Objective: Build parts in Quick Start Guide

Startup Pro/E

Open completed pin-cube part to demo the end point

Create a new directory and set working directory (r-click on directory, Set Working Dir)

(Follow along using the tutorial)

Create block part

Dimensioning
   To dimension a distance: L (one side), L (other side), C (place)
   Pro/E design intent vs your intent

Modify dimension in Part mode

Add a hole

Create pin part

To dimension radius L-click outside, L-click C/L, C-click to place
For diameter, L-click out, L-click C/L, L-click out, C-click to place

Selecting parts, features, surfaces, edges…

Create assembly

Create a drawing
Making a multi-view drawing
Create drawing of the block part (scaling)

Follow guide.

Explain difference between Erasing and deleting.

Show changing location of dimension, moving dimension from one view to another, dimension cleanup, and adding axes.

Setting units
Open notch.prt in Demos folder

Edit > Setup > Units > select units > Set > read warning box

Changing dimension and changing dimensioning scheme
Open notch.prt in Demos folder

Get front view: Saved view list > FRONT.

Double click on part to get dimensions. Change dimension. Regenerate (Ctrl-G)

Note that you can change a dimension so that part makes no sense, so be careful

Note that dimension scheme is fixed. How can you change?

R-click > Edit Definition > Placement > Sketch

Dim tool to change dim

Back in dashboard, show preview. Once click check, part is regenerated and no going back.
PRO/E THIRD LECTURE DEMOS

Relations
[TG:3-16]
Open notch.prt

Select part on menu, R-Click > Edit to show dimensions

Tools > Relations

Note that dimensions changed to symbols. Demo the toggle dimensions button in Relations dialog box.

/* notch is ¼ the width
   d3 = d4/4

Regenerate
Demo that step cannot be changed and that two go together

Copying a feature using mirror
(TG:4-12)
Open mirror.prt

Edit > Feature Operations > Copy > Mirror > Dependent > Done > select hole
(use of model tree to select) > Done > select plane to mirror about

Change hole location > Regenerate to show dependency

Hex object
Create new part, hex.prt. New feature > protrusion.

Add 2 centerlines that go through the origin, spacing lines by about 120 deg with one of them about 60 deg from vertical. Dimension one of the lines to be 60 deg from the vertical reference and dimension between lines to be 120 deg. Create a Center/Point circle (do not dimension this circle). Select circle, then Edit > toggle construction. Draw hexagon while constraining to the construction circle, the reference lines and the centerlines. Dimension across one flat; the rest will follow. Demo changing dimension.
Assign material density

Edit > Setup > Mass Props > enter density (Aluminum = 0.095 lb/in^3, Steel = 0.282 lb/in^3). Units are current units for part

Even better way:
Edit > Setup > Material > Define
Then Material > Assign

Calculate Weight and Center of Gravity
To find weight and c.g. of a part:
Analysis > Model > Mass Properties
Widen dialog box
Click eyeglass button
Advanced Demos

Map keys
Open notch.prt

Tools > Mapkeys
New > [enter key sequence and name]
Record > [do key strokes and mouse clicks, no clicks in graphics window] > Stop
> OK
Save > Close
Tools > Options > Open config

Suggestions:
sv shaded view
wv wire view
dat datums on/off
fv front view
dv default view

Raised text
Open notch.prt

Create protrusion on front surface

Text tool in sketcher toolbar > follow directions, for text line draw vertical line bottom to top

Show raised, lowered text. Change depth direction using remove material button

To edit text: Edit def > Placement > Edit sketch

Compute lengths or surface area: Analysis > Measure > follow directions.

View order that model was created: Tools > Model Player. Click for/rev arrows to step through model creation.
### Make a spring using a helical sweep:
Create new part. Insert > Helical Sweep > Protrusion > Constant | Thru Axis | Right Handed > Done. Select Front datum for sketch plane and Top datum for top reference. Create a vertical center line that passes through the origin. Create a vertical line of height 20.0 located 5.0 to the right of the origin. Accept the sketch. Enter pitch value of 4.0. Sketch a circular cross-section (the cross-section of the spring) of diameter 1.5. Accept the sketch. Preview the feature.

### Blends:
Create a new part. Insert > Model Datum > Plane. Reference to FRONT, offset by 100. Select Sketch Tool icon at the right and sketch on the FRONT plane. Sketch the first cross section (a square). Accept the sketch. Create a second cross section sketch on DTM1, for example a rectangle with rounded sides. Each section in the blend must have the same number of vertices. Create the blend. Insert > Blend > Protrusion > Parallel | Regular Sec | Sketch Sec > Done > Straight > Done. Follow the message prompts. Select the FRONT sketch plane. Flip arrow to point towards the DTM1 plane. In sketcher, use the Create an entity from and edge tool to pick up all four edges of the square. Note the order of the vertices which must be consistent in the next section. When done, select the arrow tool then right press > Toggle Section. Use the edge tool to pick up the next section. Accept the sketch. Enter the depth of the section. Select Preview from the Protrusion dialog box, then OK.

### To insert a dimension onto a drawing:
Insert > Dimension > New References. Create dimension. If doing baseline dimensioning: Insert Dimension > Common Reference. Select baseline, then L-click/C-click to create dimensions from baseline. You must add dimensions when the Constraint Manager adds constraints that drive dimensions that don’t show up with Show All. Some designers chose not to use Show/ Erase and instead create all dimensions from scratch. This is only recommended in special circumstances.)

### To create a reference dimension:

### To show the tolerance for selected dimension:
Select dimension > R-press > Properties > Value and tolerance.
Cutting a keyway into a cylinder: (From Chap 8/9 assignment) Pre-make cylinder. Extrusion for cut. Sketch on Front plane. With the "Use" tool, click on the top edge of the cylinder to create a Sketcher line. Delete the line so that a reference line remains. Draw the box defining the cut, snapping to the new reference line. Exit sketcher. Two-sided removal. Create drawing and show depth dimension. While in drawing, show changing dimensions and regenerating. Go back to part and show change.

Mechanism Motion: Assemble straight line mechanism and slider crank, with constraints and show dragging and moving. (See animate.txt in ProE-Mechanisms folder.)