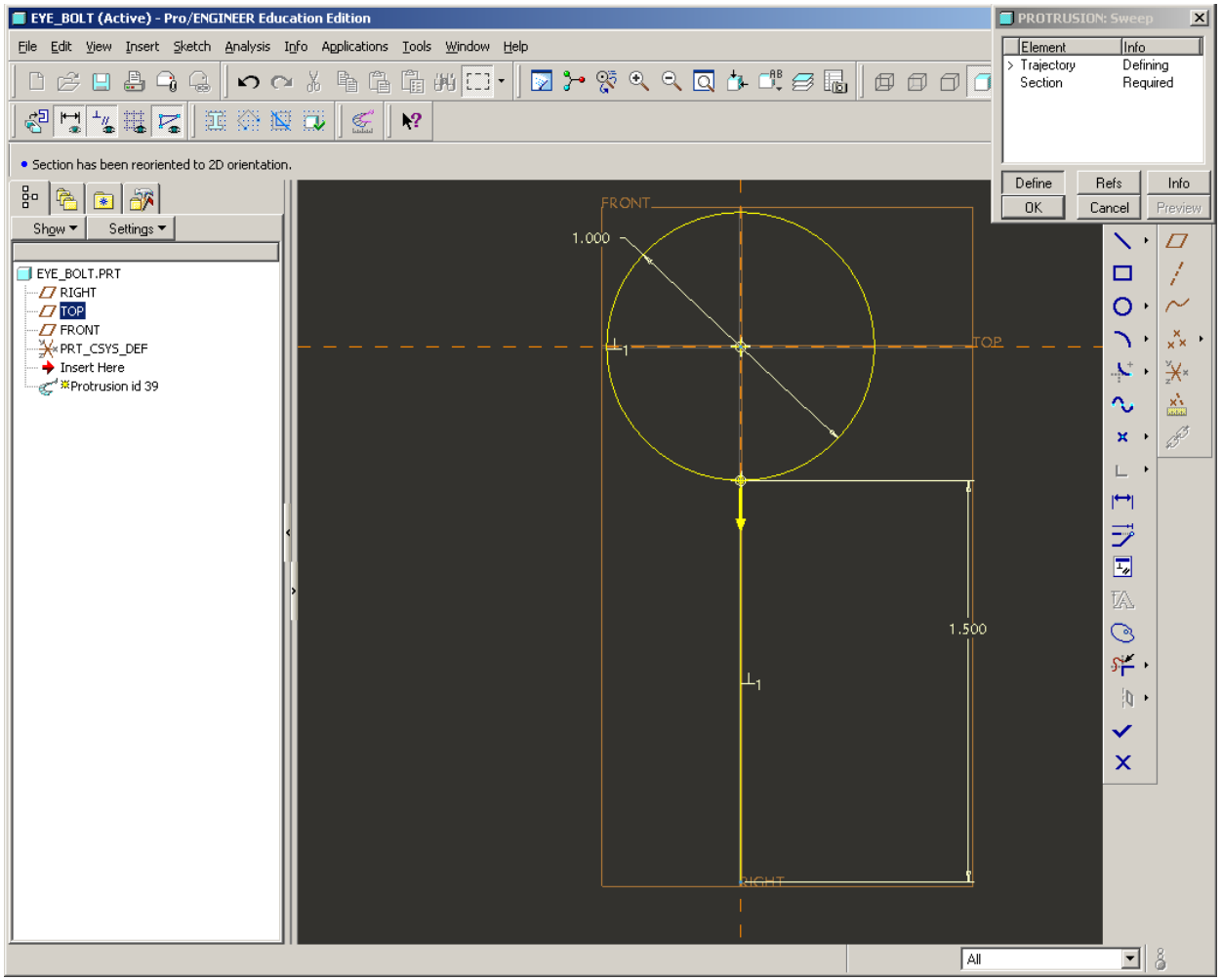
EXERCISE – Sweep

Create a 1” circle to scale sketch



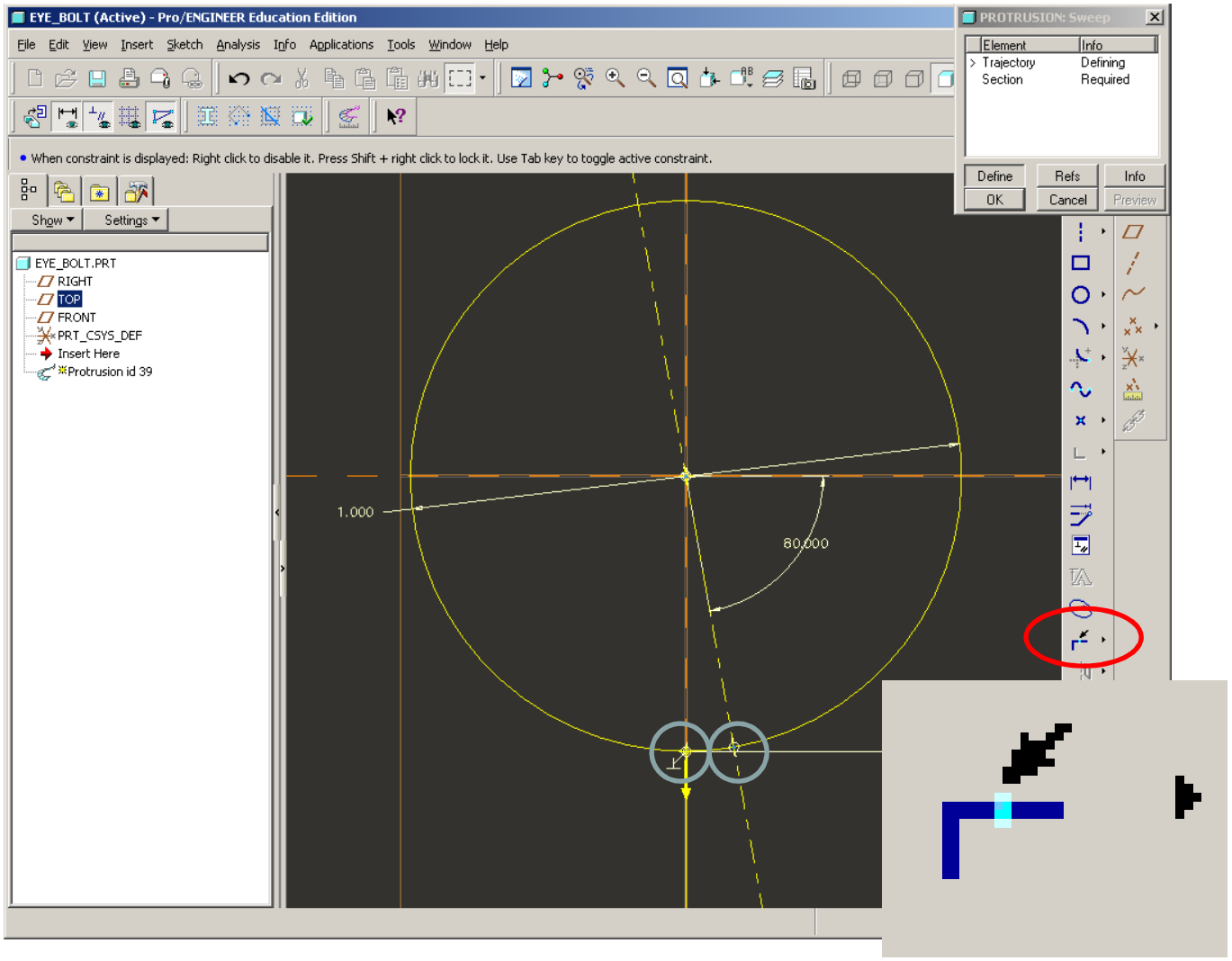
EXERCISE – Sweep

Create a 1.5” leg from circle.



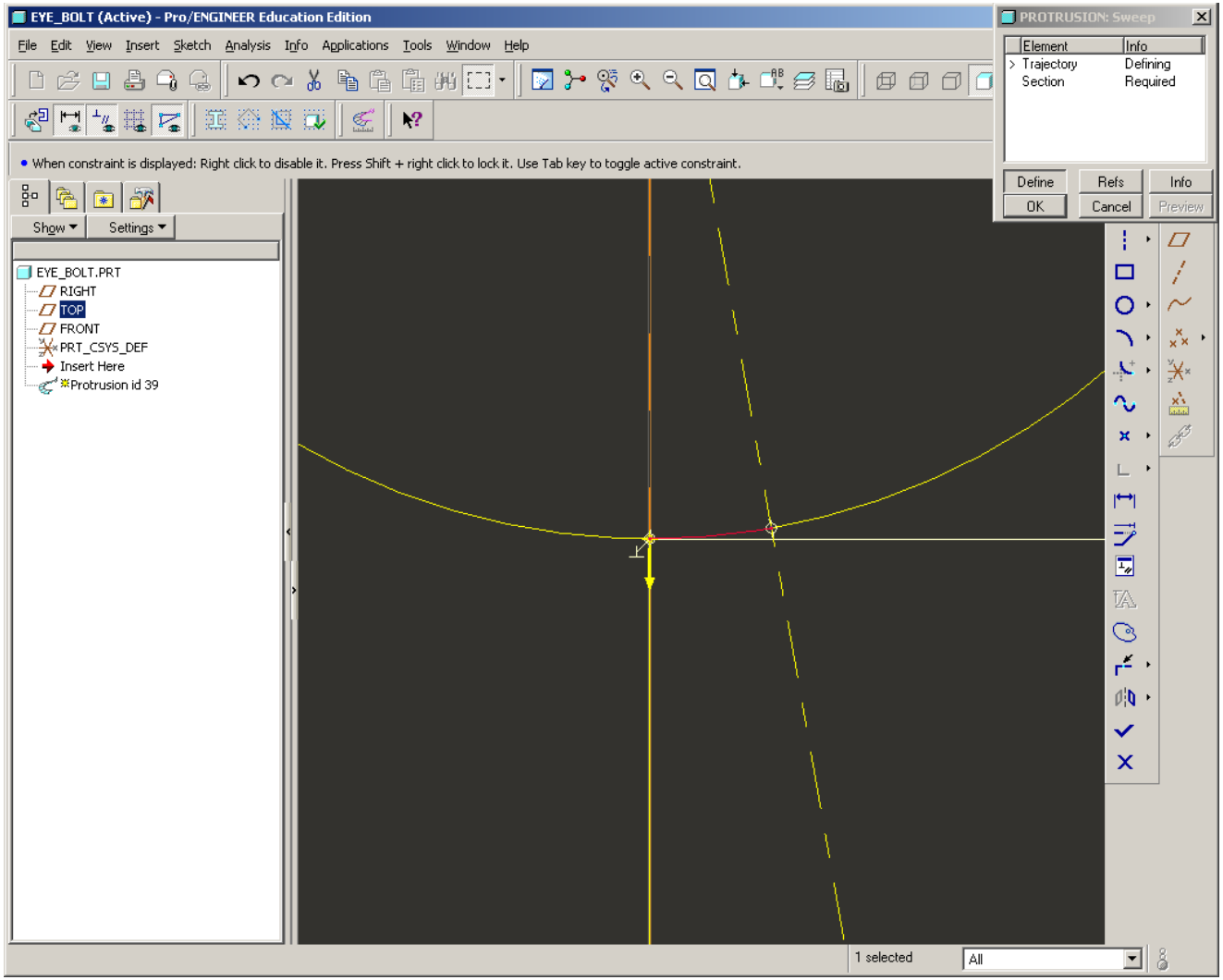
EXERCISE – Sweep

- Sketch a Centerline at 80deg angle to datum
- Divide circle at intersection with Datum and Centerline



EXERCISE – Sweep

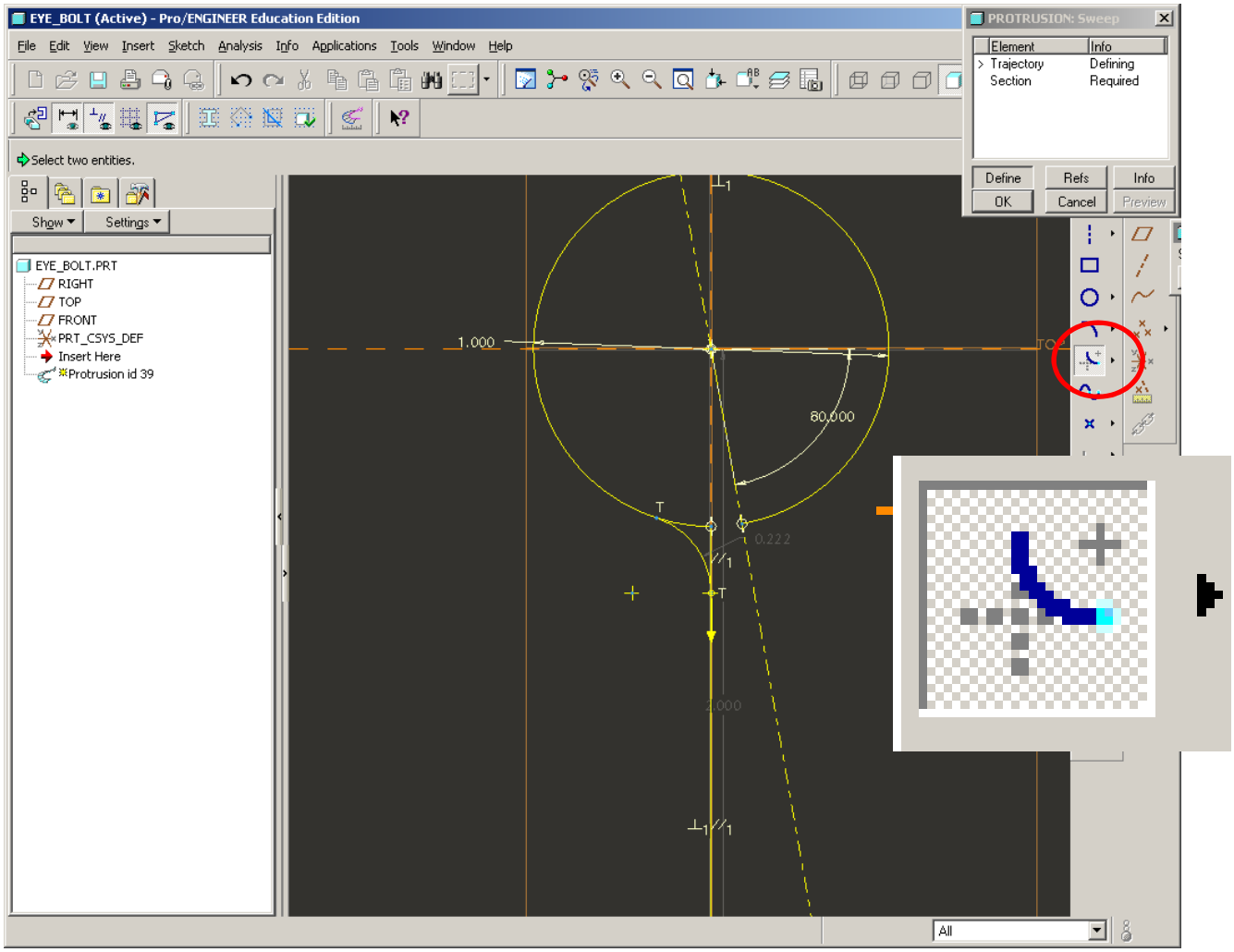
- Delete short section of circle



EXERCISE – Sweep

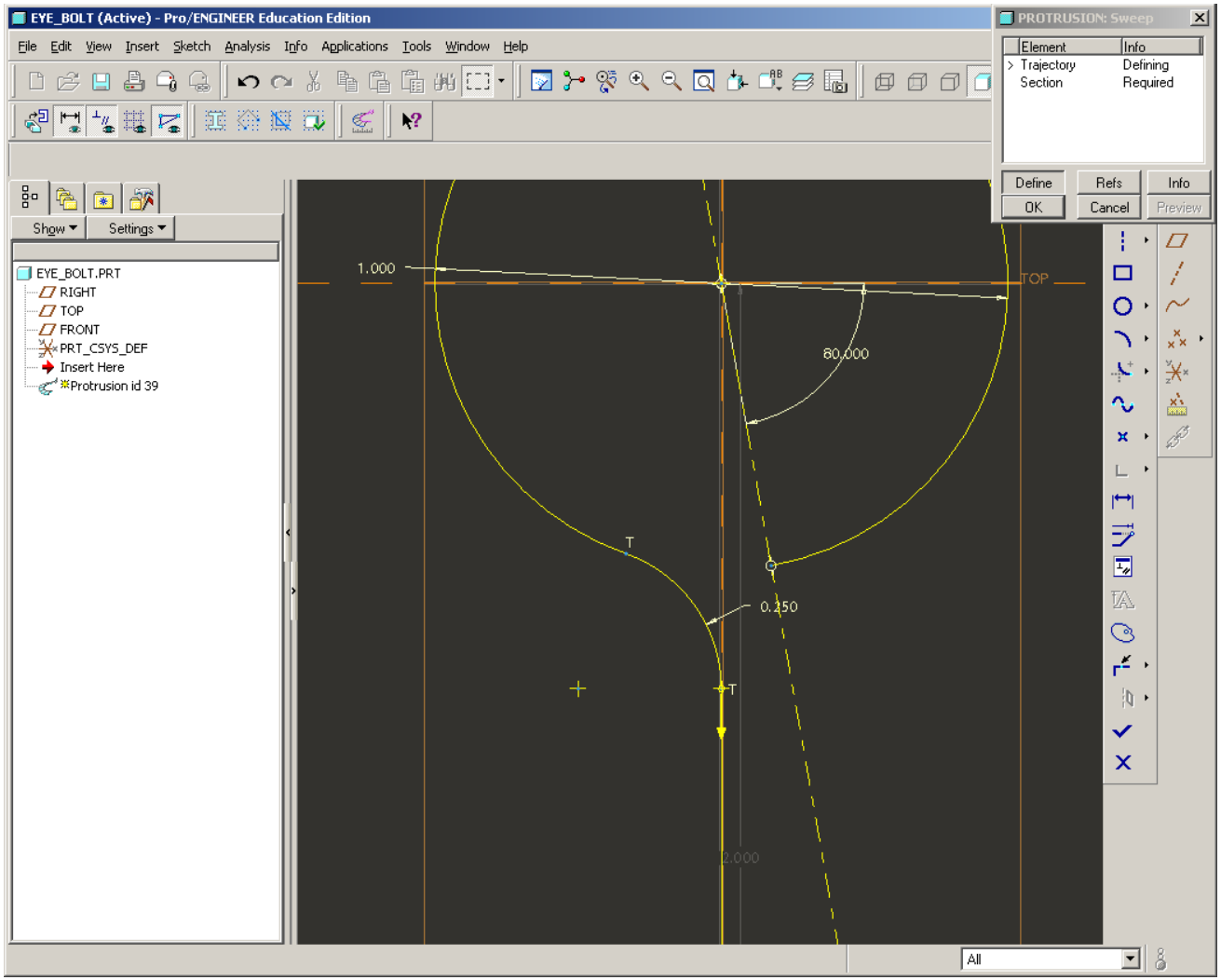
•Create a fillet

between circle and leg

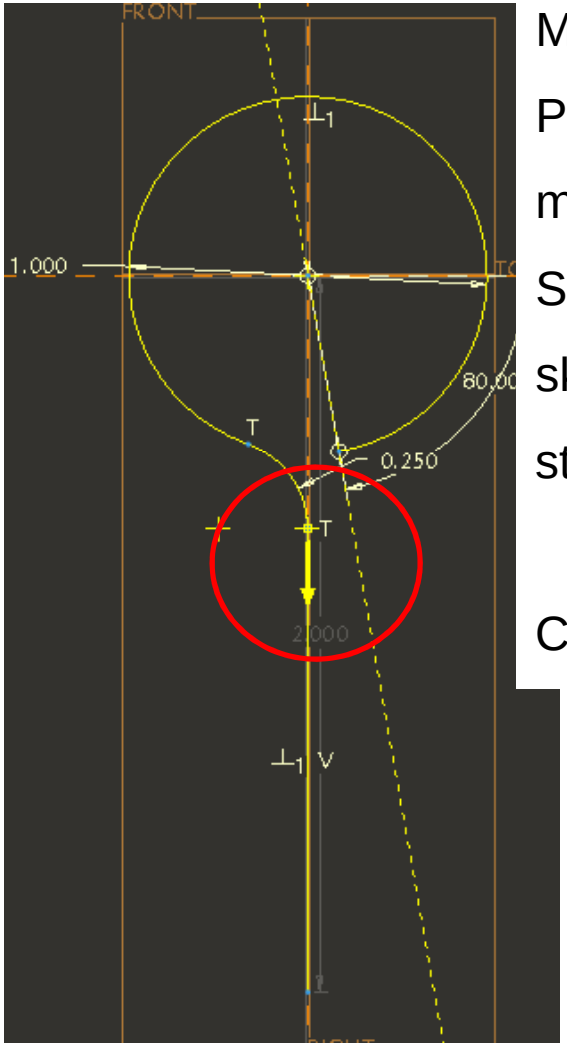


EXERCISE – Sweep

- Create a fillet between circle and leg
- Resize to .250in
- Delete short pieces



EXERCISE – Sweep

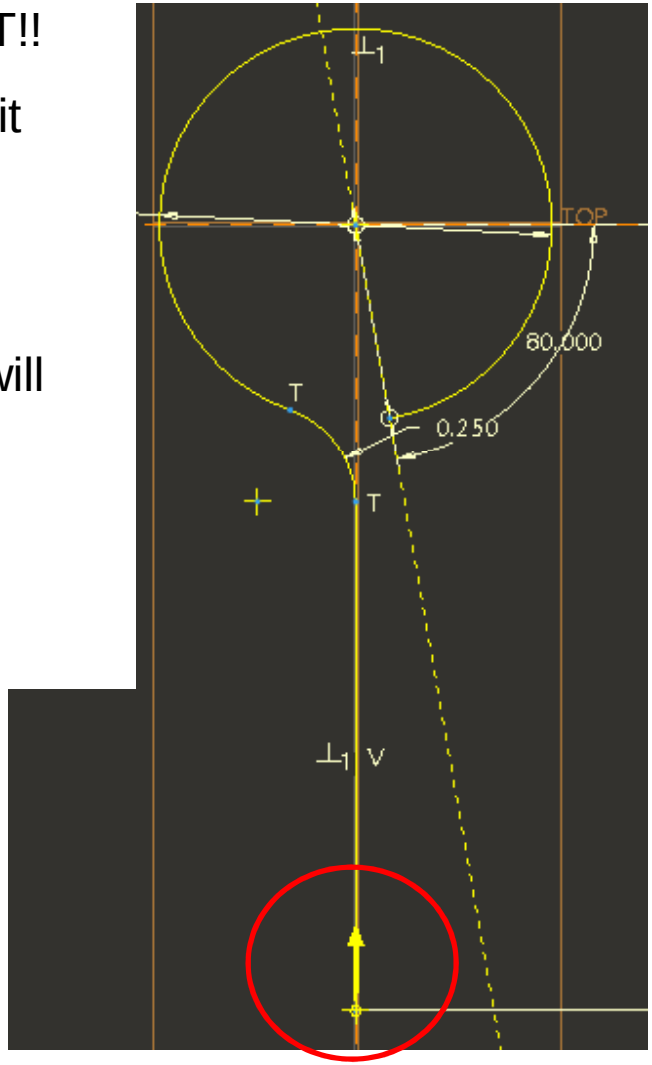


MOST IMPORTANT PART!!

Put the Start Point where it makes sense.

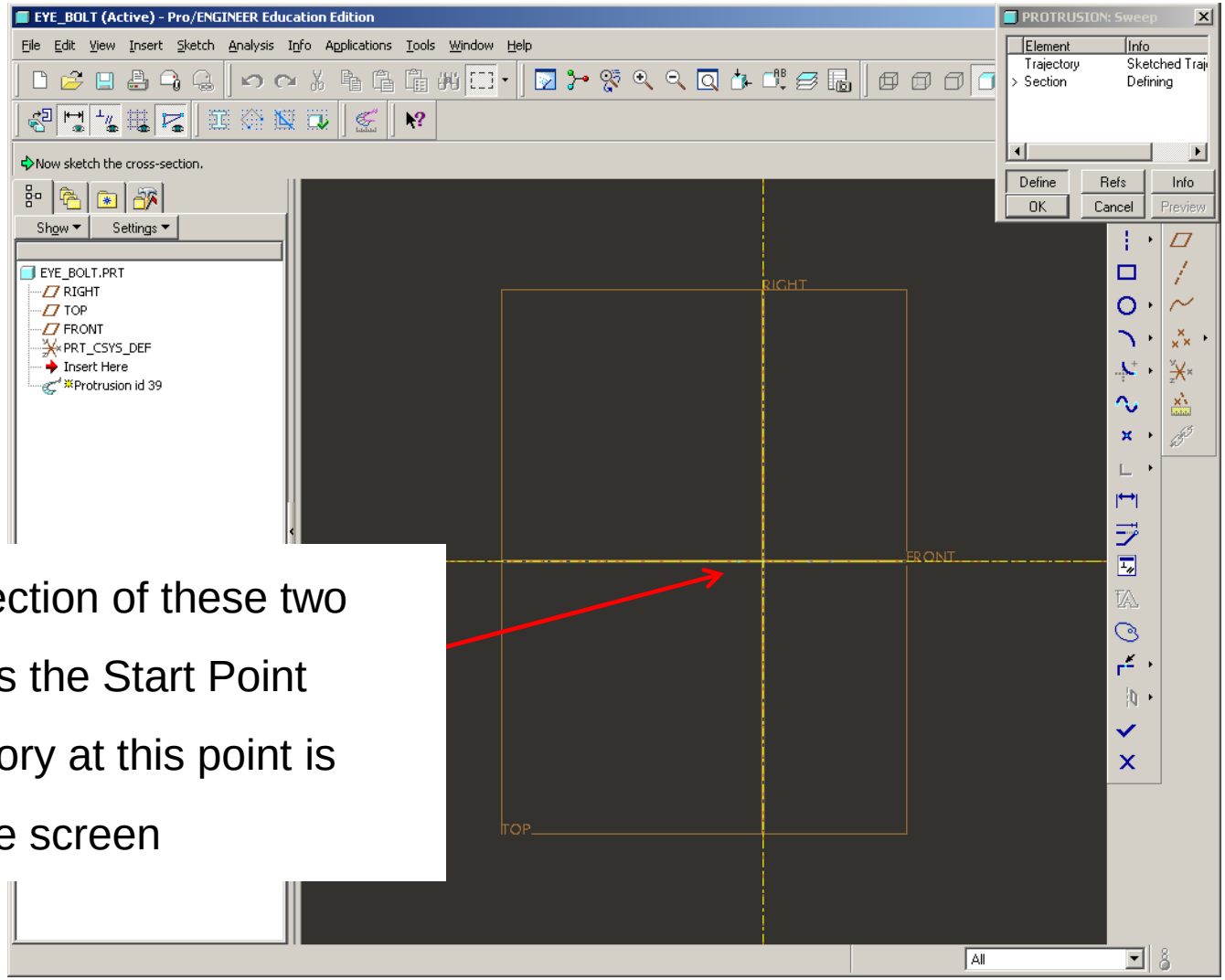
Select an endpoint on the sketch where the sweep will start > RMB > Start Point

Click Done on the sketch



EXERCISE – Sweep

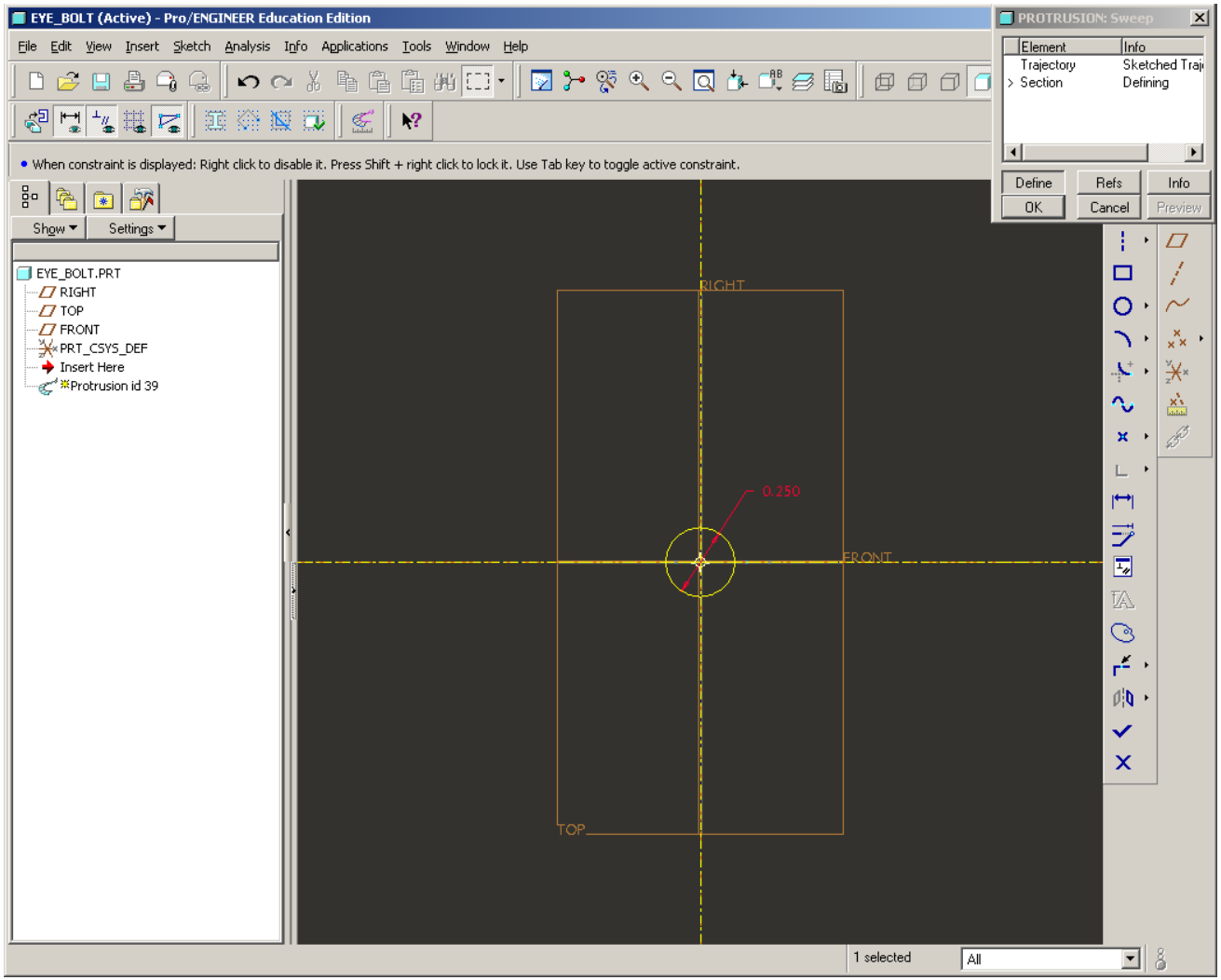
- Now sketch the parts cross section



- The intersection of these two centerlines is the Start Point
- The trajectory at this point is normal to the screen

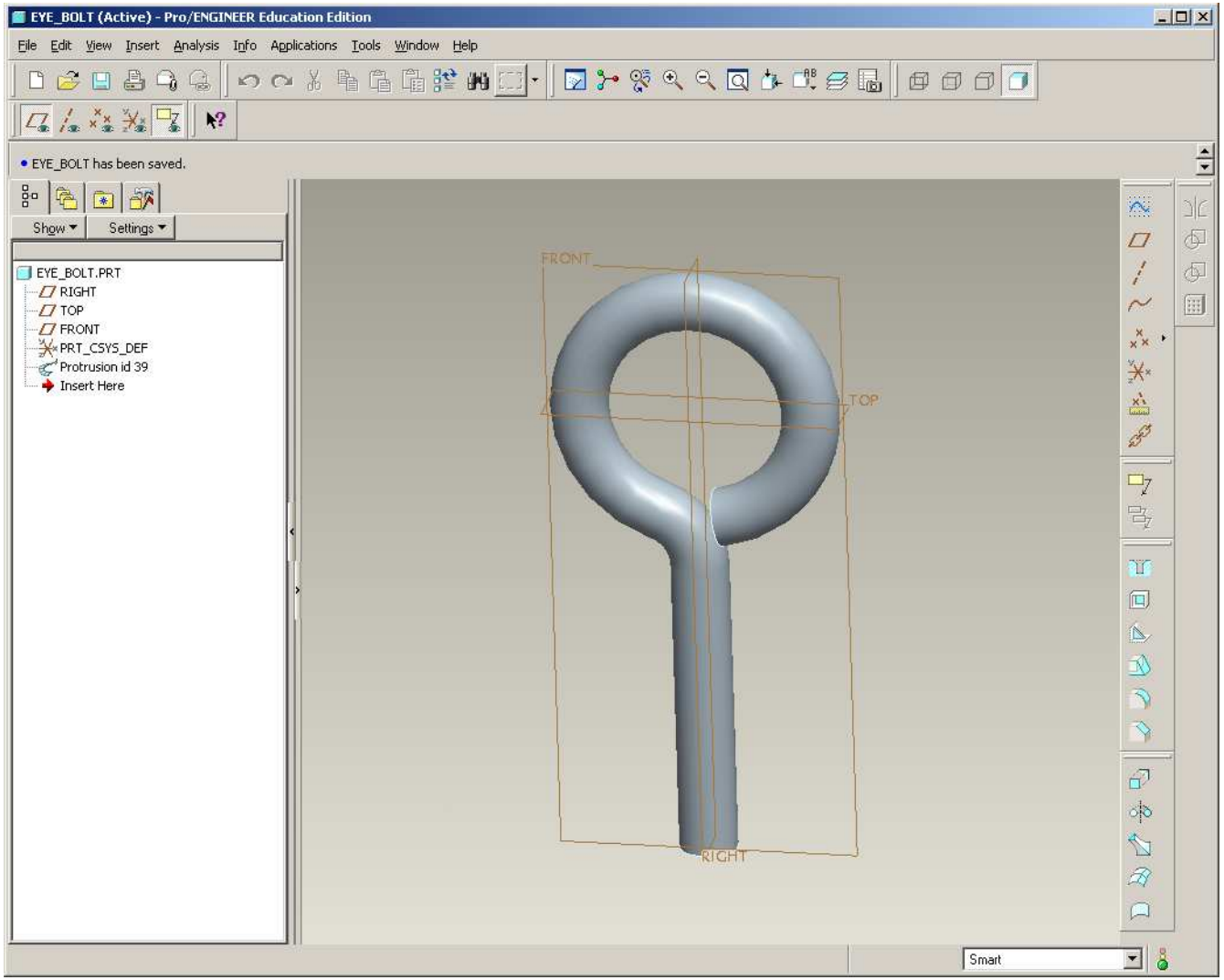
EXERCISE – Sweep

- Sketch a .250in circle at the intersection
- Click Done on the sketch
- Click OK on the dialog box



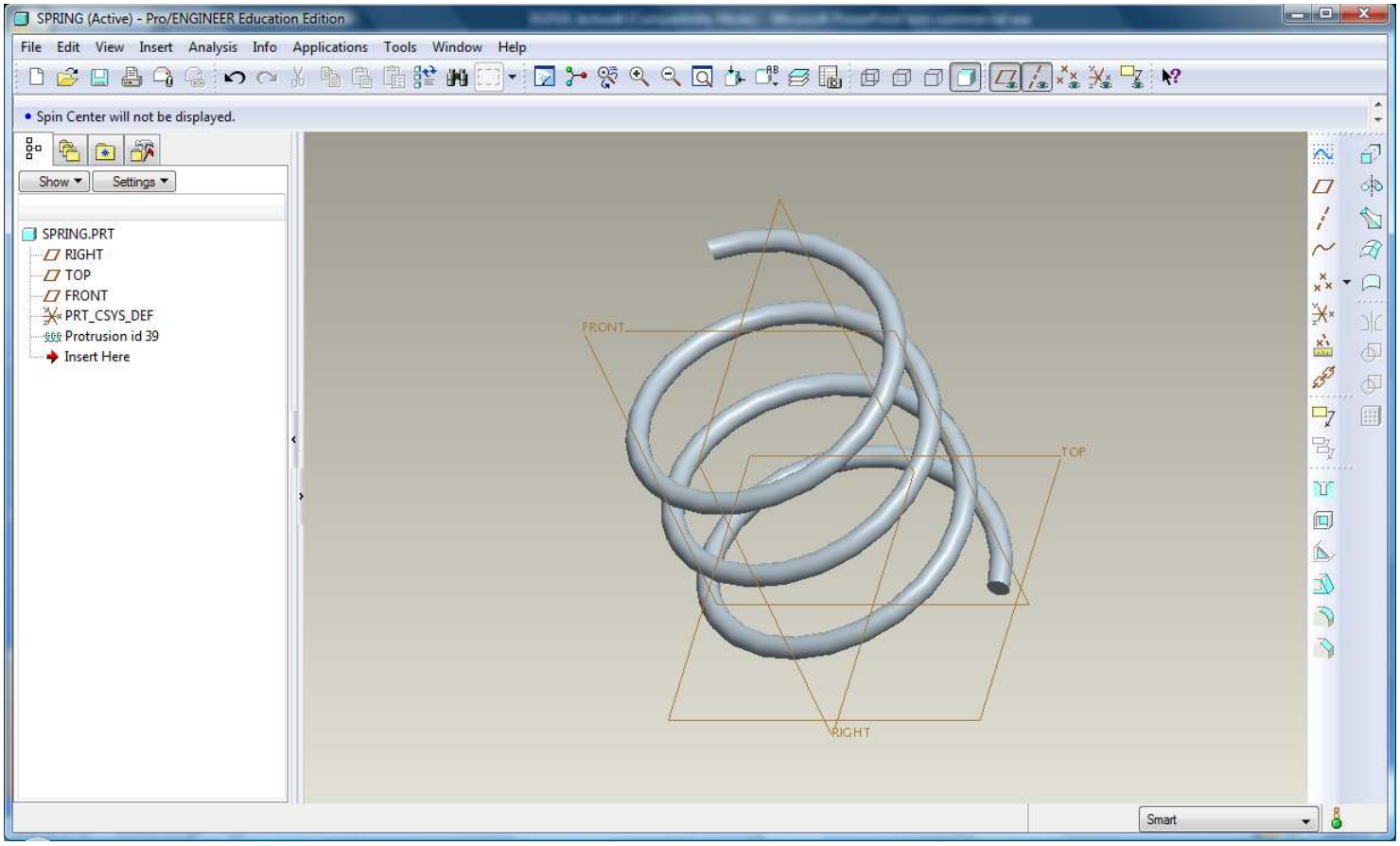
EXERCISE – Sweep

- Rotate to view final geometry



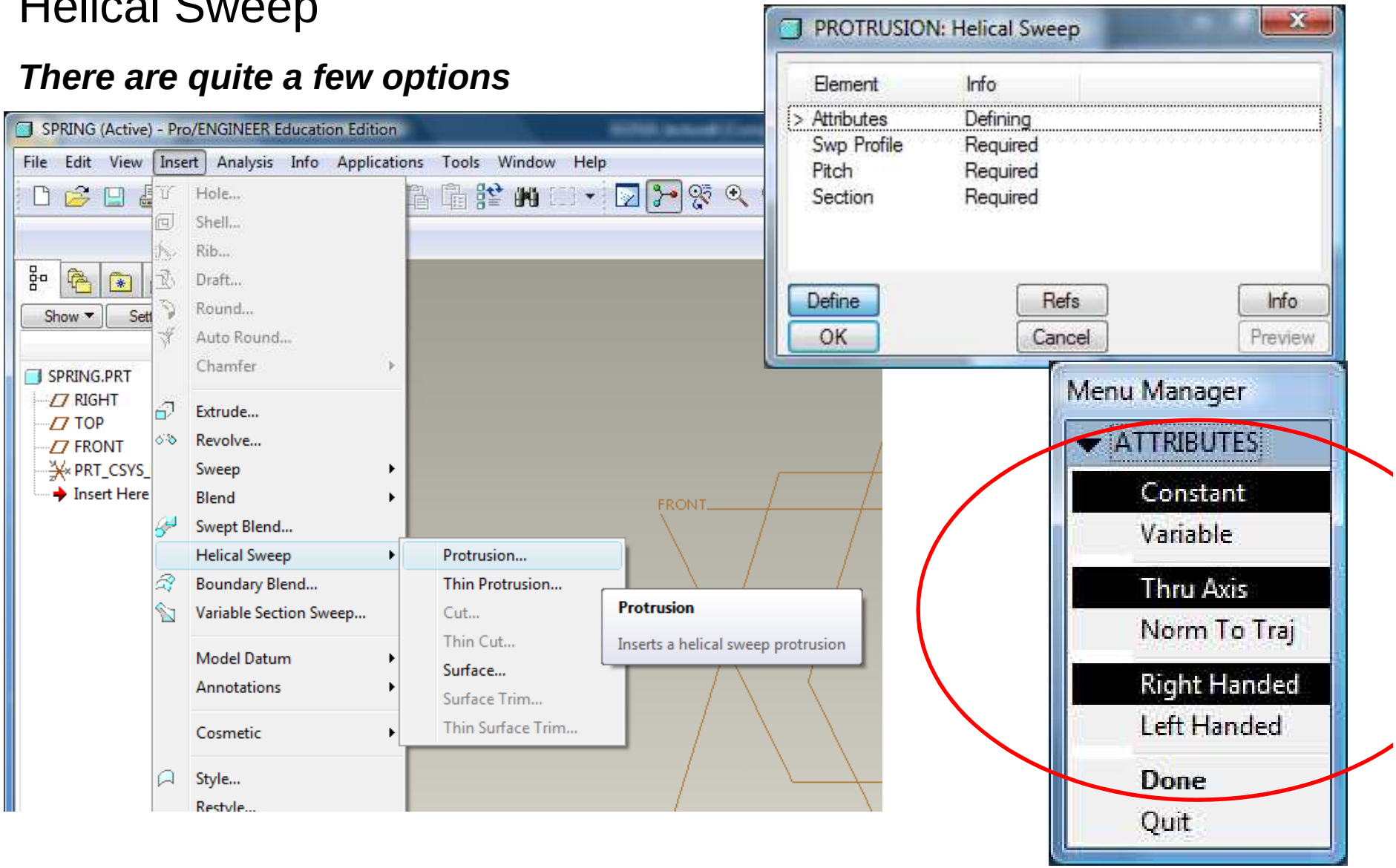
Helical Sweep

Sweep seems like the right way to make a spring, but how do I sketch the trajectory? Pro/E does it for you….



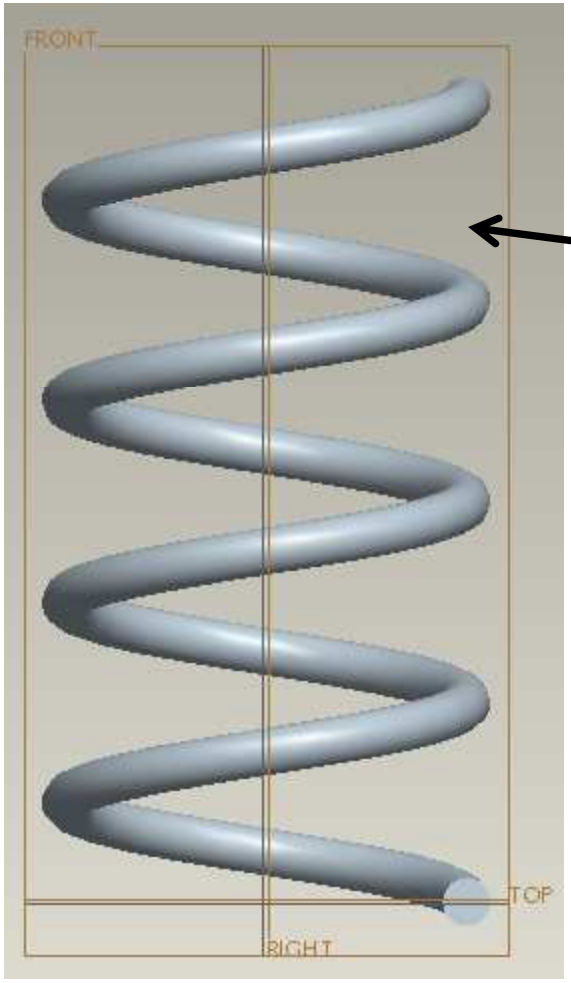
Helical Sweep

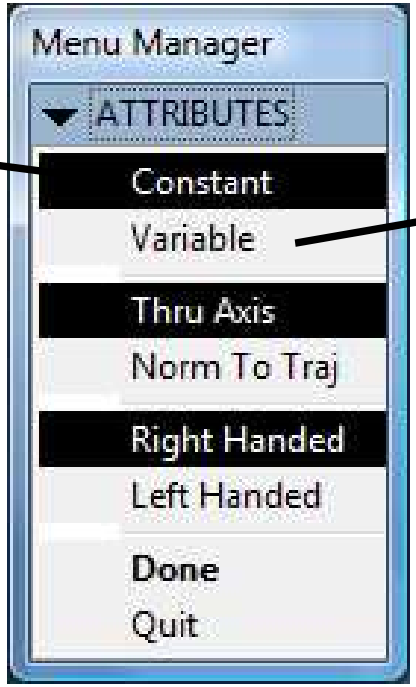
There are quite a few options

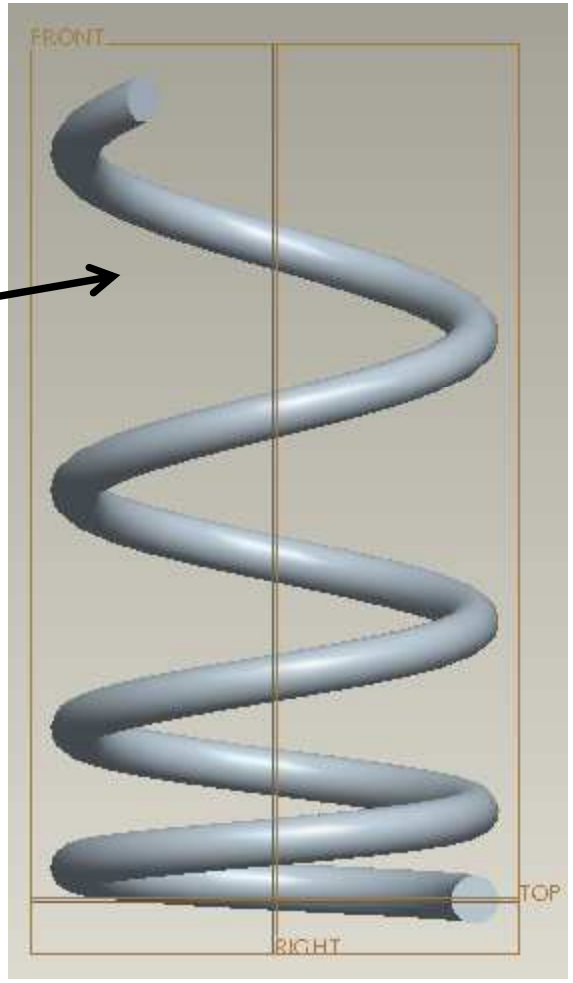


Helical Sweep

Pitch – Inches per Turn

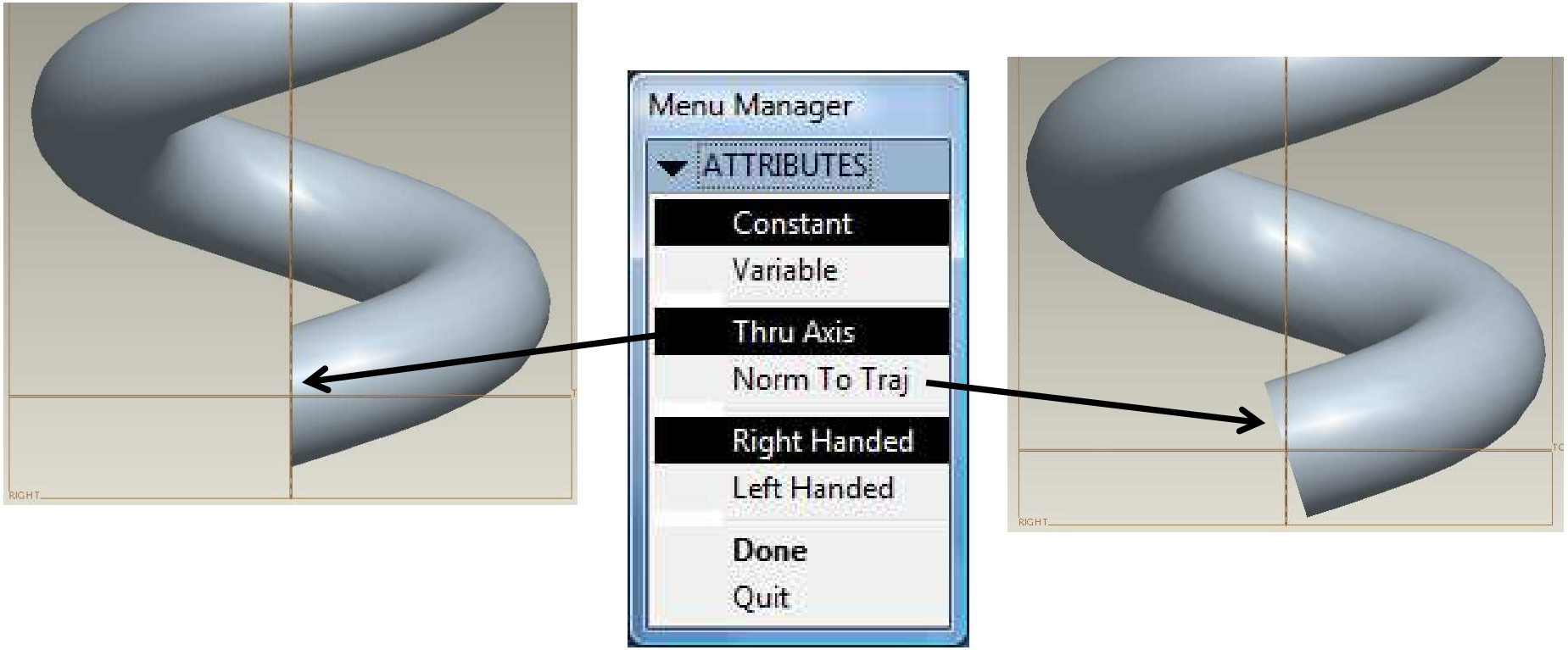






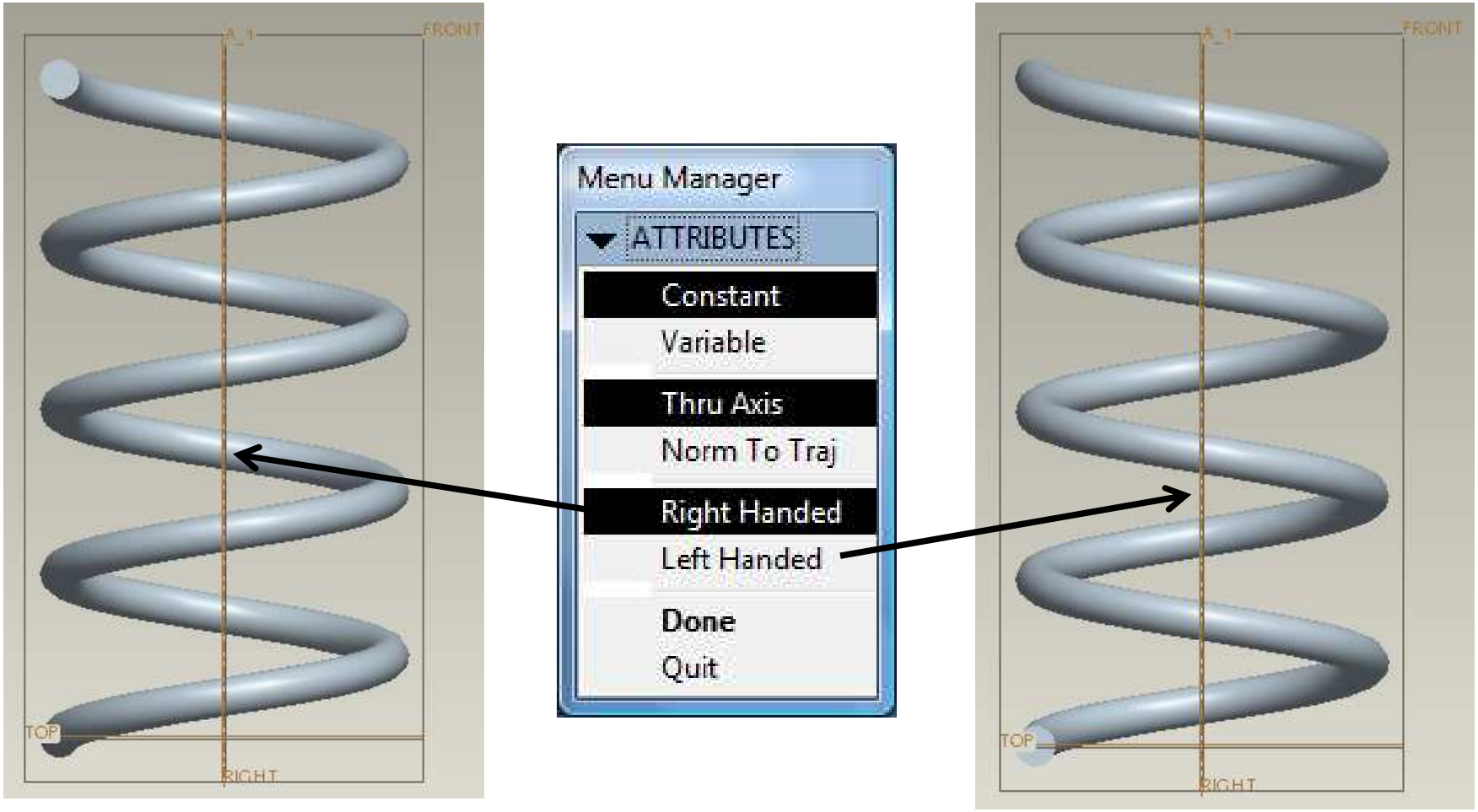
Helical Sweep

Section Alignment



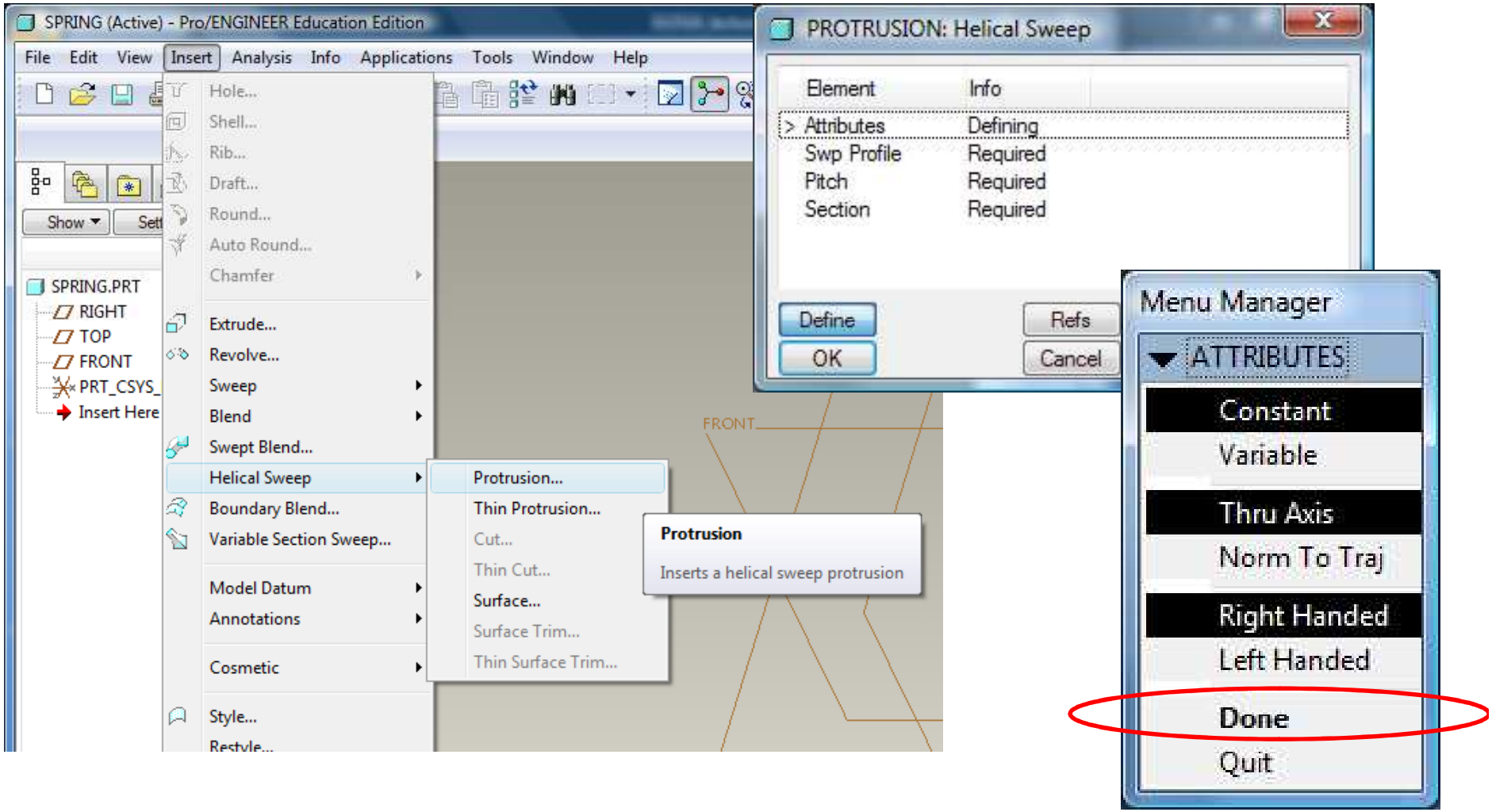
Helical Sweep

Helix Direction



EXERCISE - Helical Sweep

Let's make a basic square-wire compression spring



EXERCISE - Helical Sweep

In the same way we set up the sketch plane for Sweep Trajectory, establish a sketch plane

Step 1 – Select sketch plane

FRONT plane for this exercise

Step 2 – Accept or Flip viewing direction

ACCEPT for this exercise

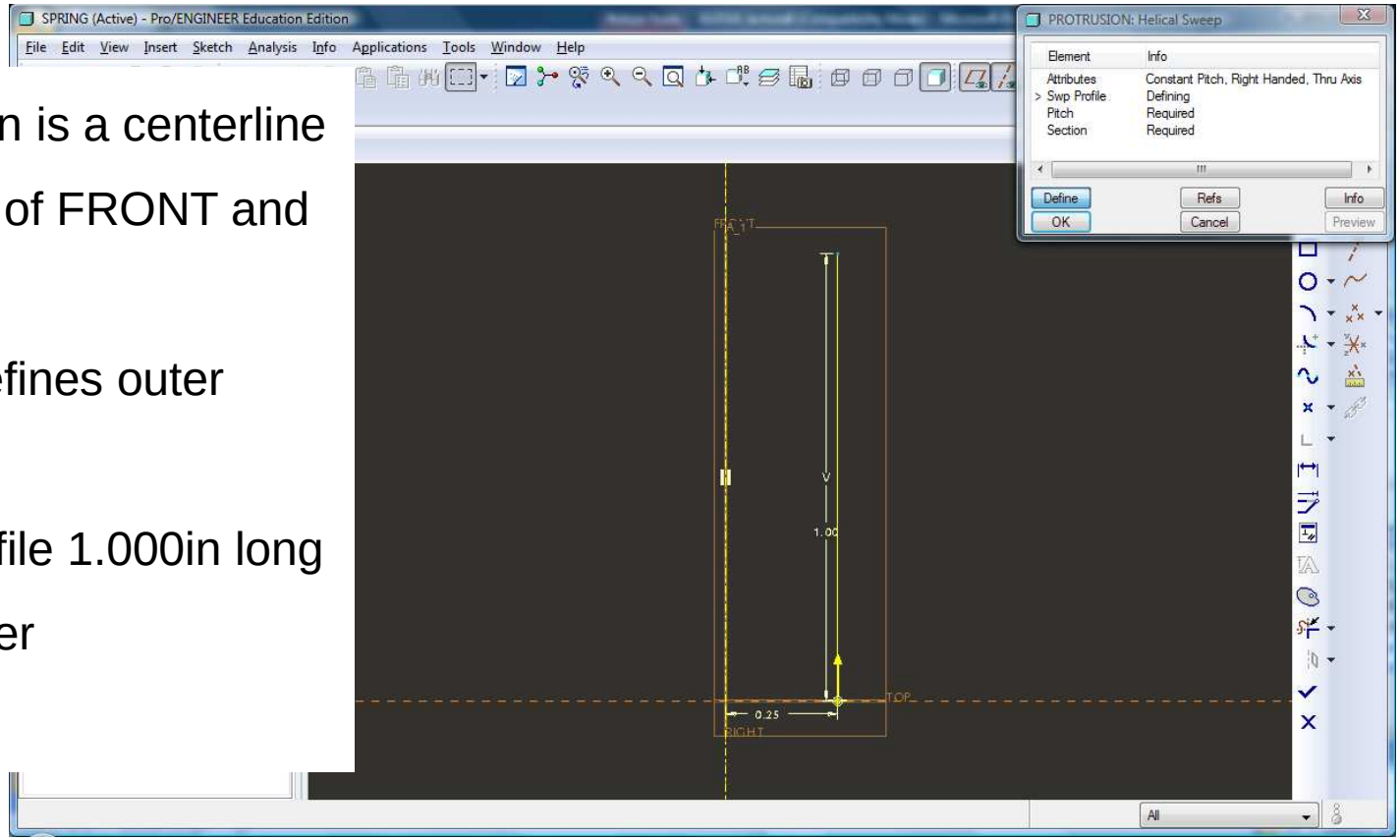
Step 3 – Select reference plane

Use TOP datum as Top reference for this exercise

EXERCISE - Helical Sweep

Sketch an Axis of Revolution and Sweep Profile.

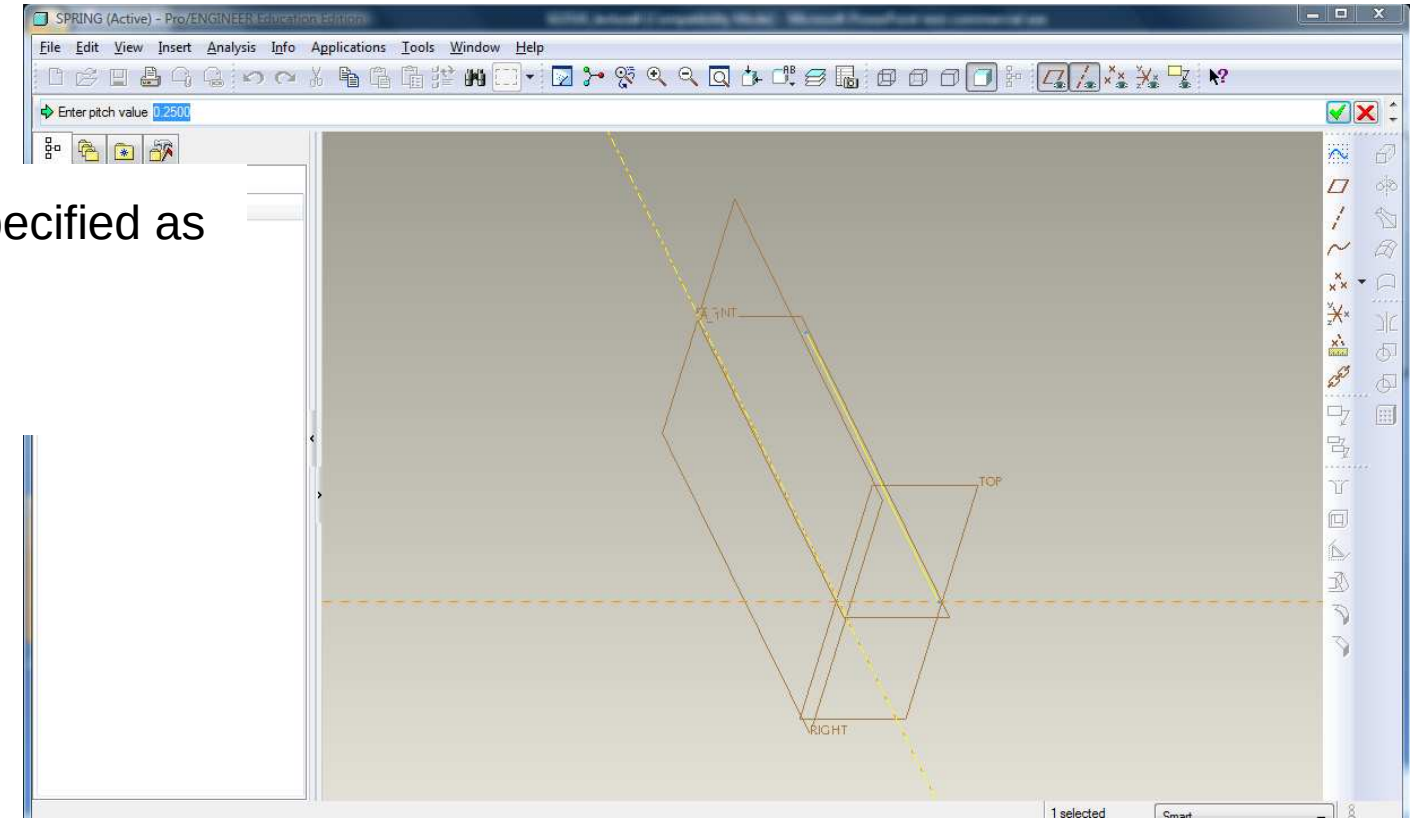
- Axis of Revolution is a centerline at the intersection of FRONT and RIGHT datums
- Sweep Profile defines outer bounds of sweep
- Make sweep profile 1.000in long and .500in diameter
- Click Done



EXERCISE - Helical Sweep

Specify Pitch

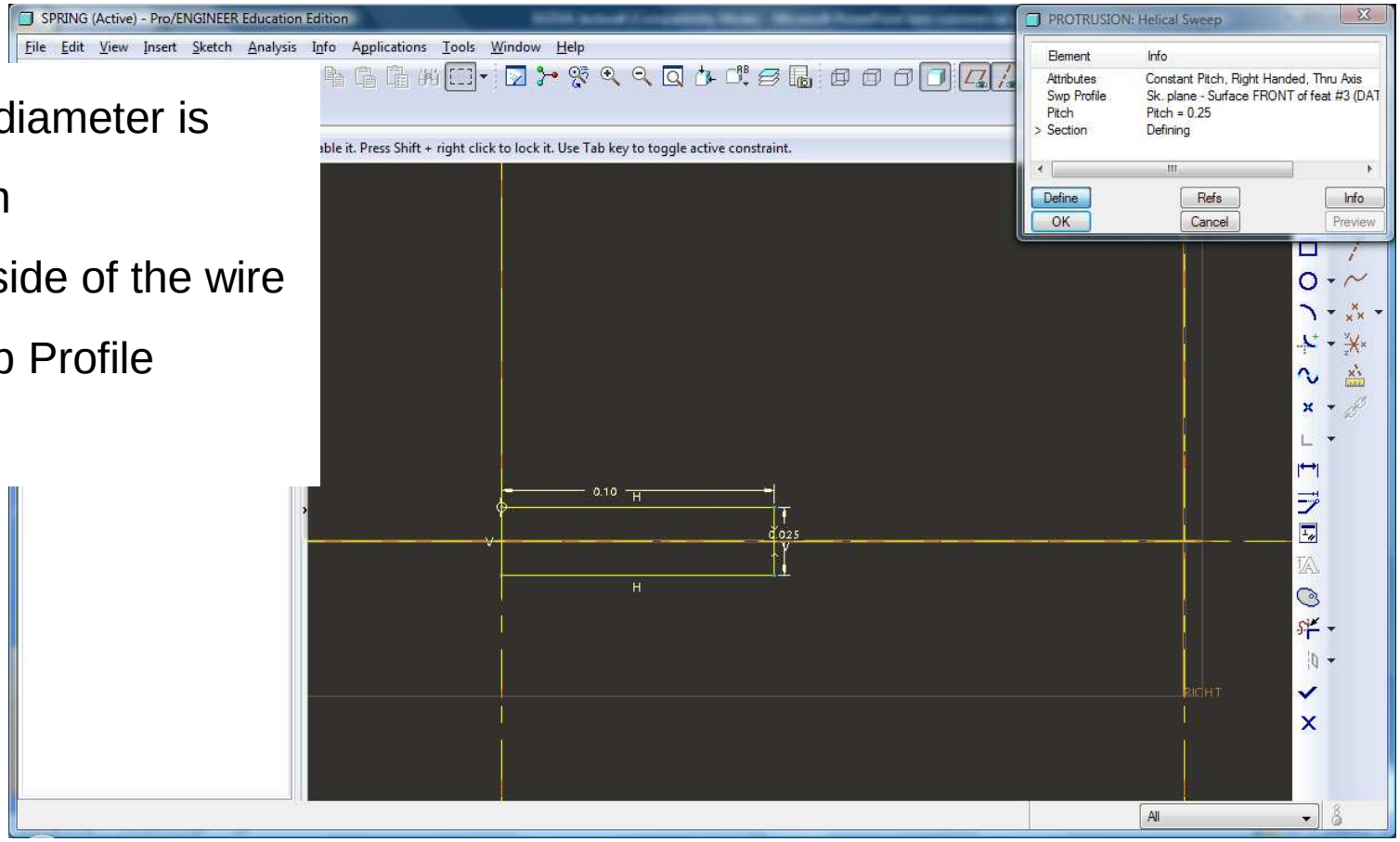
- NOTE: Pitch is specified as length per turn.
- Enter .25in/turn.



EXERCISE - Helical Sweep

Sketch X-Section of spring wire.

- Square wire diameter is .100in X .025in
- Align the outside of the wire with the Sweep Profile
- Click Done



EXERCISE - Helical Sweep

Done.