

Pro/ENGINEER Wildfire 2.0 Tips

W. Durfee

Keyboard/mouse shortcuts

CTRL+D	Default view
Middle-Button	Spin
SHIFT+Middle	Pan
CTRL+Middle	Zoom (press, drag up-down)
CTRL+Middle	Turn (press, drag left-right)
Scroll Wheel	Zoom
CTRL+G	Regenerate
CTRL+R	Repaint
CTRL+A	Activate window
---	Refit (no key, use toolbar icon)

Parts

To change background color: View > Display Settings > System Colors. Change Background and uncheck Blended Background.

To dimension: Dimension tool > L-click start > L-click end > C-click to place.

To dimension a diameter: L-click outside > L-click center line > L-click outside > C-click to place.

To assign a material and/or density: Edit > Setup > Mass Props > enter density (Aluminum = 0.095 lb/in³, Steel = 0.282 lb/in³). Units are current units for part. An even better way: Edit > Setup > Material > Define. Then Material > Assign.

To find weight and c.g. of a part: Analysis > Model Analysis > Model Mass Properties > Compute

To add relations: Double-click on part to show dimensions. Tools > Relations. Note that dimensions changed to symbols. In the Relations dialog box, use the toggle button to toggle between dimension symbols and numbers. Add relation like this (line 1 is a comment)
/* notch is 1/4 the width */
d3 = (d1)/4

To copy a feature: Edit > Feature Operations > Copy > Same Refs | Select | Dependent > Done > select feature (use of model tree to select) > Done > In GP VAR DIMS box select feature location dimensions to change > Done. Follow messages.

To mirror a feature: Edit > Feature Operations > Copy > Mirror | Select | Dependent > Done > select feature (use model tree to select) > Done > select plane to mirror about.

To make a pattern: Select feature. R-click > Pattern. (Or select the feature then the Pattern tool at the right.) Follow prompts to enter increment value, and total number of instances.

To set units: Edit > Setup > Units > select units > Set > read warning box

To edit dimensions: Double click on part then on dimension. Or right click on feature in menu tree > Edit. Regenerate when done.

To change dimensioning scheme or to re-sketch part: Right click on feature in menu tree > Edit definition. Takes you to dash board. Placement > Edit Sketch. In the Sketch dialog, select Sketch.

Raised text: Create protrusion on desired surface. Text tool in sketcher toolbar > follow directions, for text line draw bottom to top.

Hex object: 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.

To change model name: File > Rename.

To view order that model was created: Tools > Model Player. Click for/rev arrows to step through model creation.

To make a spring, use 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. 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 color a part: View > Color and Appearance. Follow directions in Appearance Editor.

Drawings

To select view, dimension, note or other object: Left click, object will turn red. Once selected can move or can right press to reveal pop-up menu of options.

Create drawing on C size, print to A size sheet.

To insert 3-D view: Insert > Drawing View > General. Click on drawing to place. In Drawing View box: View Type > pick orientation, Scale > pick custom scale. Display in No Hidden format.

To change obscure drawing parameters, with nothing selected, right press > Properties. From the top right menu pick Drawing Options. Scroll down the long list.

To change dimension font size, follow directions above under obscure drawing parameters.

To place text: Insert > Note > No Leader | Enter | Horizontal | Standard | Default > Make Note. Pick point in title block, enter text for note. Hit CR twice to finish. Done/Return to close note

window. Select and move note to center. Right press > Properties to edit or to change font size, type, alignment, etc.

To place text in a title frame: Use insert Note.

To change font size/style: Format > Text Style

To move a view: Select view (turns red) > L-press and move. If views won't move, click the padlock icon at the top

To change dimension to another view: Select (turns red), right press > Move Item to View > select new view.

To insert a dimension: 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: Insert > Reference Dimension > New reference. Create dimension.

To show the tolerance for selected dimension: Select dimension > R-press > Properties > Value and tolerance.

To show tolerance for all dimensions: View > Display Settings > Model Display > General tab > check Dimension tolerances > Apply > OK

To change anything about a dimension: Select dimension > R-press > Properties.

To change number of decimals in a dimension: Format > Decimal Places > follow directions in message area.

To change border frame: File > Page Setup. Select from drop-down menu under Format

To print: Print to PDF and view with Adobe Reader before printing. File > Print > OK. In the Print dialog, select Adobe PDF, then OK. View and print from Reader.

If border not coming out or if print is offset: File > Print > Configure . On Model tab, change Scale. Or on Page tab, change Offset. Experiment with the settings, checking results on the PDF file before printing a hard copy.

To insert table for BOM or change records: Table > Insert > Table. Follow instructions in message window. After table is created, double-click in cell to enter text.

To change directions of arrows: Select dimension (turns red). R-press > Flip Arrows

To change drawing scale: Double click on SCALE at bottom left.

To embed a dimension in a note: "This dimension is &D# ..." where # is the dimension number.

For multi-sheet dwg: Insert > Sheet.

To place more than one model on a drawing: File > Properties > Drawing Models > Add Model. Use Set Model to switch between the active model for view insertion.

Do not worry about yellow extension lines overlaying object lines; Pro/E will fix at print time.

If you want a different dimensioning scheme, go back to Part mode and redefine the feature.

To compute lengths or surface area: Analysis > Measure > follow directions.

To create raised text: Create a protrusion feature on desired surface. In Sketcher, select Text tool icon. Follow directions in message window. To place text L-click lower left then upper left of first character.

To change view type on 3-D view. Double-click view, View Display > Display Style = No Hidden

To print a drawing: On the top toolbar, click the Send to PDF button (not available on all systems).

Assemblies

To move a partially constrained part: Component Placement > Move > Translate.

To edit a part from assembly view: Right click on part in menu tree > Open. This opens part view for editing.

To change constraints: Right click on part in menu tree > Edit Definition.

To change mate offset parameter: Double click on part or right click on part in menu tree > Edit. Assembly parameter will appear in yellow, double click to edit.

Exploded view: View > Explode > Explode View

To move components in exploded view: View > Explode > Edit Position

General

Pro/E tutorials at <http://www.ptc.com/community/proewf/newtools/tutorials.htm#proe1>

Map keys (custom key stroke operations): 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 for map keys: sv = shaded view, wv = wire view, dat = datums on/off, fv = front view, dv = default view

Want others to see your parts, including spinning and zooming? Have them download ProductView Express from www.ptc.com. This is a web browser plug-in that enables viewing of Pro/E parts in Internet Explorer (not Mozilla). To use, open a part from Explorer.

Config file: For customizing Pro/E. Tools > Options. Deselect "Show only options" box to see all options. At top, change Sort from Alphabetical to By Category. After making changes, click the Save a copy button. To load config file, Tools > Options > Open a config file.