EN1740 Computer Aided Visualization and Design

Spring 2012

1/31/2012

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PLEASE WAIT TO LAUNCH PRO/E

IF ALREADY OPENED, PLEASE CLOSE

PLEASE WAIT TO LAUNCH PRO/E

PLEASE CLOSE IF ALREADY OPENED

Last time:

Tonight:

- …*when, where, why, who*, *what?*
- Syllabus
- Projects
- Assignment 1
- Intro to Pro/Engineer
  - How to get around
  - File management

Tonight:

- Creating models in Pro/Engineer:
  - Customizing and formatting the Pro/E environment
  - Introduction to Datums
  - Sketcher
  - Solid feature creation
    - Extrude
    - Revolve
- ***There's going to be a lot of click-along***

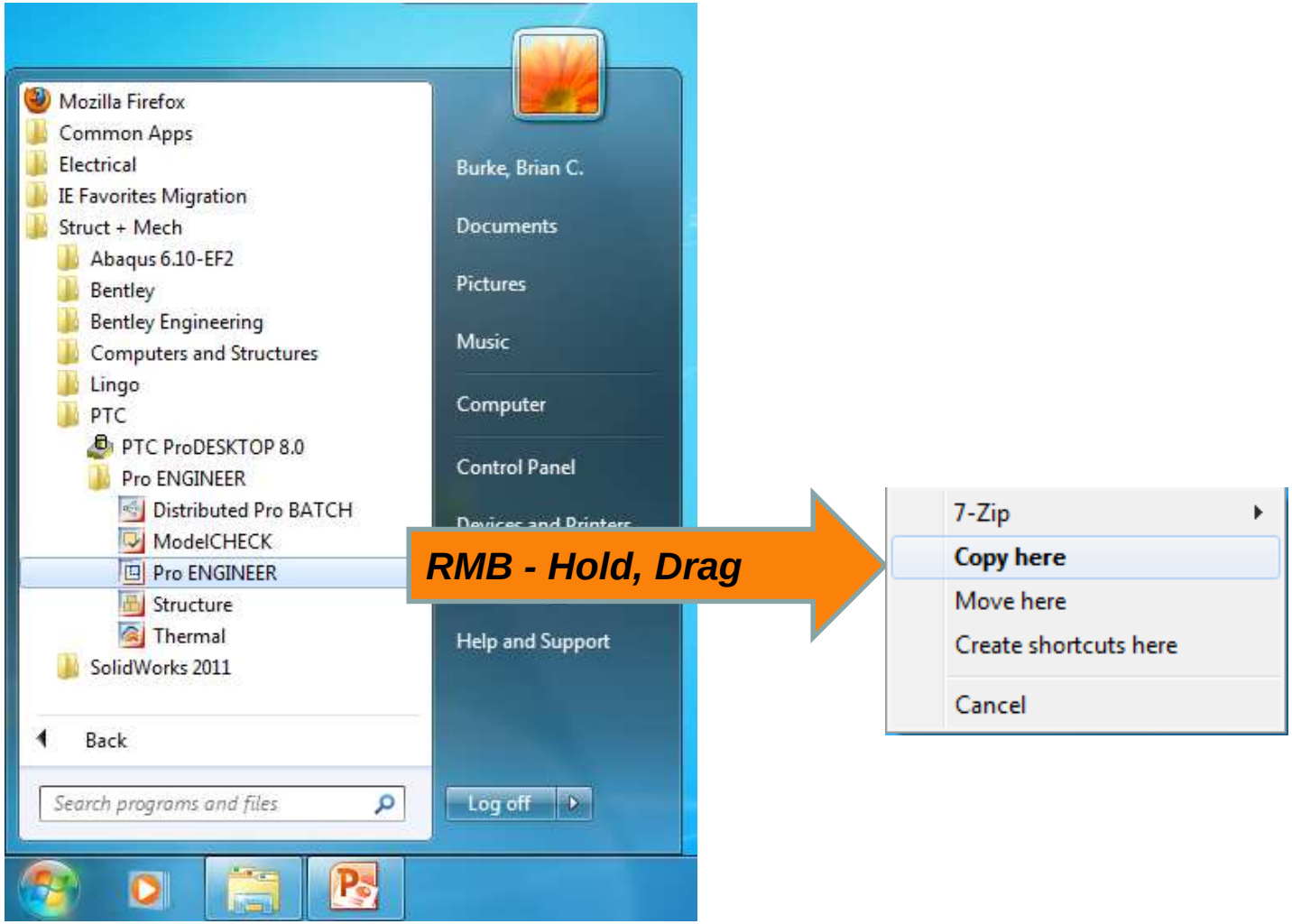
Key TLA's

- *LMB* – Left mouse button
- *MMB* – Middle mouse button
- *RMB* – Right mouse button

Standardizing and customizing Pro/E

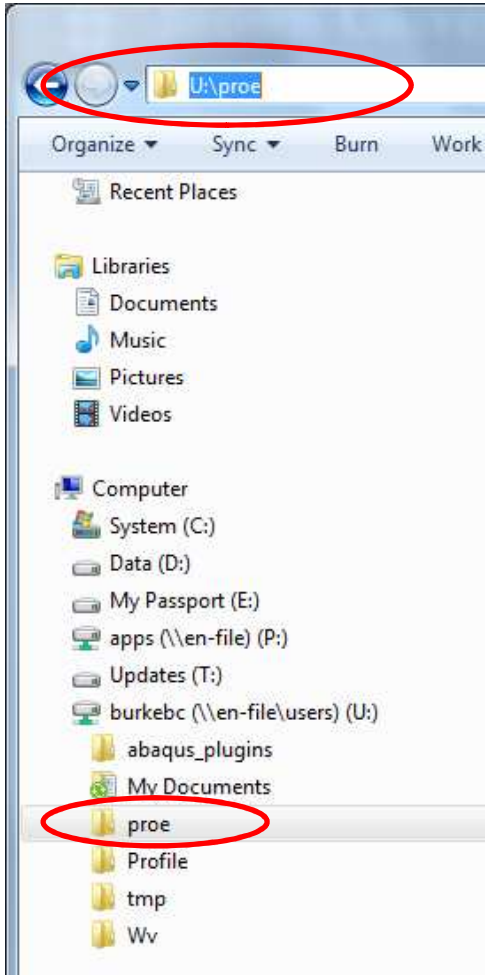
- Many aspects of Pro/Engineer can be customized
  - Decimal places, system colors, default templates, etc.
  - These can be ***BIG*** time savers
- User settings are stored in a series of files
  - config.pro – General preferences and “mapkeys” (macros)
  - config.win.X – Pro/Engineer window settings
  - drawing.dtl – Detailed drawing parameters
  - tree.cfg – Model tree configurations
- All of these files are read out of the launch directory at startup
- ***We need to create and specify this directory to set preferences***

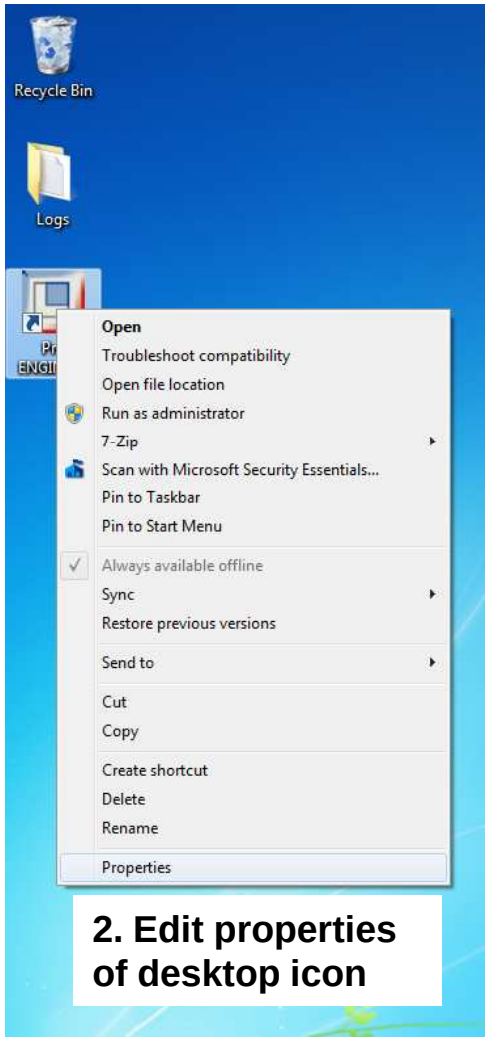
EXERCISE - Create a desktop icon for Pro/E

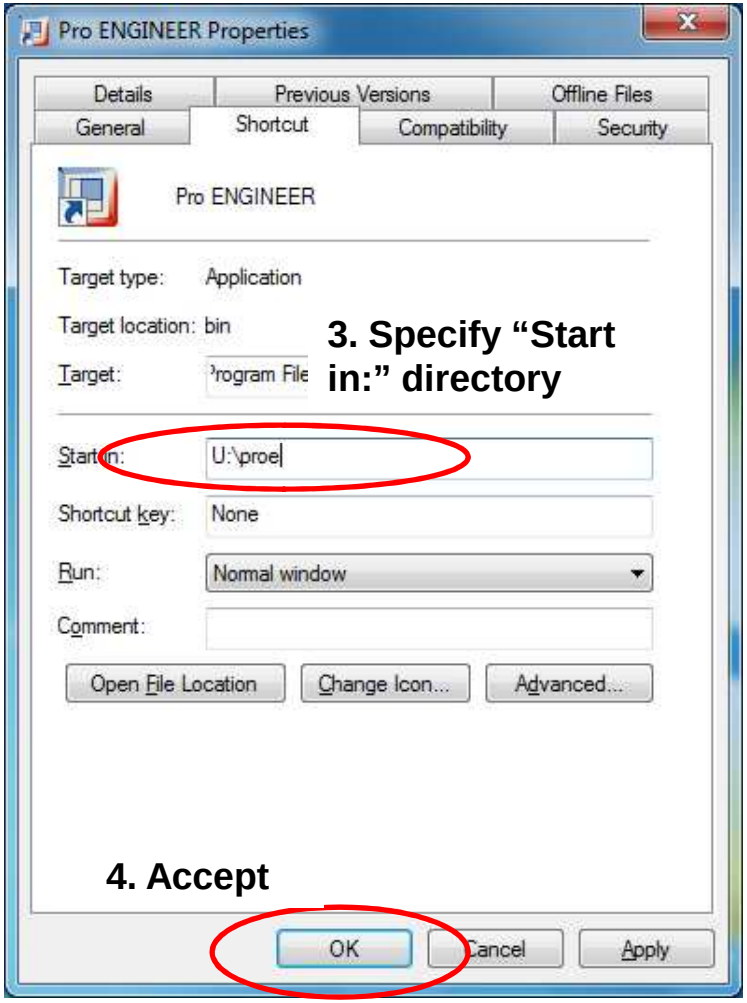


EXERCISE - Specify launch directory

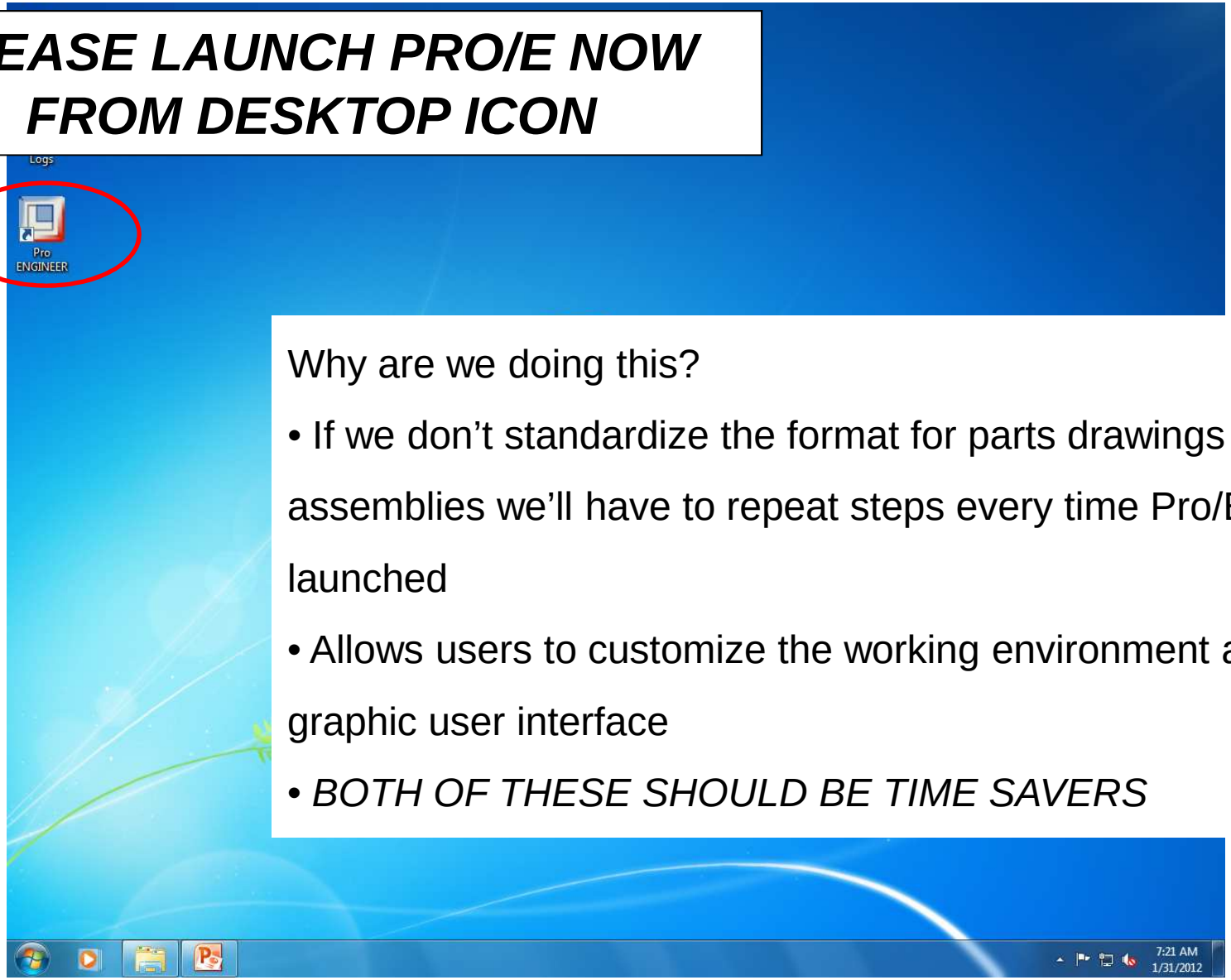
1. Make sure directory exists







PLEASE LAUNCH PRO/E NOW FROM DESKTOP ICON

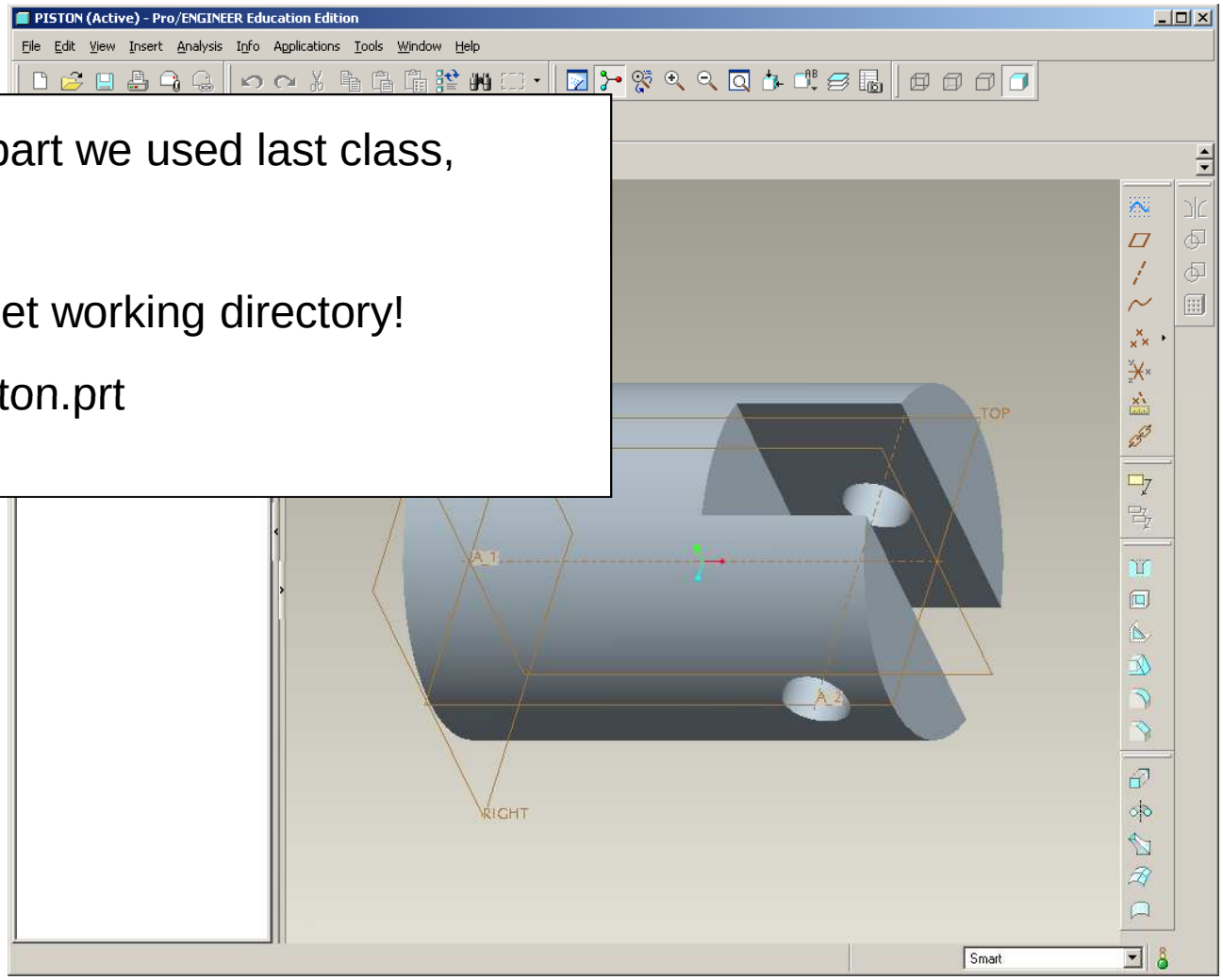


Why are we doing this?

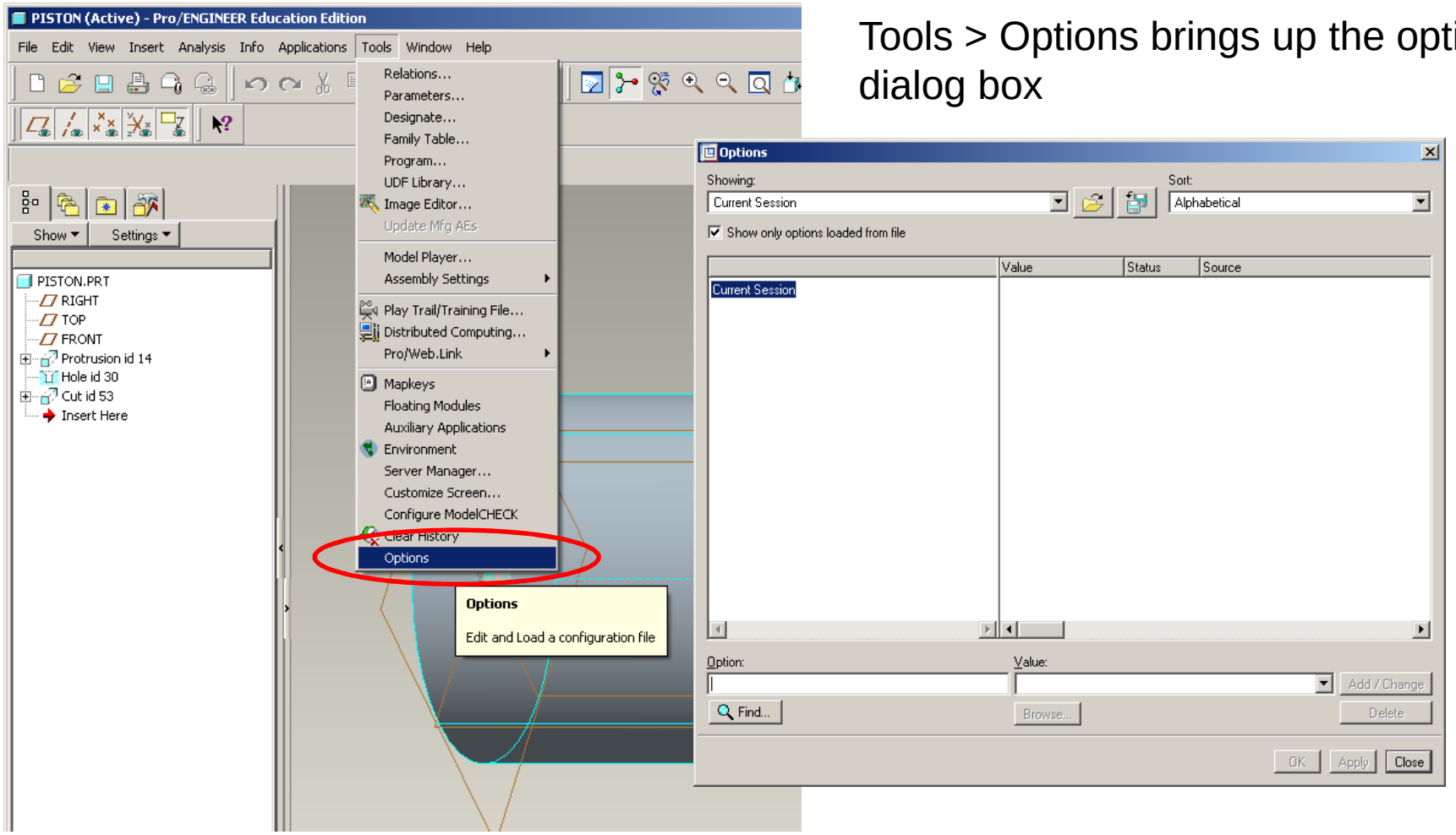
- If we don't standardize the format for parts drawings and assemblies we'll have to repeat steps every time Pro/E is launched
- Allows users to customize the working environment and graphic user interface
- *BOTH OF THESE SHOULD BE TIME SAVERS*

EXERCISE - Specifying configuration options

- To begin, open the part we used last class, piston.prt
  - Remember to set working directory!
  - File > Open piston.prt

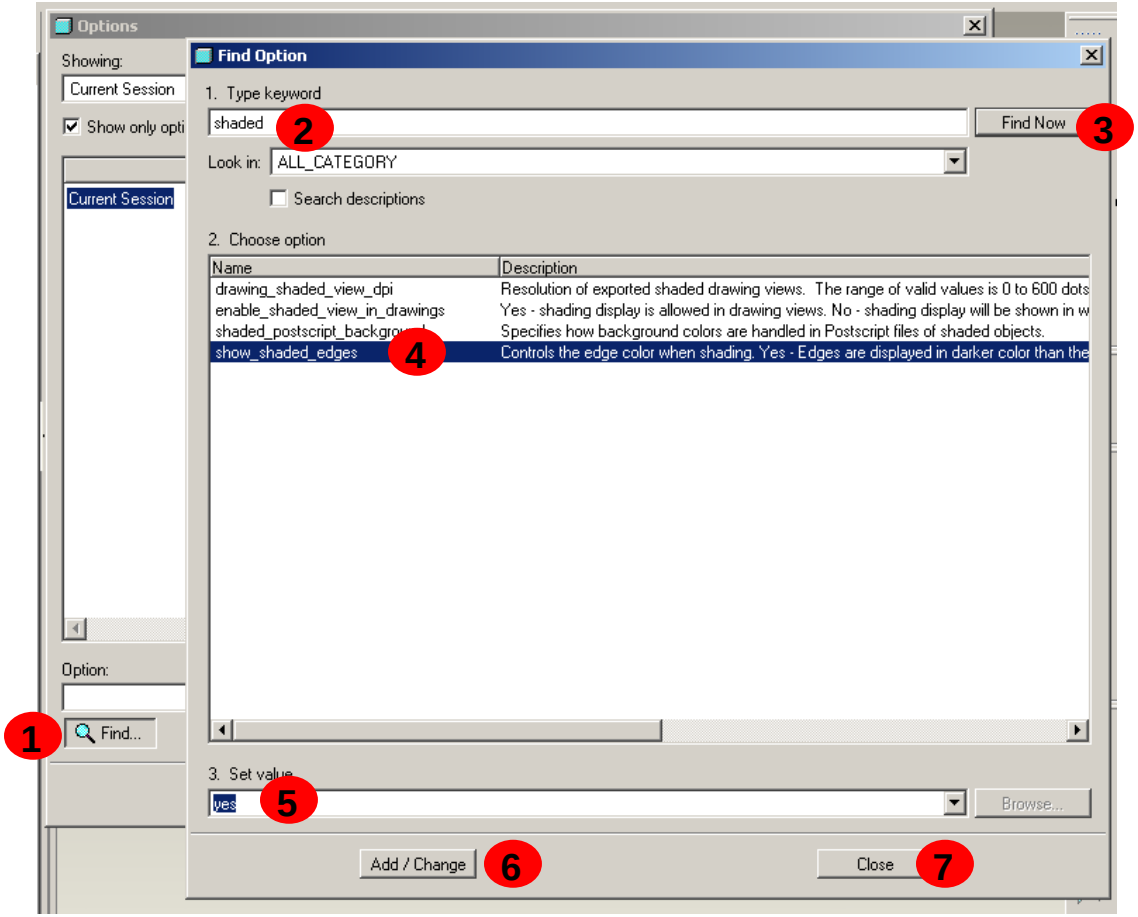


EXERCISE - Specifying configuration options



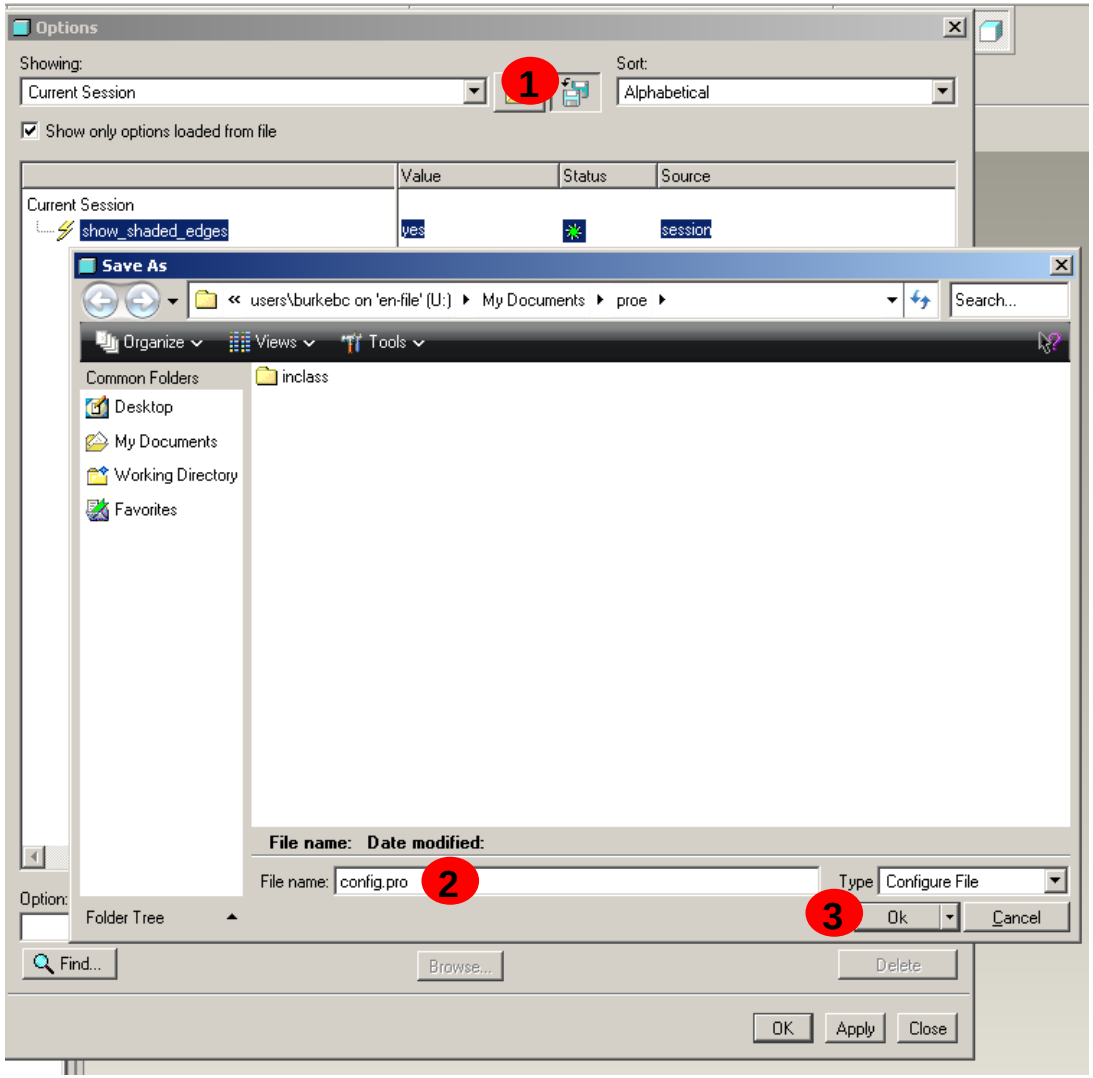
Tools > Options brings up the options dialog box

EXERCISE - Specifying configuration options



1. Click Find on Options Dialog
2. Enter “shaded”
3. Click Find Now
4. Select “show_shaded_edges”
5. Change Set Value to “yes”
6. Click Add/Change
7. Click Close

EXERCISE - Specifying configuration options

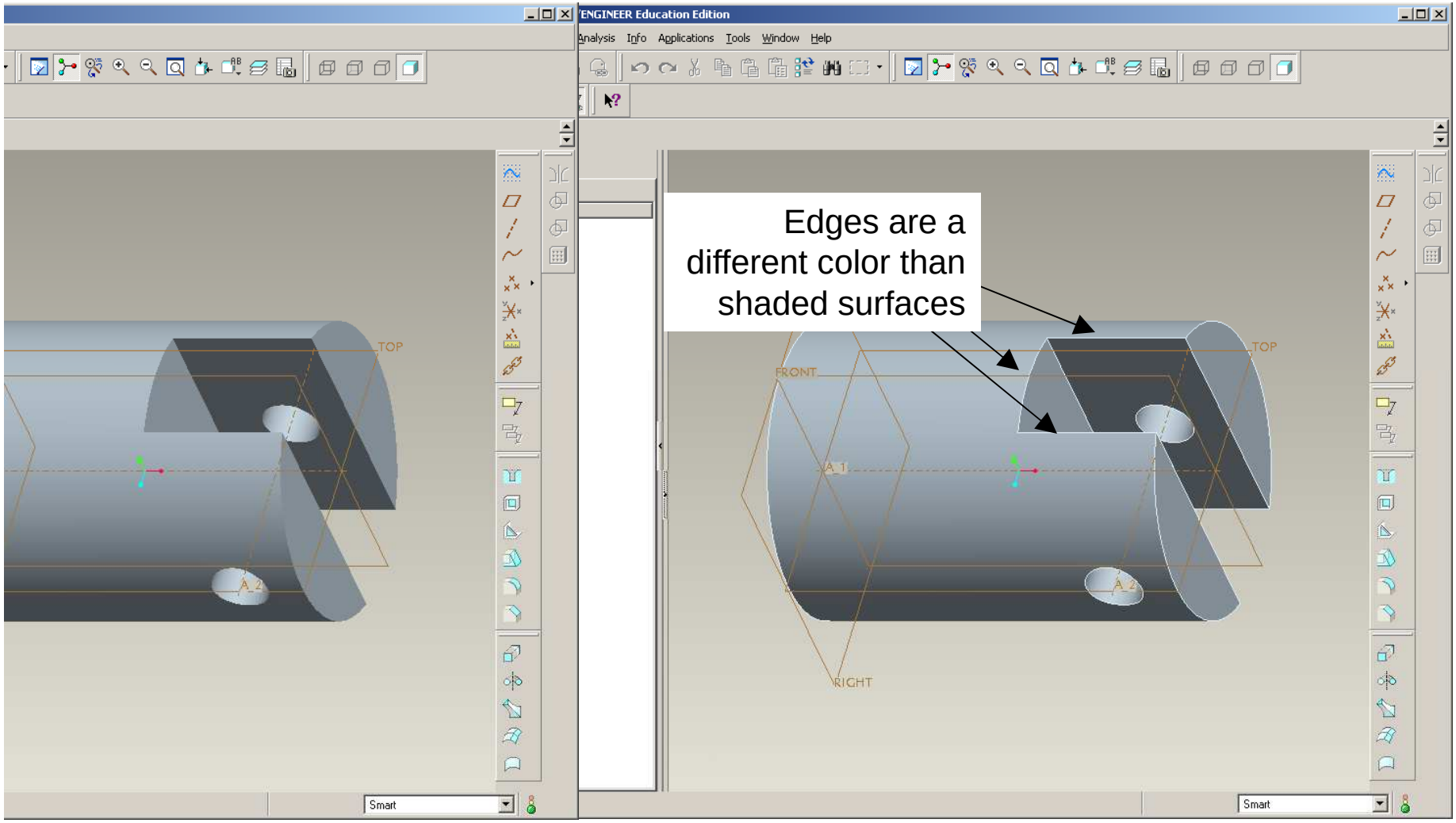


1. Back at Options Dialog, Click
2. Save a Copy Icon
3. Enter “config.pro” for file name

   Click Ok

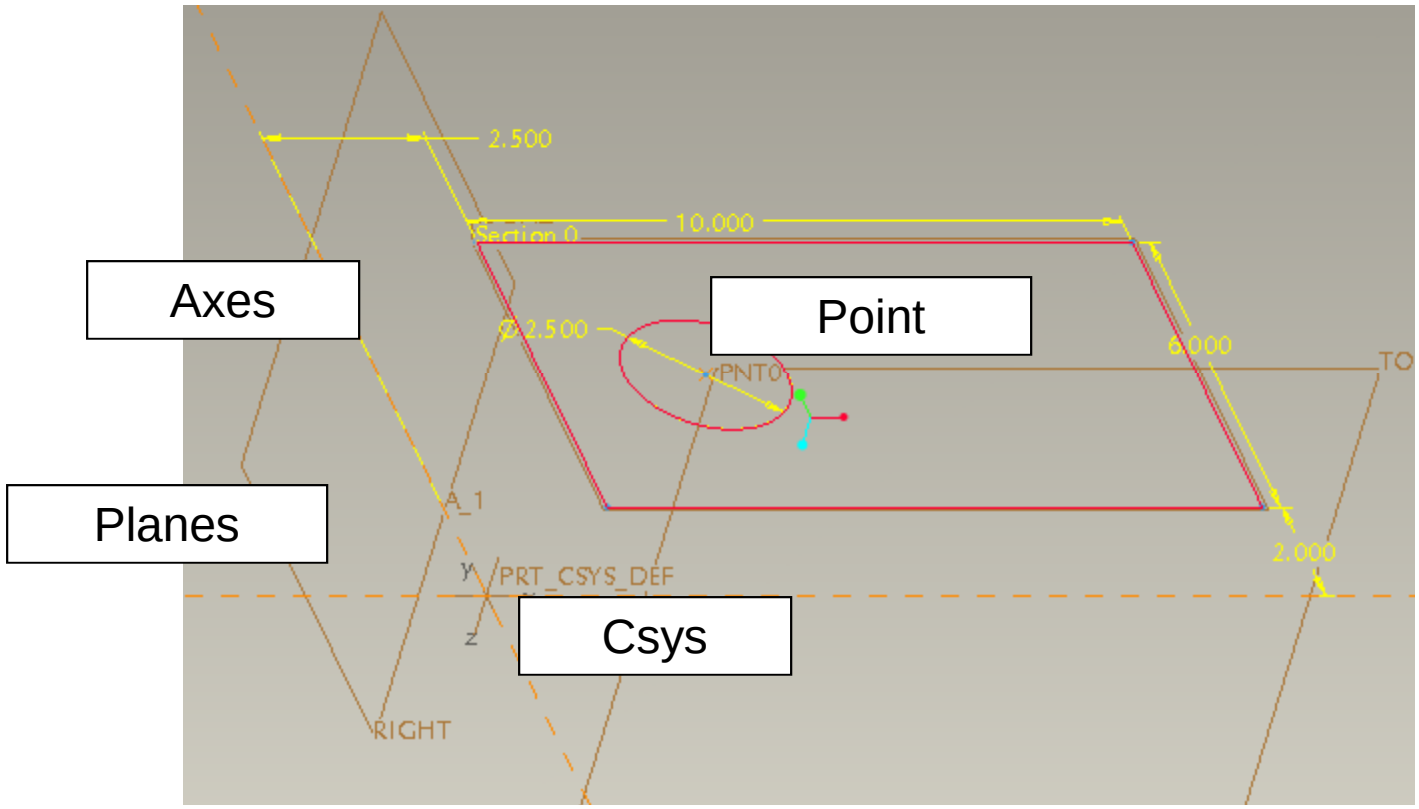
   MAKE SURE YOU’RE IN THE …\proe DIRECTORY

What did that option do?



Introduction to Datum Features

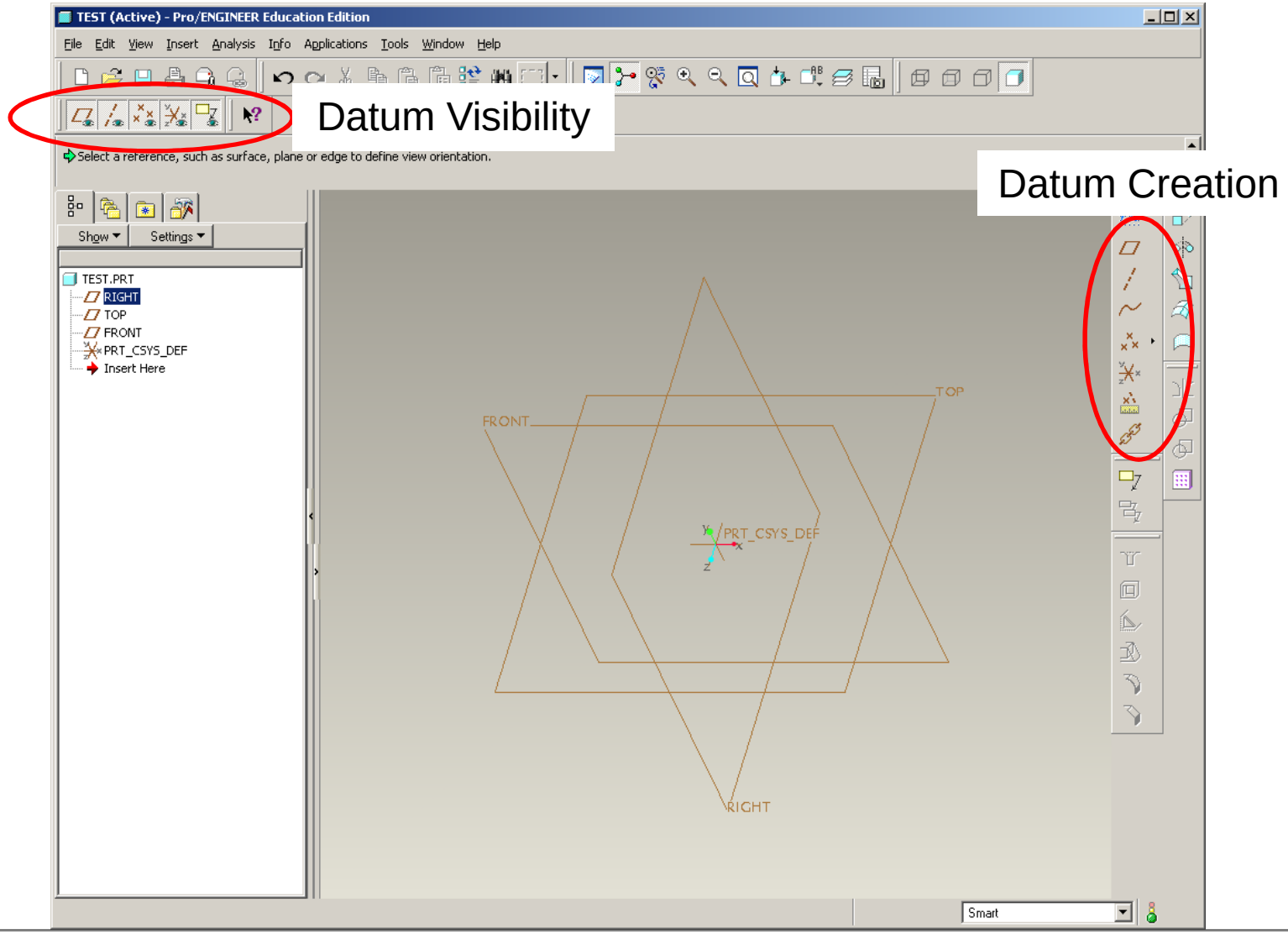
Datums are spatial features that locate a component or assembly in a coordinate system



Datums can have significance beyond building the CAD model

- Engineering Drawings and Quality Inspections

Datum Tools


TEST (Active) - Pro/ENGINEER Education Edition
File Edit View Insert Analysis Info Applications Tools Window Help
Datum Visibility
Select a reference, such as surface, plane or edge to define view orientation.
Datum Creation
Show Settings
TEST.PRT
RIGHT
TOP
FRONT
PRT_CSYS_DEF
Insert Here
TOP
FRONT
PRT_CSYS_DEF
RIGHT
Smart

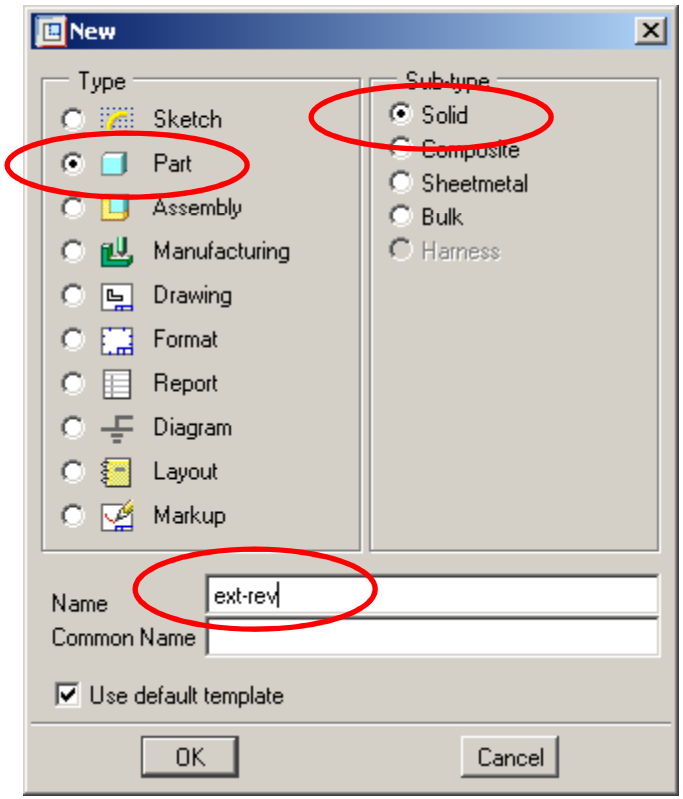
EXERCISE - Create a new part file

- Create a new subfolder within U:\proe called “inclass”

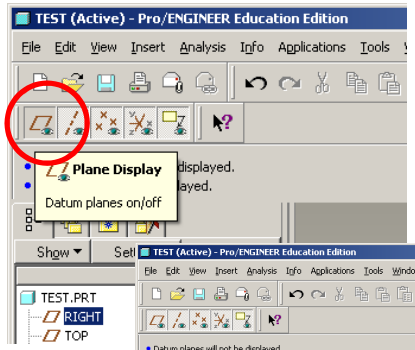
- Set working directory
  - File > Set Working Directory > Choose U:\proe\inclass

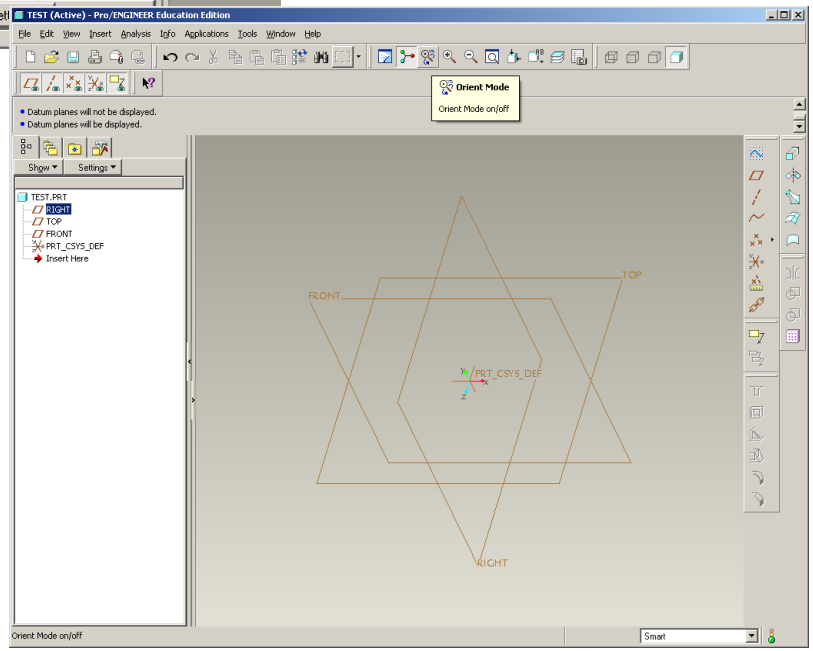
- Create a new part
  - File > New >

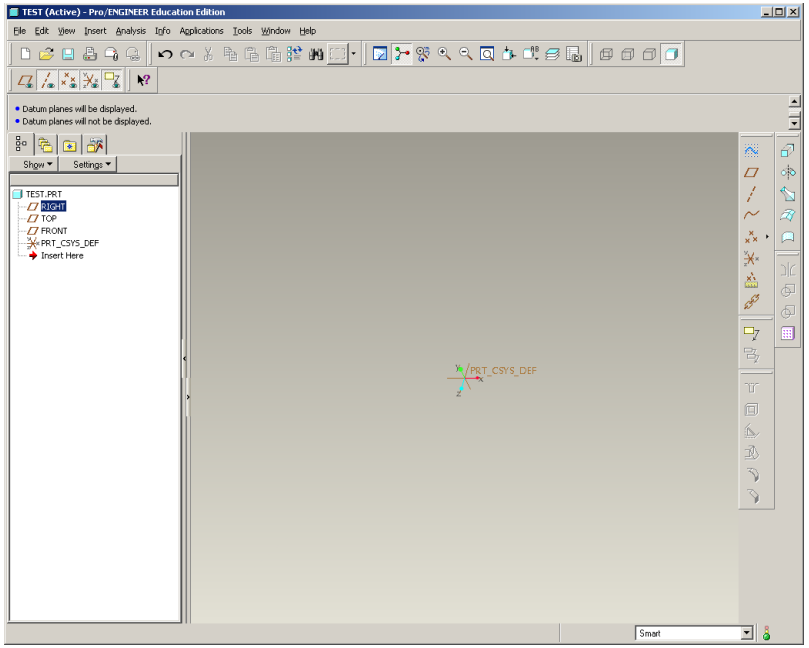
- Name file **ext-rev**



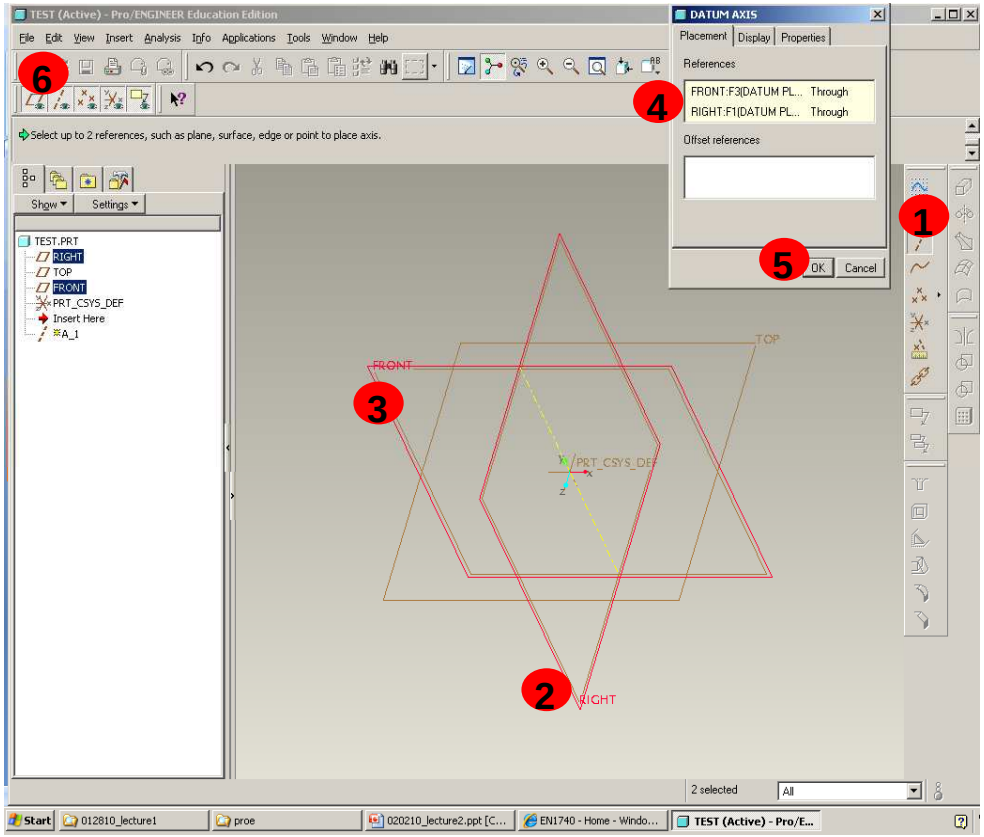
EXERCISE – Turn Datum Planes on and off







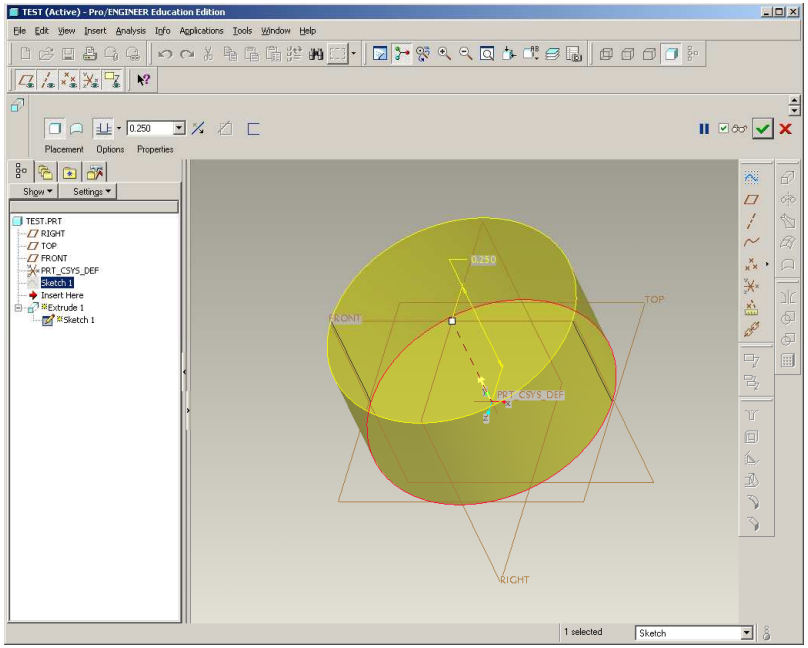
EXERCISE – Create an Datum Axis



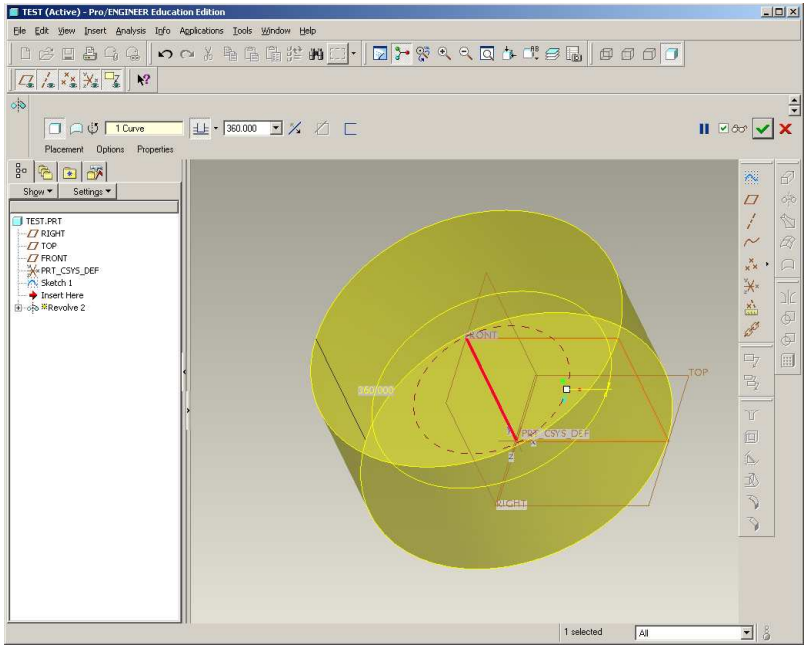
1. Click on Datum Axis Tool
2. Click on the RIGHT Datum Plane
3. Holding the Ctrl key, Click on the FRONT datum plane
4. Verify FRONT and RIGHT are the references
5. Click Ok
6. Turn the Datum Axis on and off with the Axis Display tool

NOTE: Steps 1-5 can be also be done by performing steps 2, 3, 1 (in that order)

Majority of 3D geometric features are created through two types of processes…

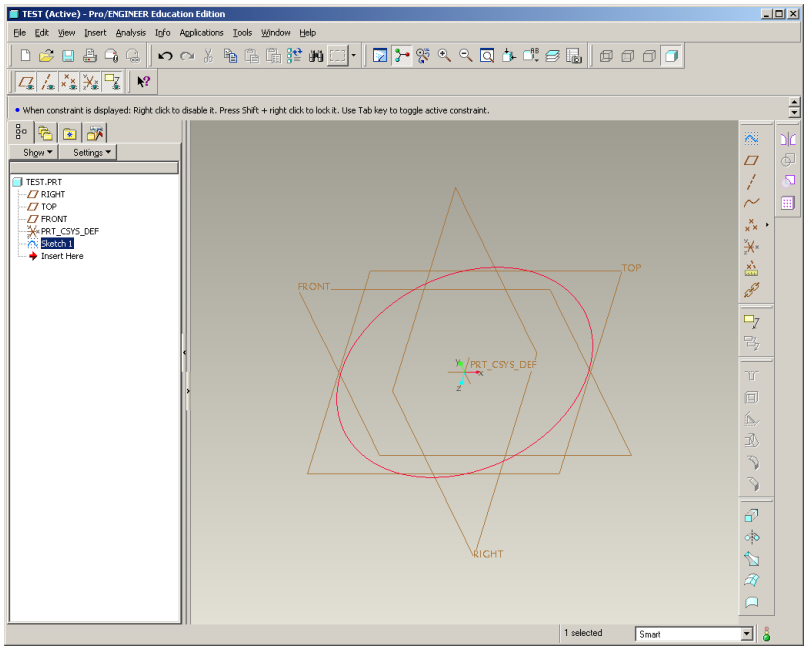


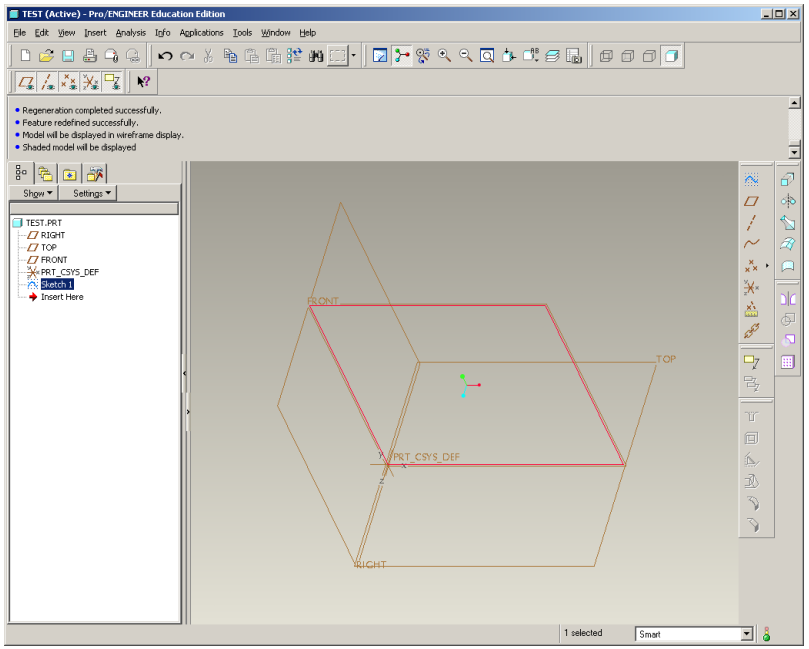
Extrude



Revolve

….which are defined using 2D sketches

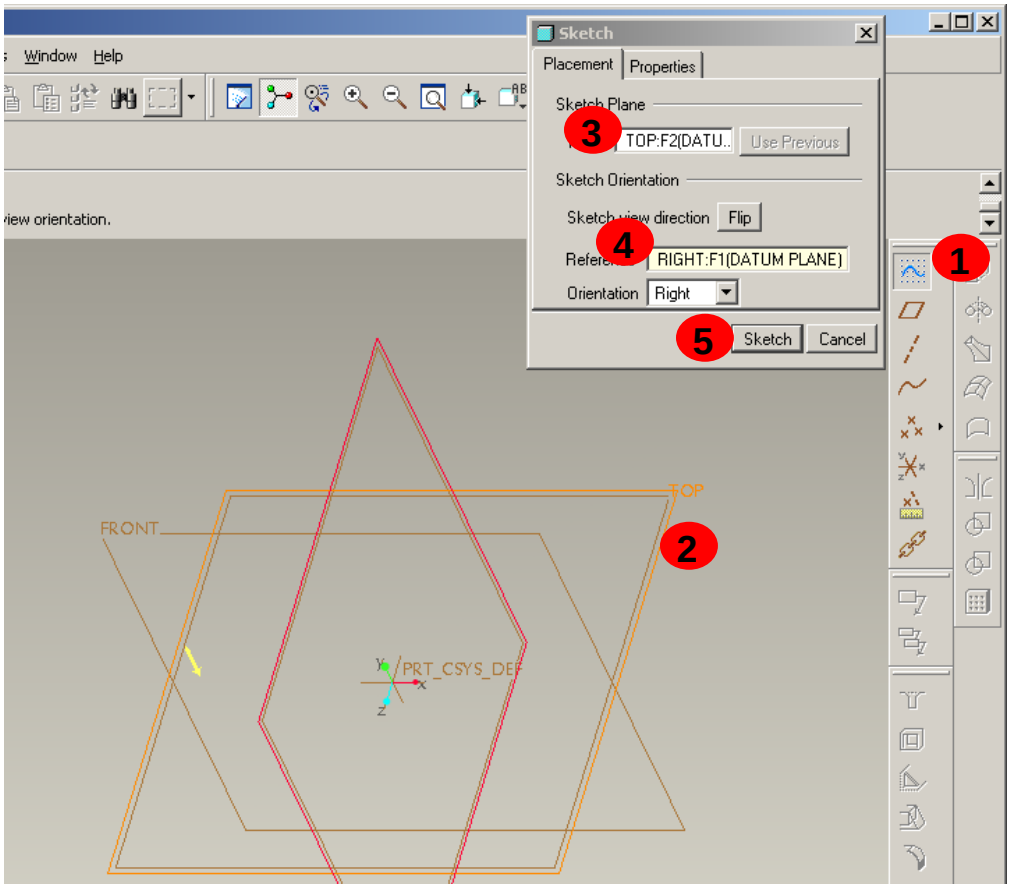




Sketch features used as bases for these and many other 3D features

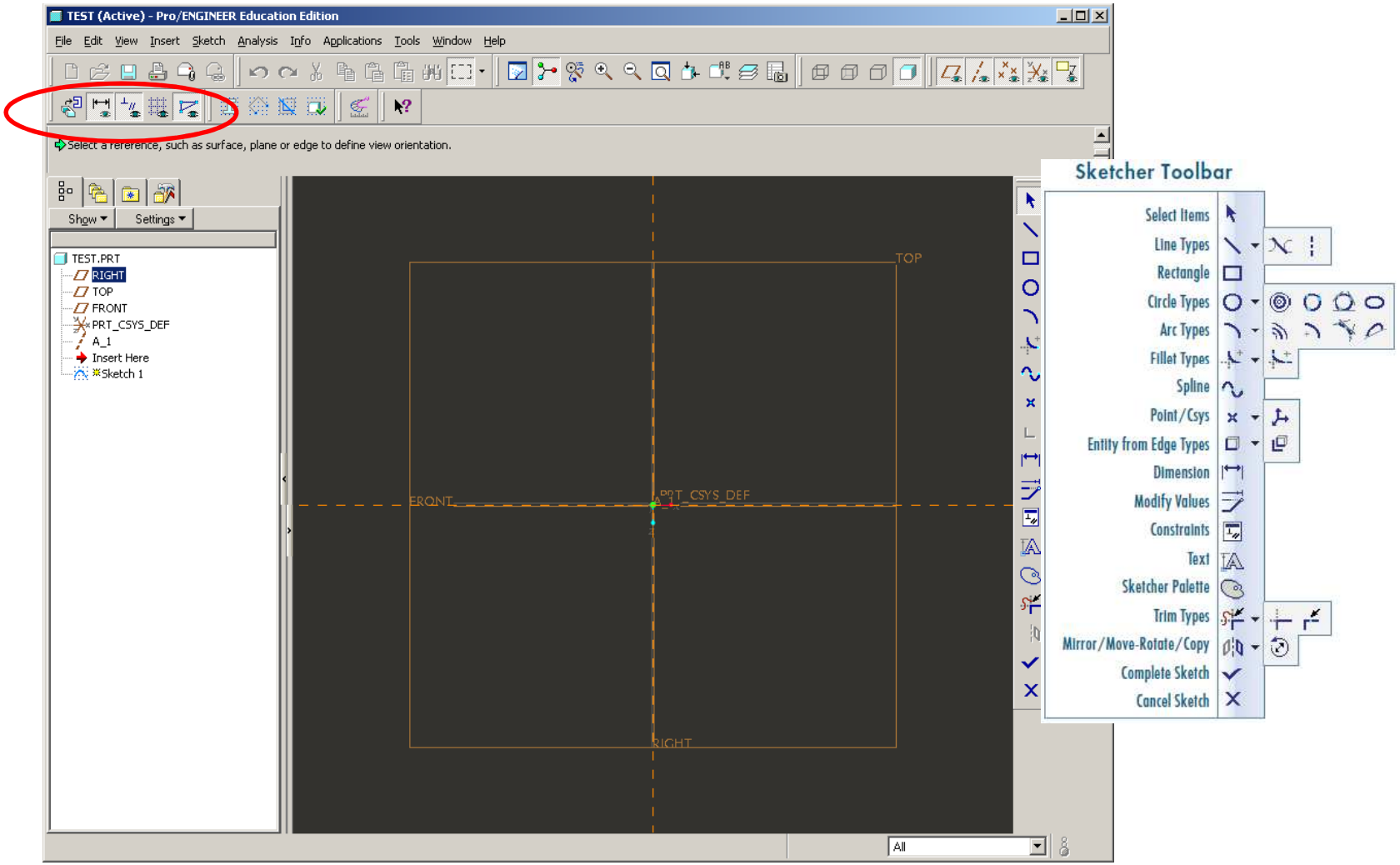
EXERCISE - 2D sketches are created in Sketcher Mode

- There are a number of ways to launch Sketcher
- To start, we'll make a stand-alone sketch

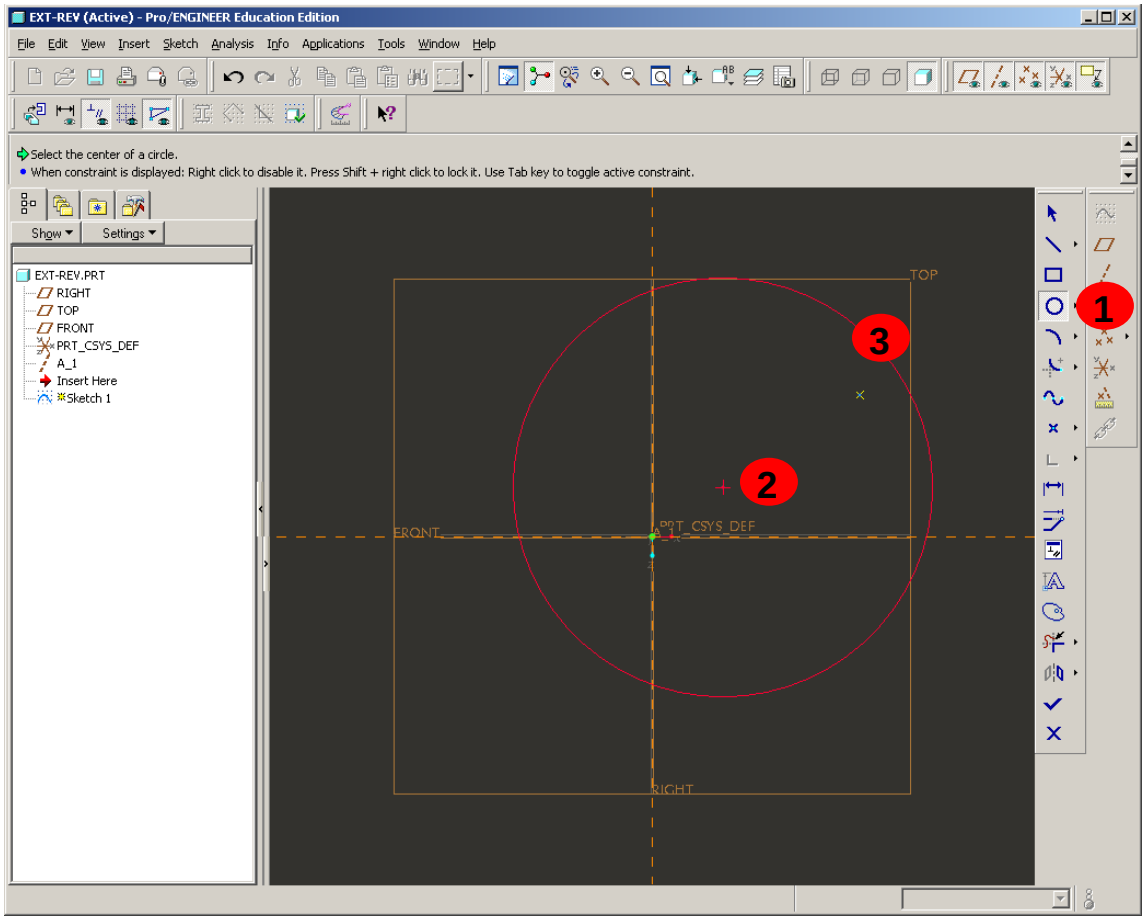


1. Click on Sketch Tool
2. Click on "Top" datum plane
3. Make sure "TOP:F2…" is the Plane
4. Make sure "RIGHT:F1…" is the Reference
5. Click Sketch to continue

EXERCISE – Welcome to sketcher

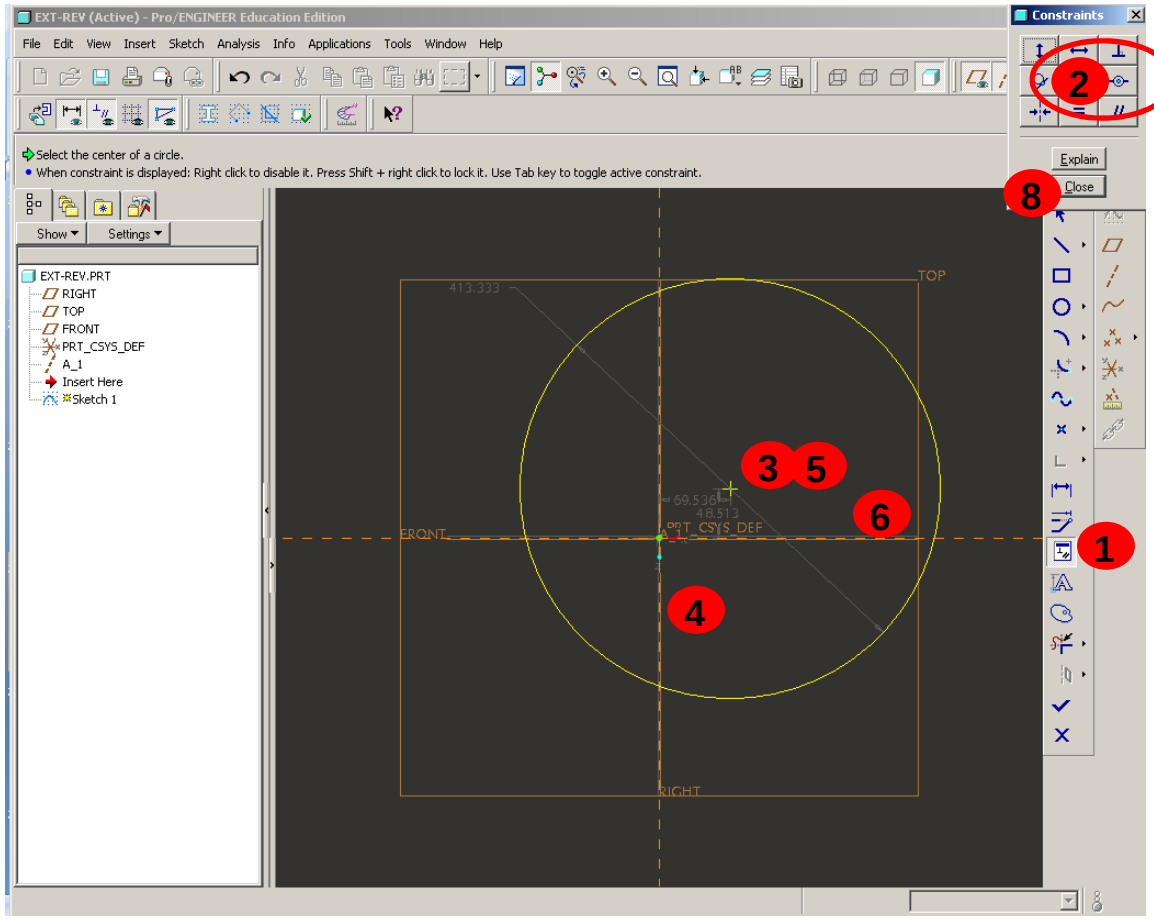


EXERCISE – Draw a Center-Point circle



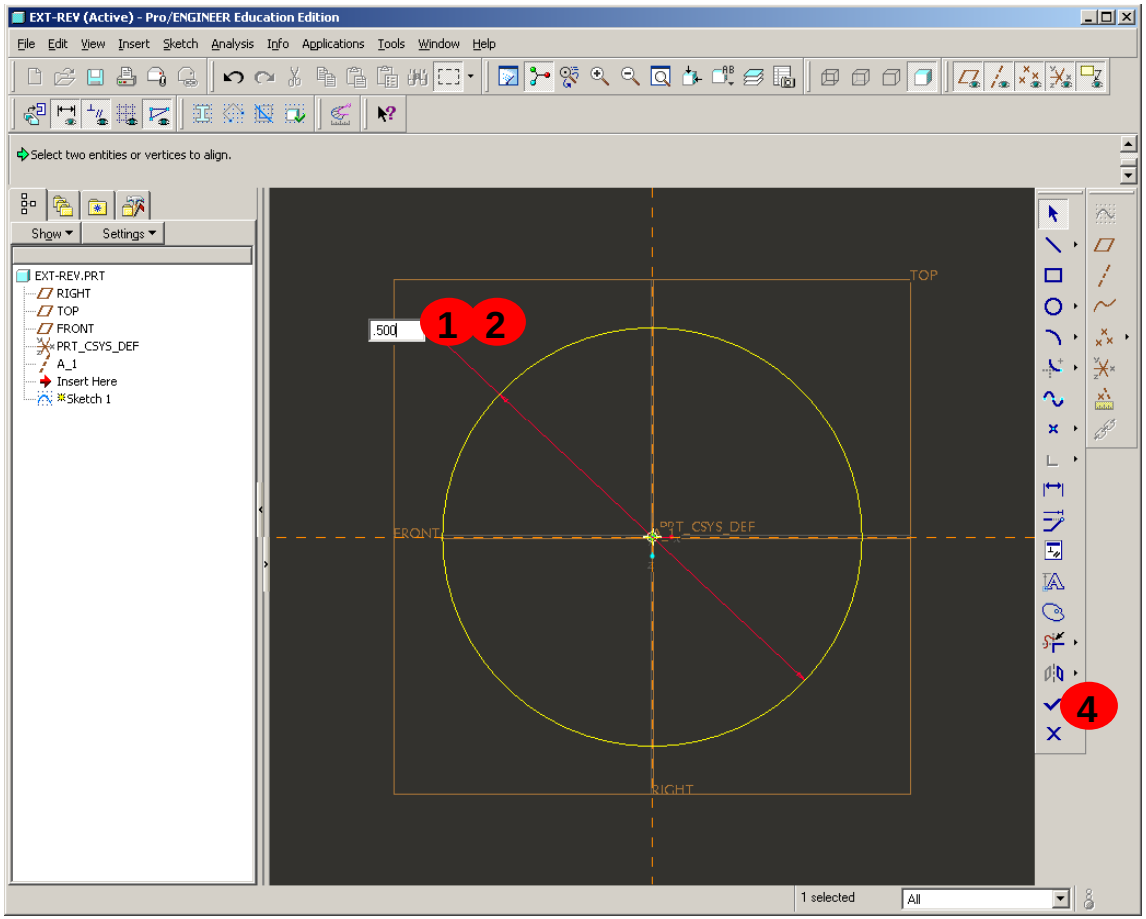
1. Click on Center and Point Icon
2. LMB a center point location
3. LMB at a point on the circle (i.e. at the correct diameter)
4. MMB to end creation

EXERCISE – Constrain Circle



1. Click on Constraints
2. In Constraints Pop-up, Click on Coincident
3. LMB on circle's center point
4. LMB on RIGHT datum plane
5. LMB on circle's center point
6. LMB on FRONT datum plane
7. MMB to end
8. Close Constraints Pop-up

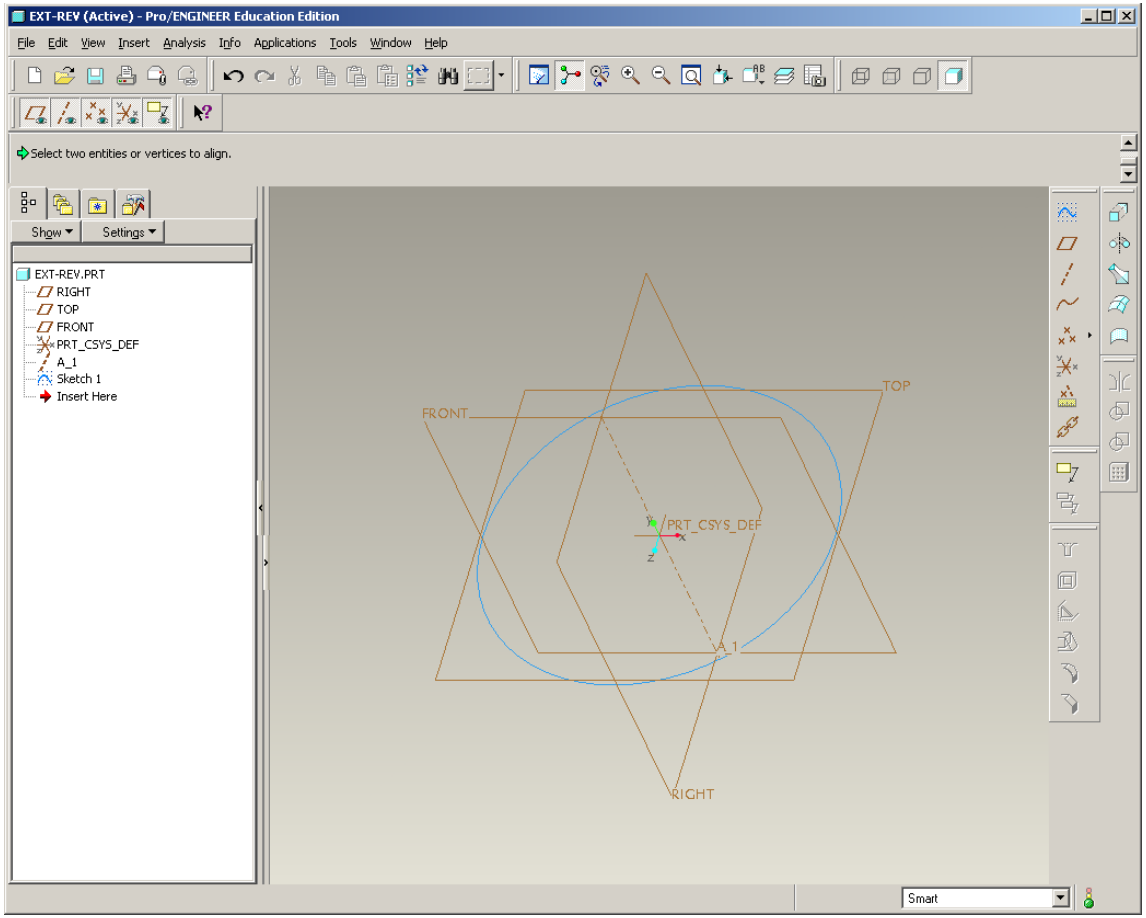
EXERCISE – Dimension Circle



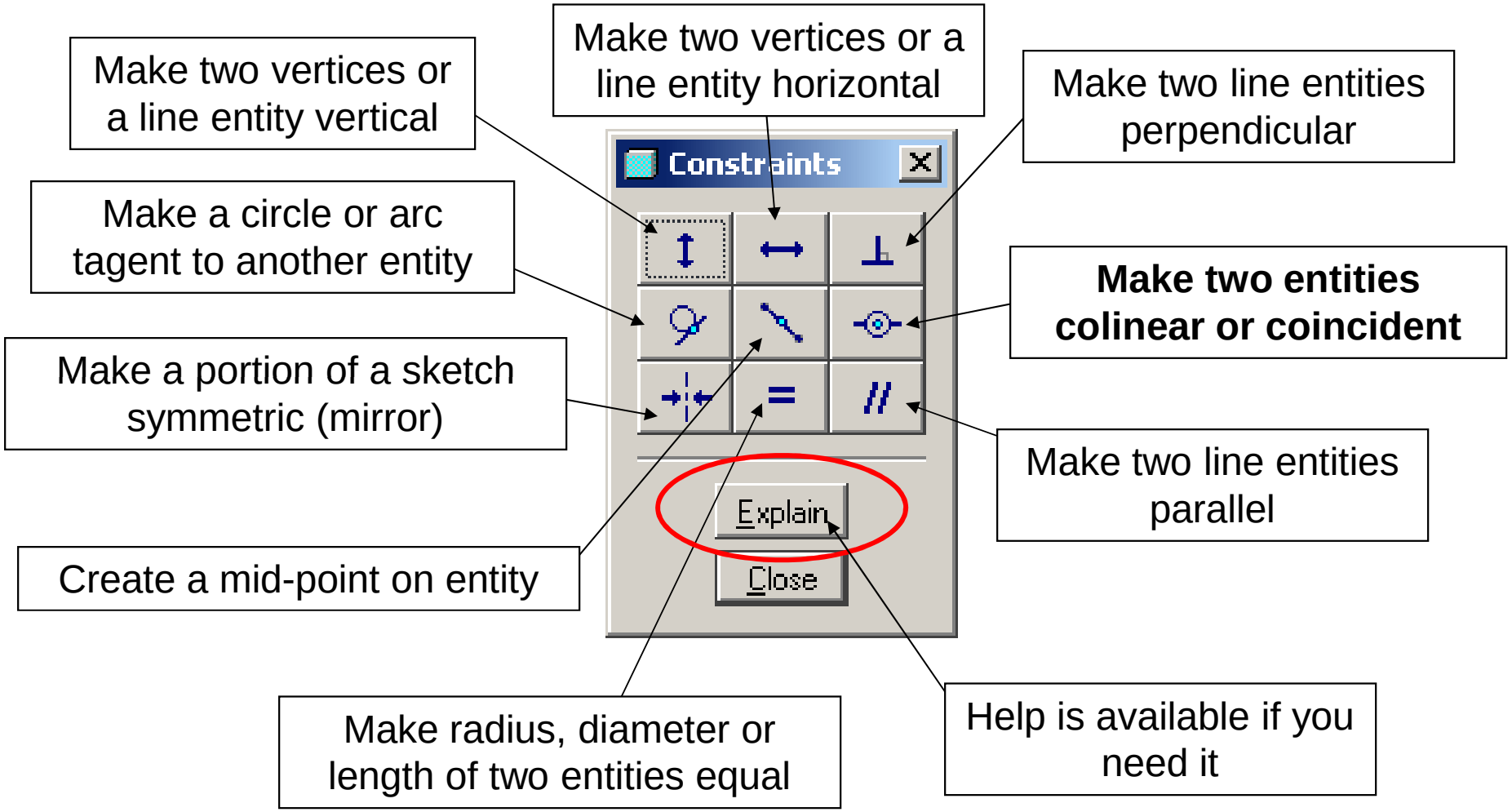
1. Double-click on diameter dimension
2. Enter .500
3. Enter or MMB
4. Click on Done

EXERCISE – Done

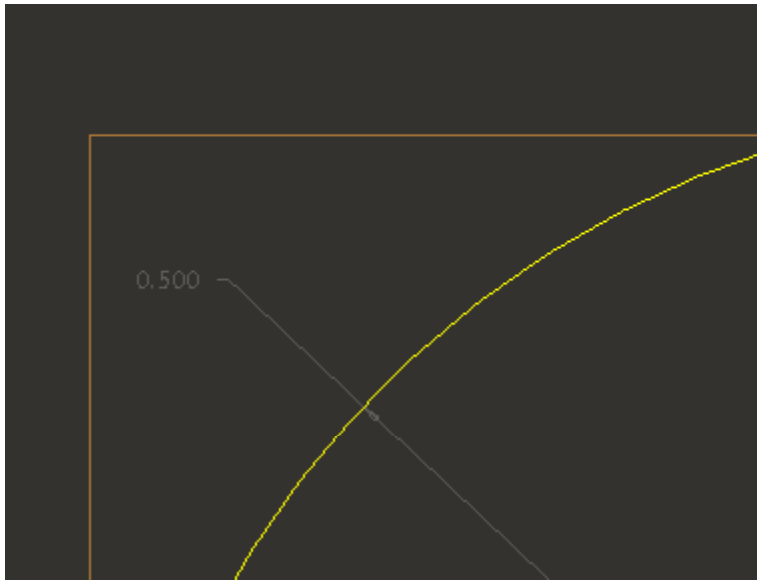
Press Ctrl+D to see the default view of the sketch just completed



Sketcher Notes – Constraints

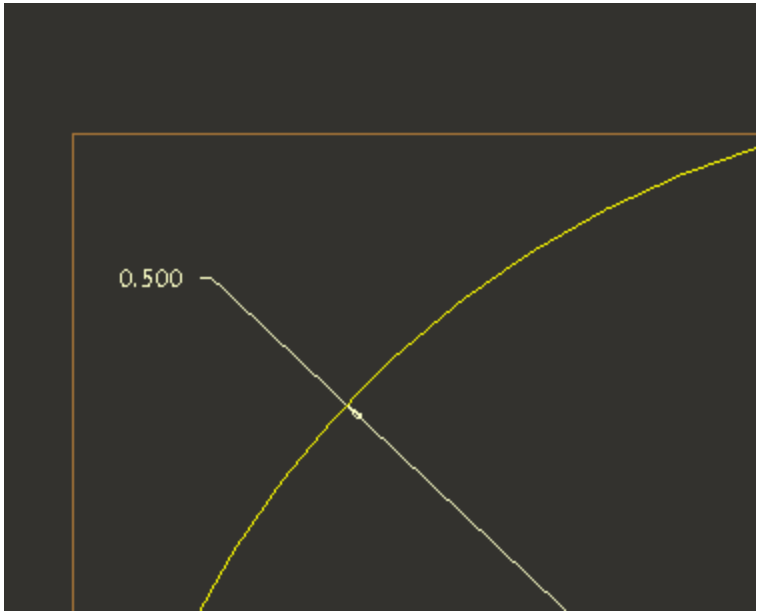


Sketcher Notes – Strong and Weak Dimensions

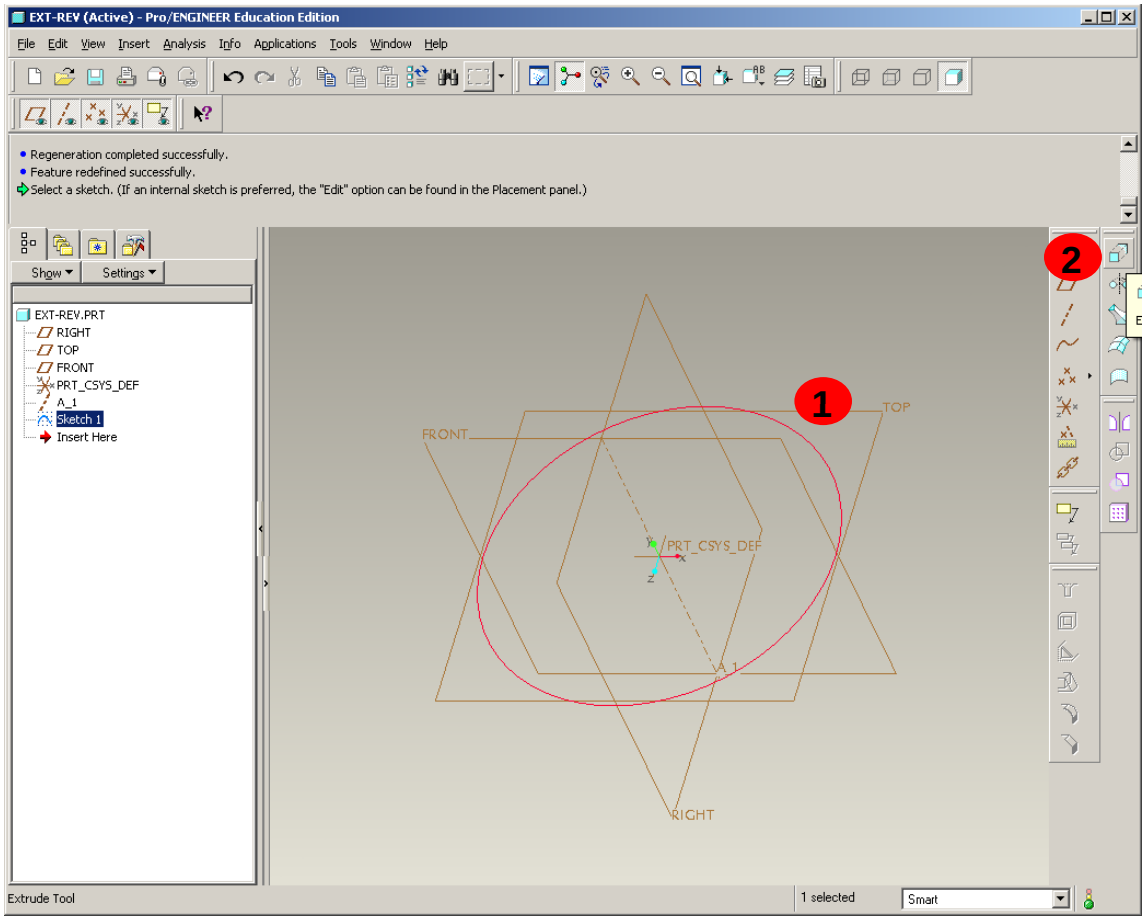


Weak dimensions appear in *GRAY. These dimensions are assumed by Pro/E*

STRONG dimensions appear in *YELLOW. These dimensions* have been identified by the users as being critical to design intent.

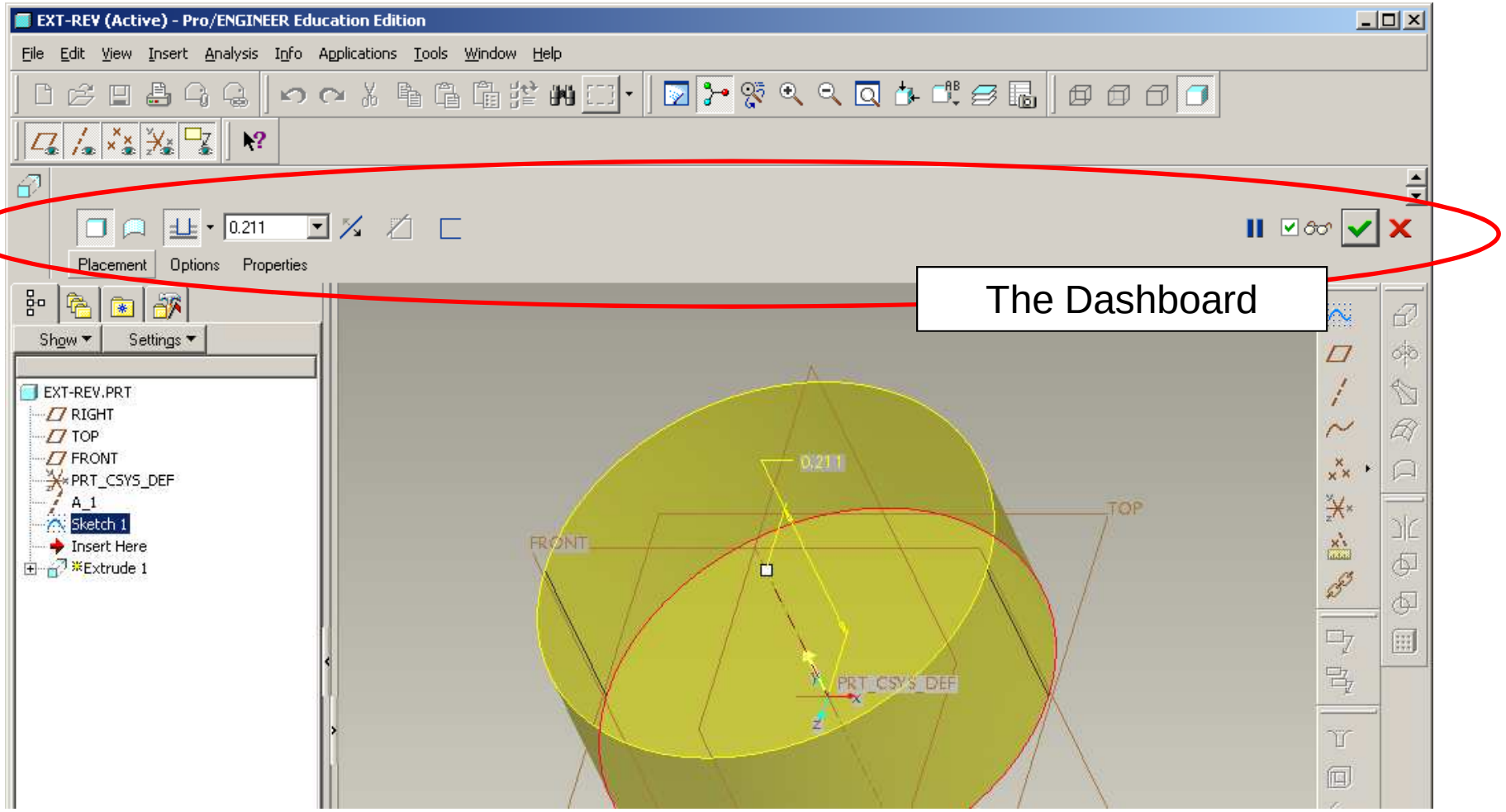


Extrude sketch to create 3D model

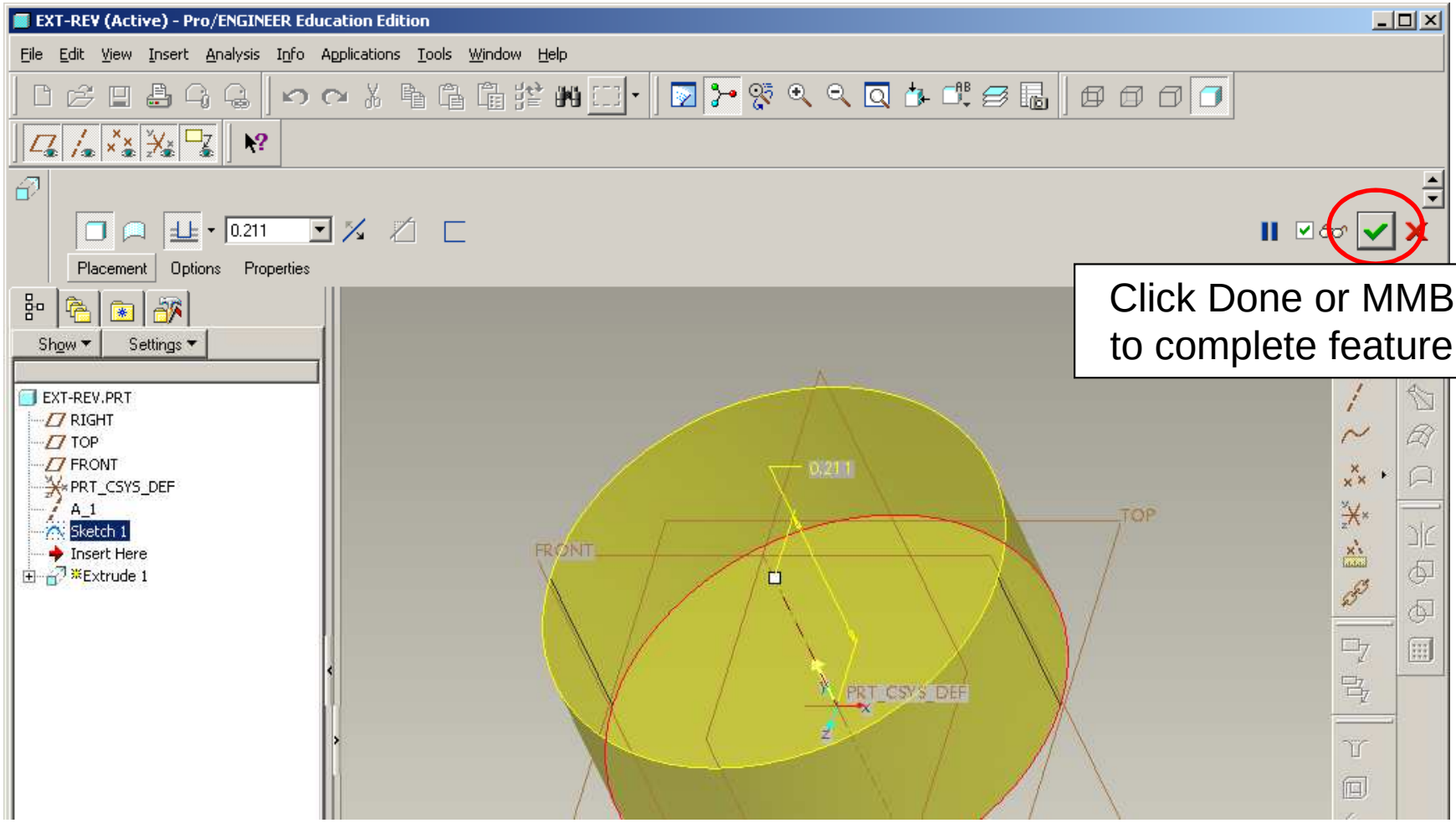


1. Click on sketch in the graphic window
2. Click on the Extrude tool

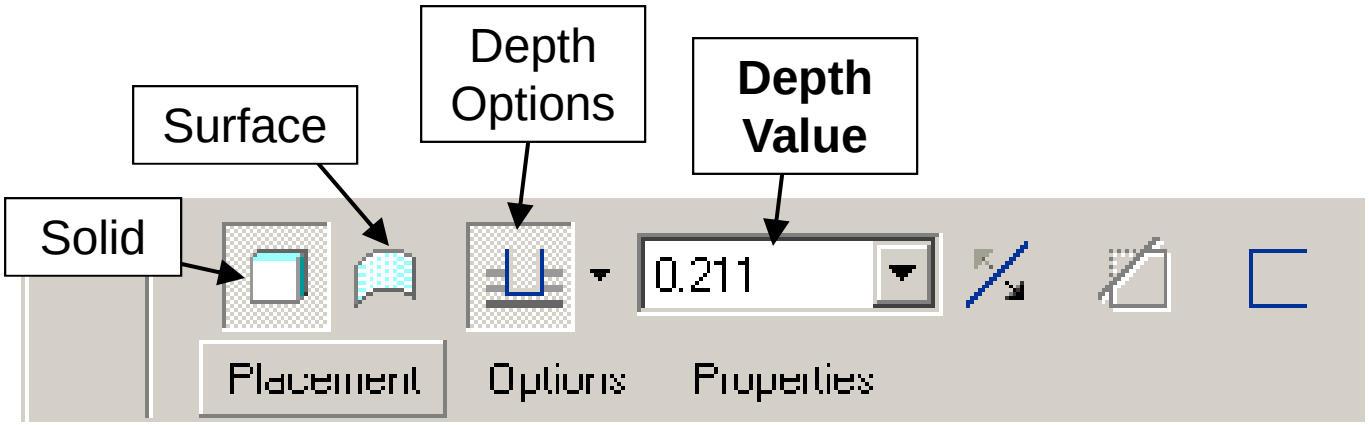
Solid Feature *Dashboard*

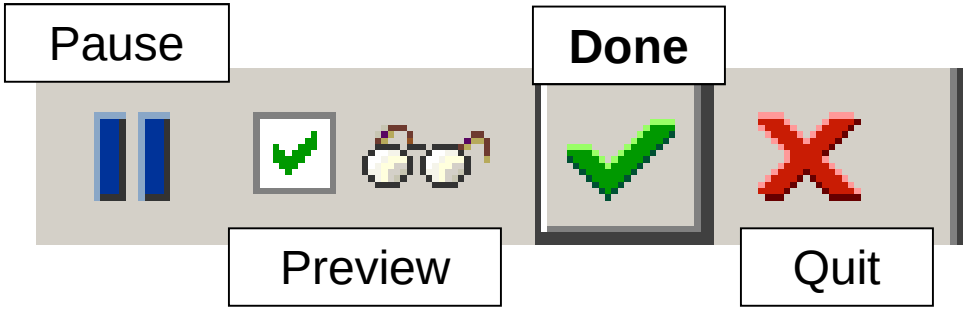


Extrude sketch to create 3D model



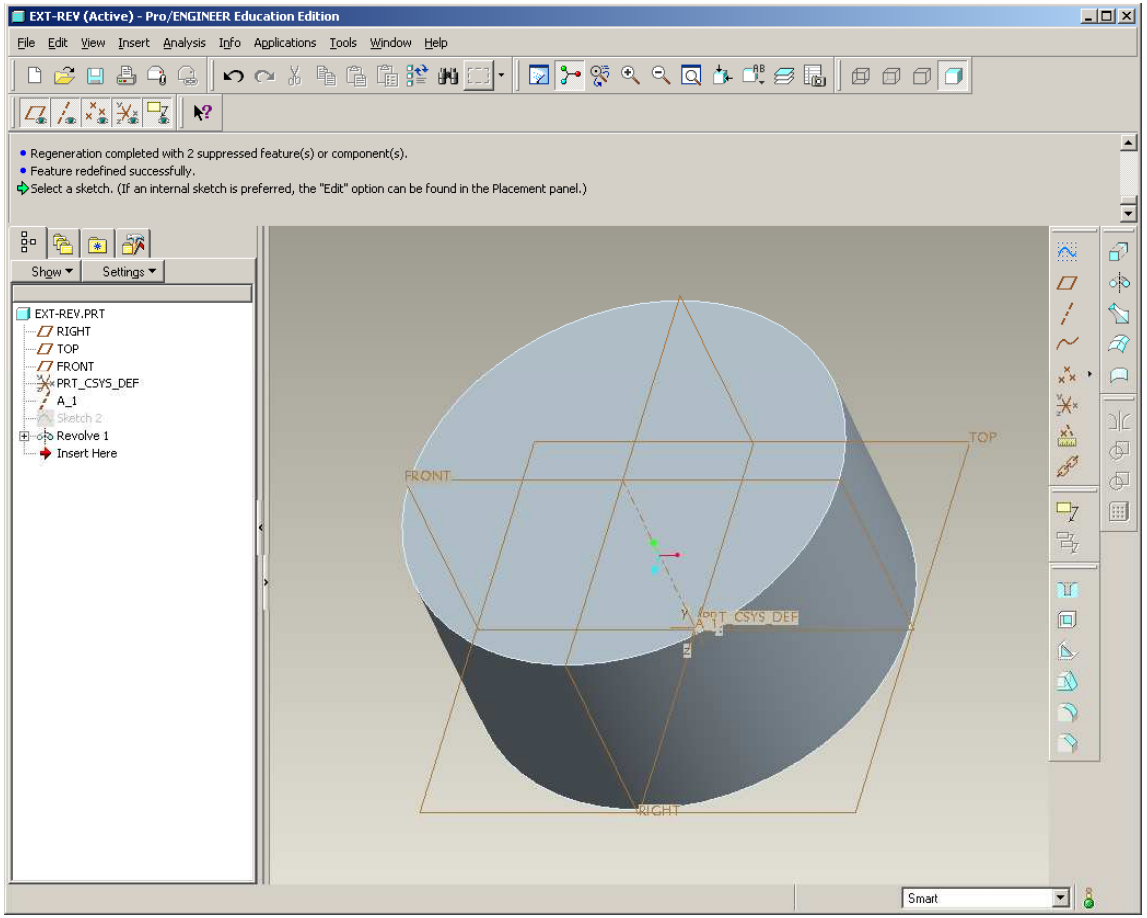
Solid Feature *Dashboard*





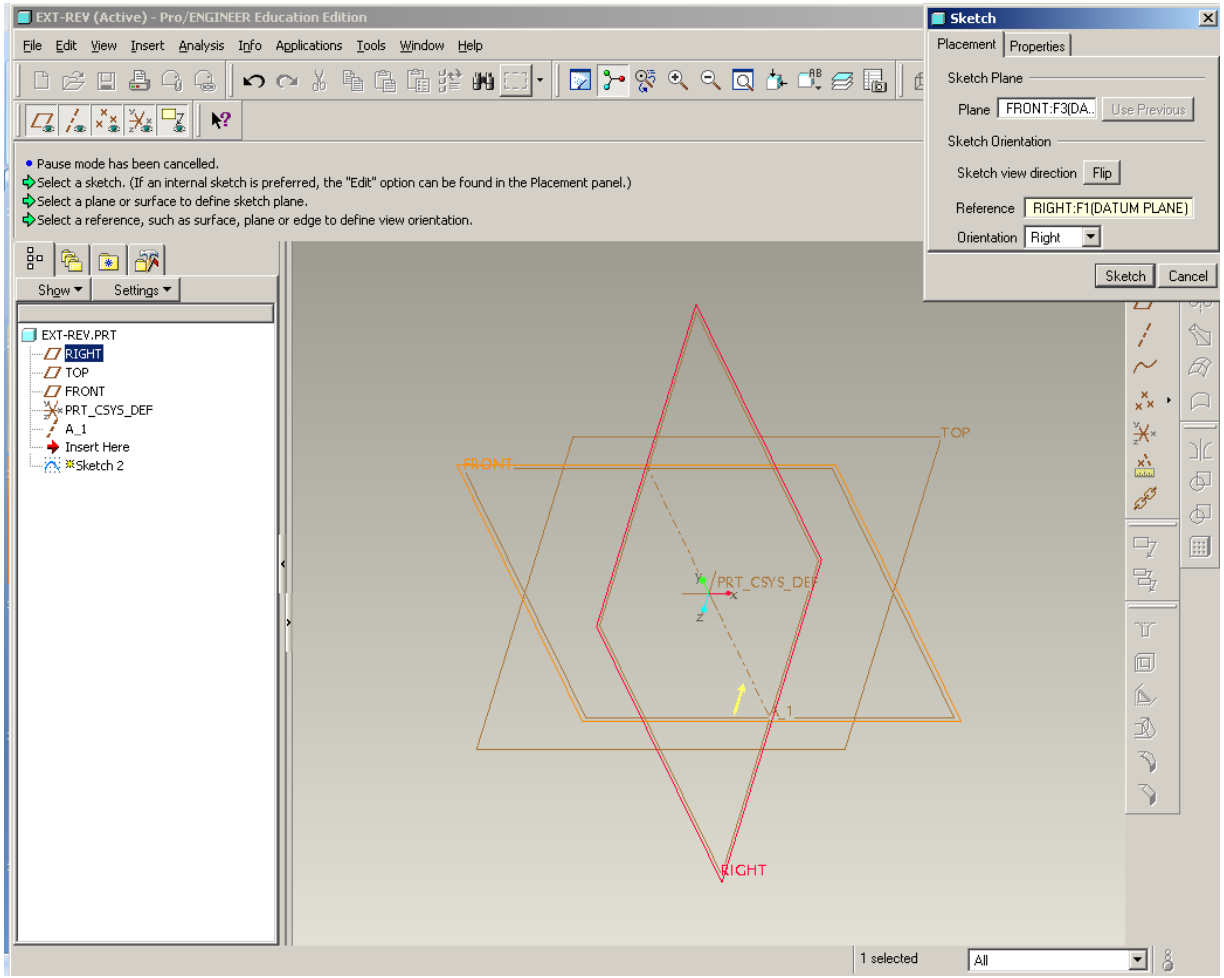
EXERCISE - Make the same solid with a *REVOLVE*

Done.



EXERCISE - Make the same solid with a *REVOLVE*

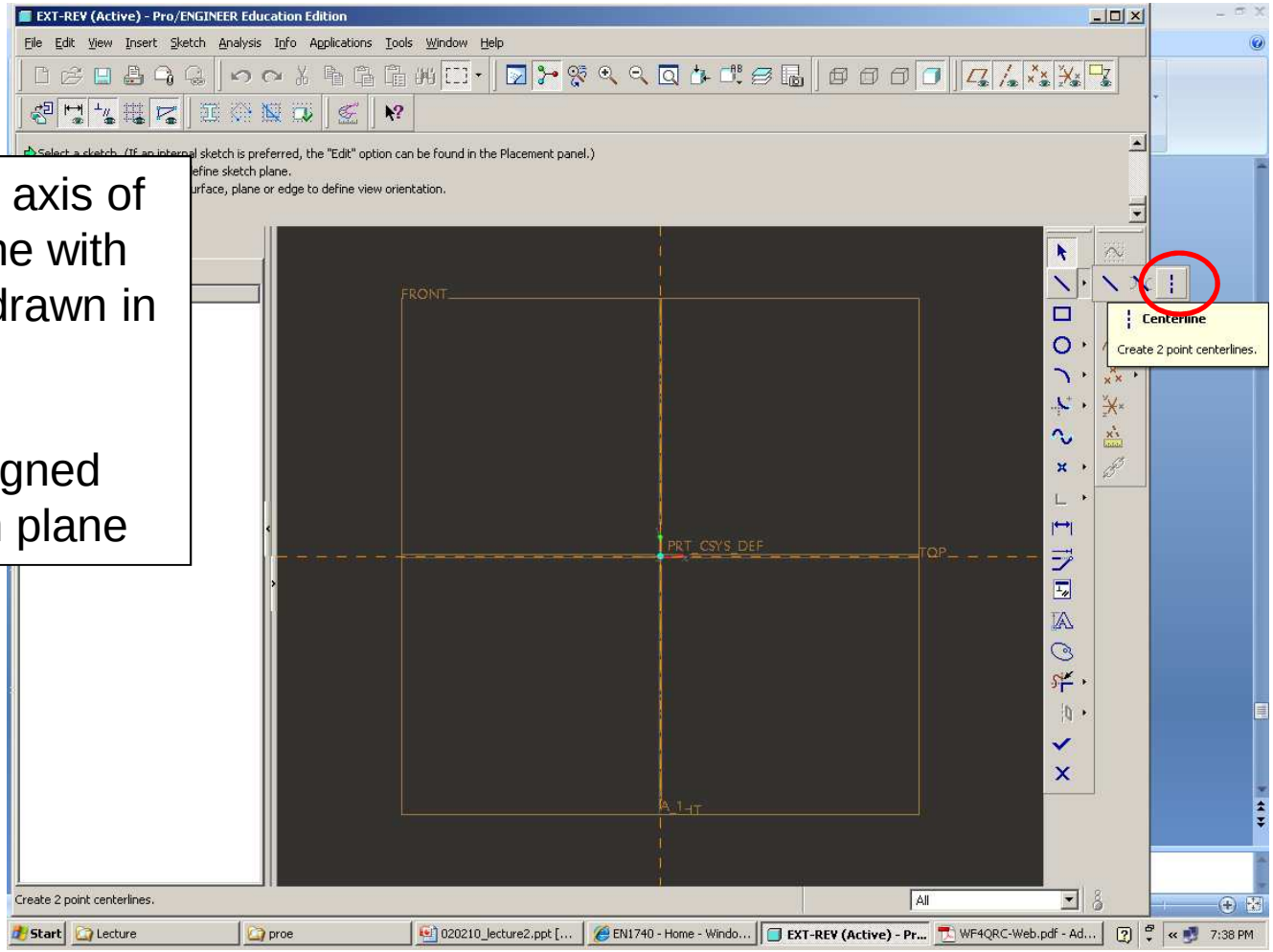
Create a sketch
- Sketch on the FRONT datum
- Use RIGHT datum as Reference



EXERCISE - Make the same solid with a *REVOLVE*

A Revolve requires an axis of revolution. This is done with the FIRST centerline drawn in the sketch.

Sketch a centerline aligned with the RIGHT datum plane



EXERCISE - Make the same solid with a *REVOLVE*

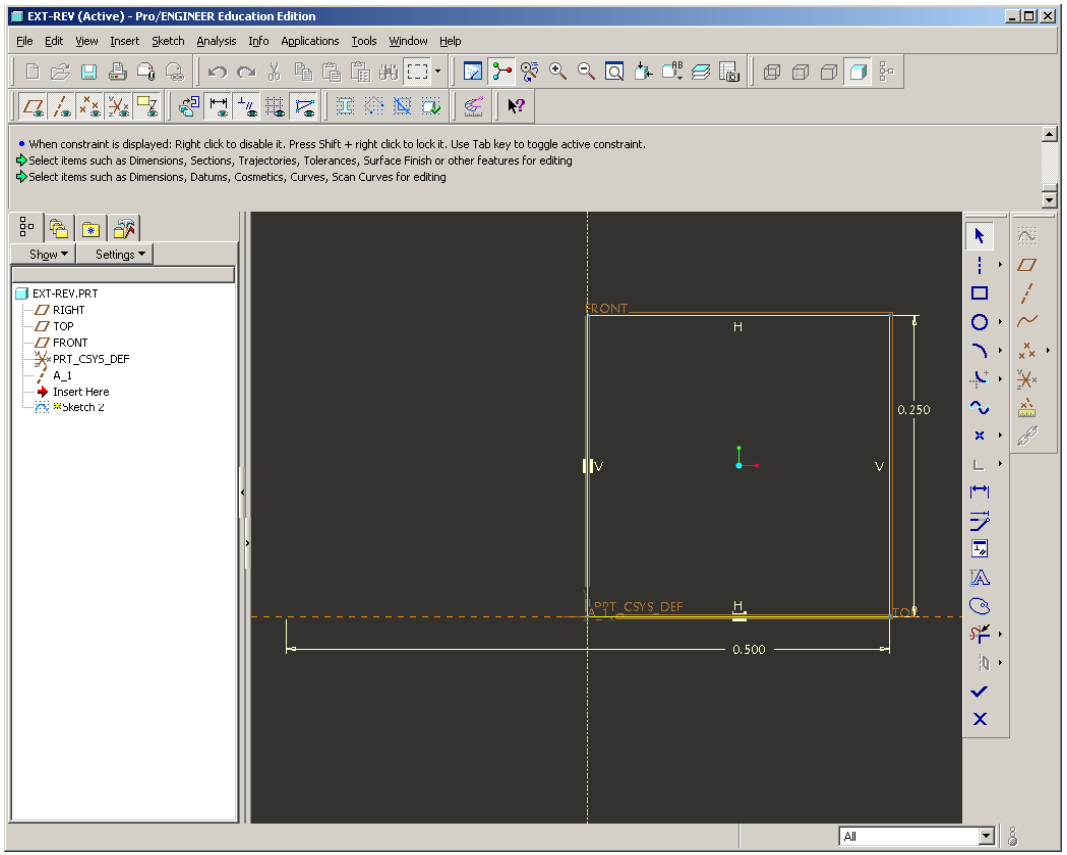
Sketch rectangle and dimension

- Align (Coincident Constraint) to RIGHT and TOP datum planes
- Height > .250
- Diameter > .500

NOTE: To dimension a diameter >

1. Click Centerline
2. Click Diameter
3. Click Centerline again
4. MMB

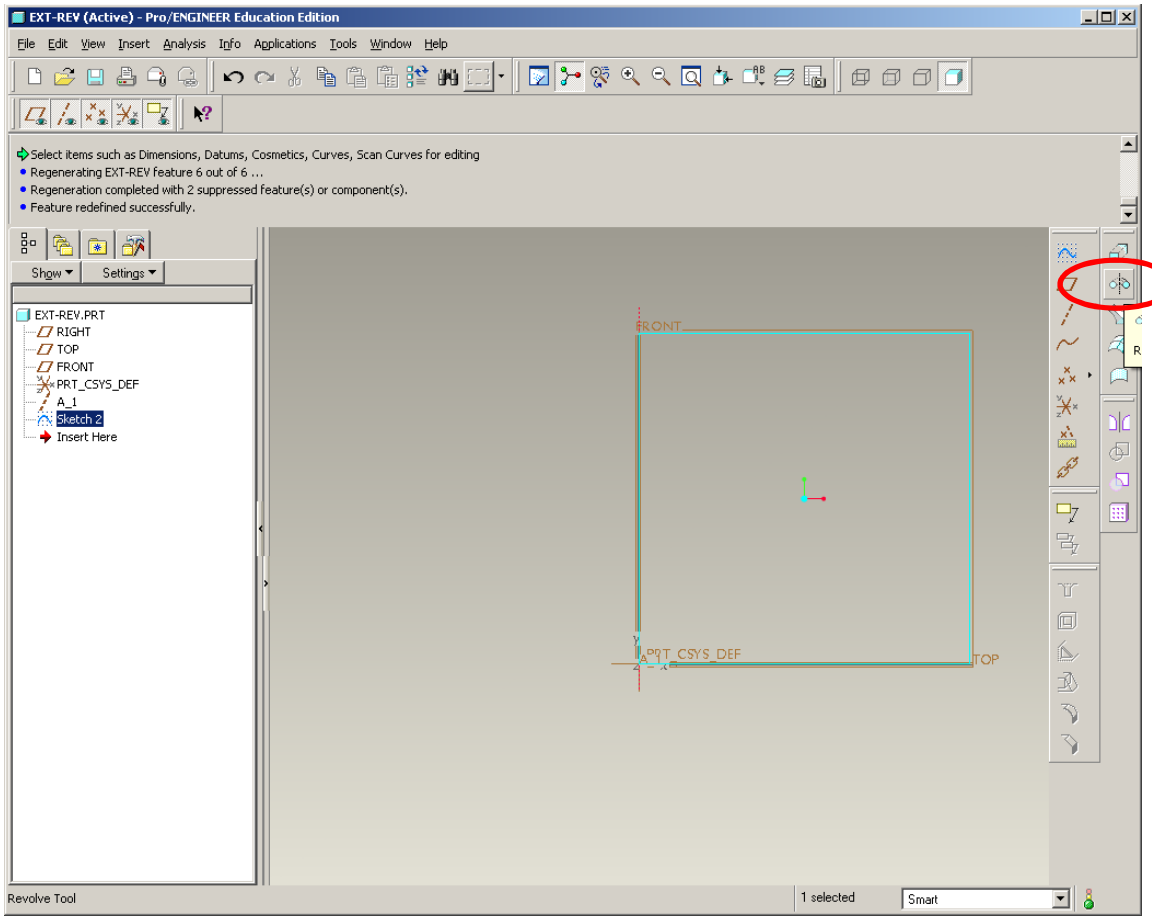
Click Done



EXERCISE - Make the same solid with a *REVOLVE*

With same sketch highlighted

- Click Revolve

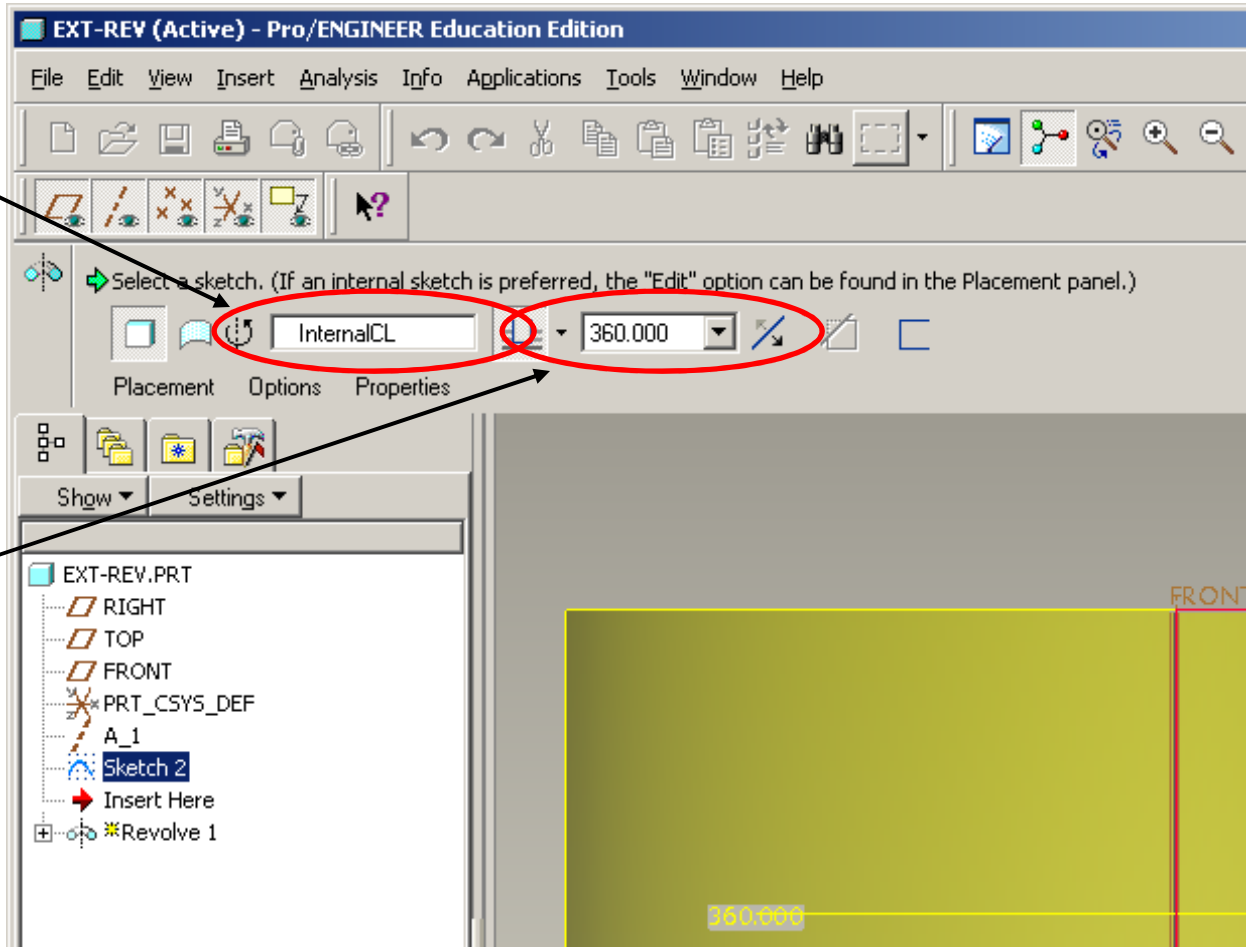


EXERCISE - Make the same solid with a *REVOLVE*

Dashboard for REVOLVE is almost the same as EXTRUDE with a few exceptions

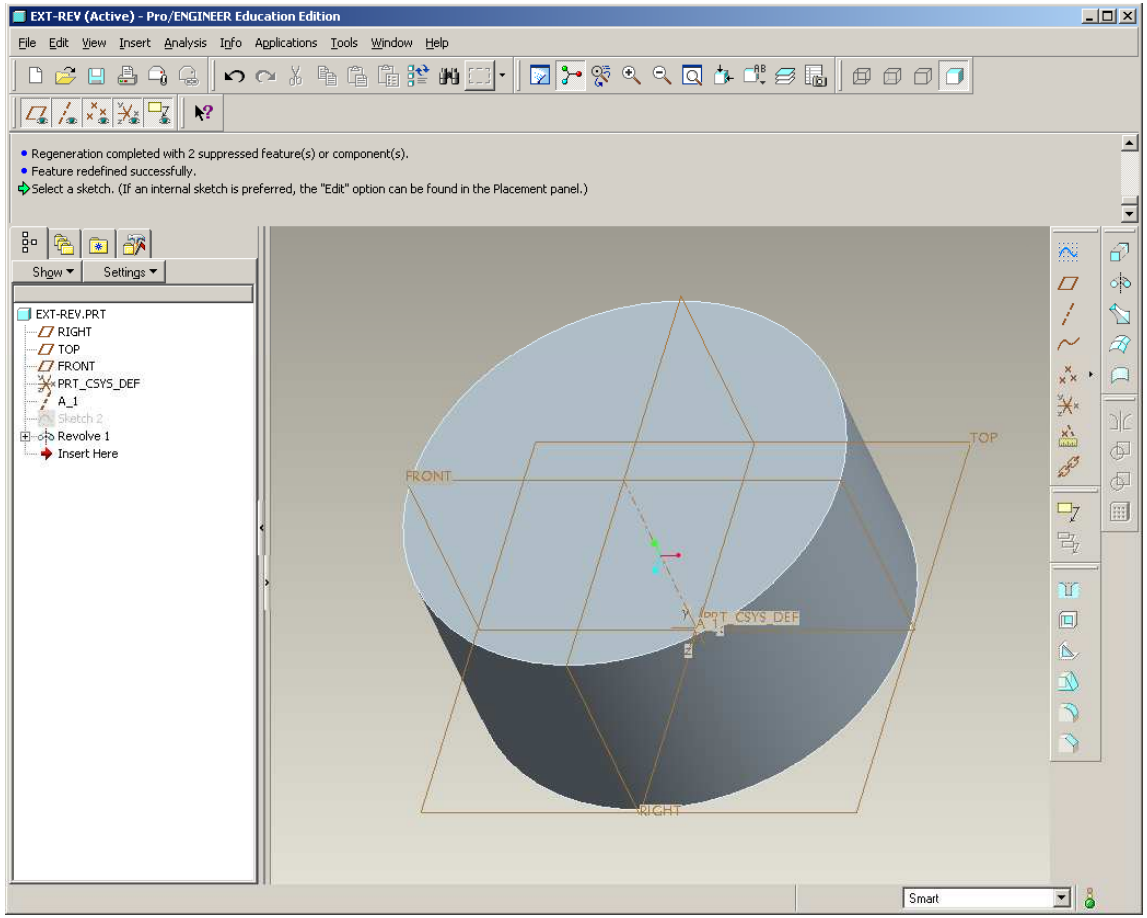
Pick a centerline to revolve about

Number of degrees to revolve through

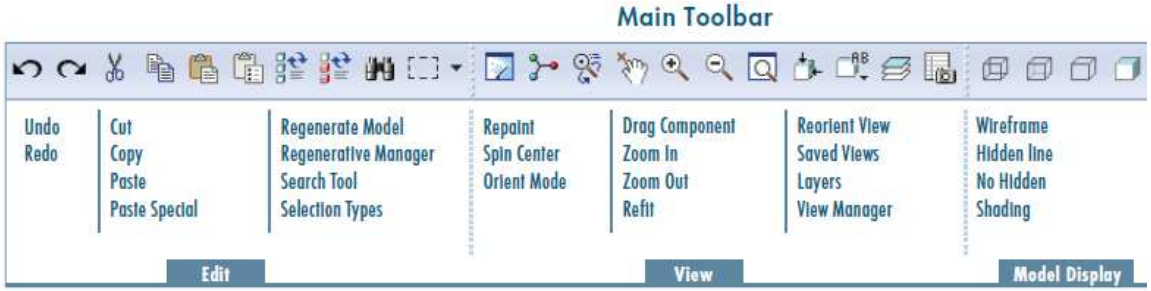


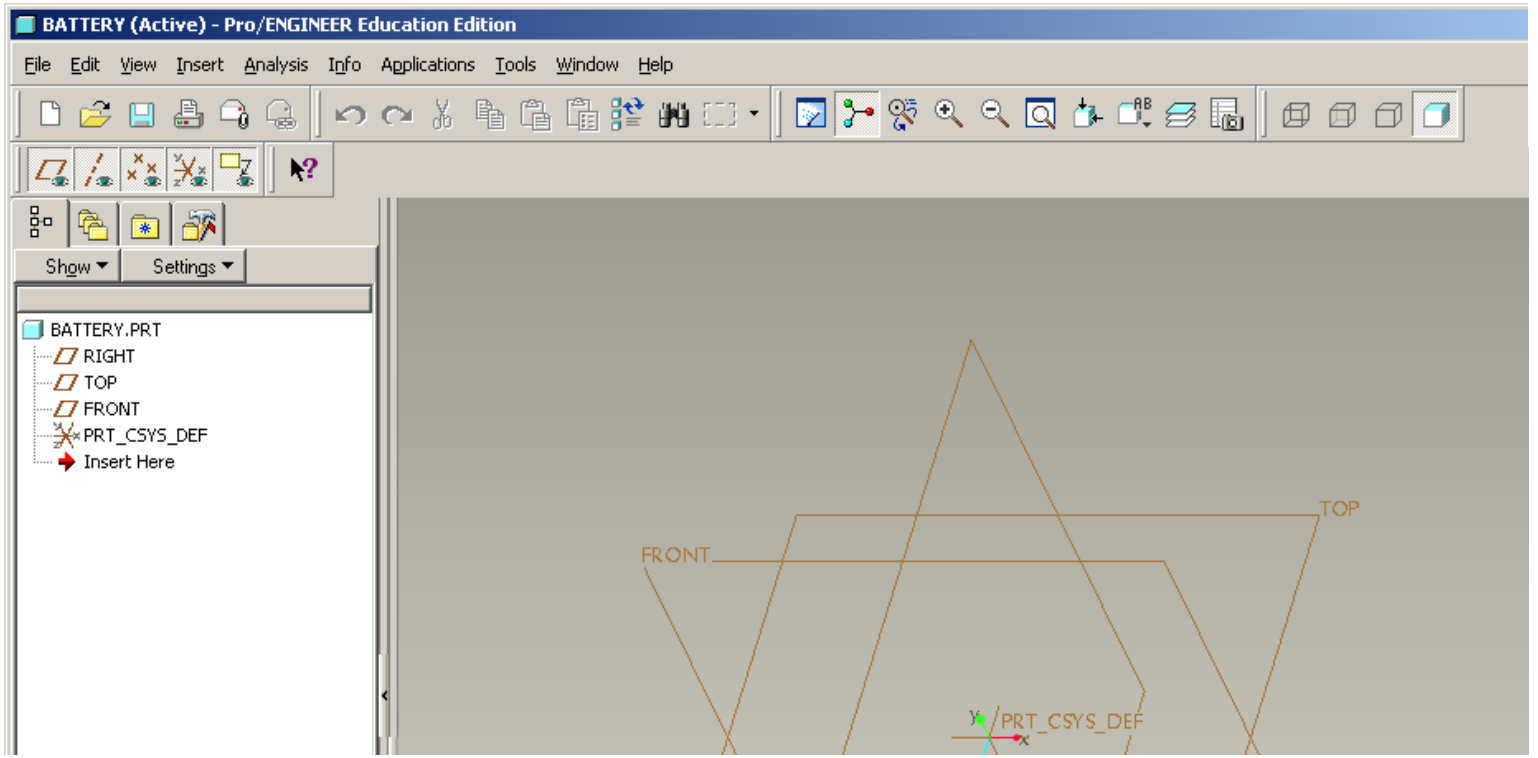
EXERCISE - Make the same solid with a *REVOLVE*

Done, again.



Pro/E Layout and Iconology





Pro/E Layout and Iconology