

## Modeling 3D



### Customizing Menus

Tools --> Customize Screen  
 Drag menu buttons to where you'd like them to reside.  
 example: Erase Not Displayed

### Display Datums while re-orienting model:

View -> Display Settings -> Model settings  
 Display while reorienting  
 Datums  
 {config.pro setting: spin\_with\_part\_entities = yes}

Performance:  
 Frame rates  
 Hidden Lines  
 Detail Levels

### Save Current Color Scheme

View --> Display Settings --> System Colors  
 File --> Save (File name.scl)  
 (system\_colors\_file)

### View Model Creation History

Tools --> Model Player

### Turn Off Threaded Hole Text Tags

Tools -> Environment  
 3D Notes (Remove Check Box)  
 OK

? Settings -> Tree Filter -> Check: Features & Annotations -> Expand tagged part -> Turn off Notes in expanded feature

### Edit Threaded Hole Text Tags

Selector = Annotation, Select note, RMB --> Properties  
 Can also edit in Drawing

### Turn Off Threaded Hole Cosmetics using Layers

Create a new LAYER  
 Name: COSMETIC\_THREAD  
 Rules -> Options -> Independant  
 Options -> Associative  
 Edit Rules ->  
 ■ Type  
 Comparison: is equal to  
 Category: All  
 Value: Hole

### Turn Off ECAD Cosmetics using Layers

Create a new LAYER  
 Name: COSMETIC\_PWB  
 Rules -> Options -> Independant  
 Options -> Associative  
 Edit Rules ->  
 ■ Type  
 Comparison: is equal to  
 Category: All  
 Value: Cosmetic

## Save Layer Status

Layer tree: RMB, Save Status

## Align a hole in an assembly with no relations:

Activate Part in the assembly to receive the hole

Insert Hole

Placement (tab bottom left)

Select primary Surface

Secondary References

Select Center Line of reference part

Change Offset to Align (both places)

Dimension Orientation Reference

Select any surface

(hole should now be aligned to CL of reference part)

Secondary References

Remove (references to CL)

Attach secondary references to datums or edges as desired

Set hole Diameter

Set hole Depth

## Copy Hole

Insert Hole

Place as desired

Select Hole

Edit --> Copy

Edit --> Paste Special

[ X ] Make copied dependant on dim of originals

[OK]

Select primary Surface

Secondary References

## Pattern Using Relations

Variables can be defined with numbers or formulas using Tools --> Relations

Within the pattern control panel, the DIMENSIONS pull up -

select the Increment dimension number and check box: Define increment by relation

Complete the pattern

Highlight the pattern feature on the part to show the pattern dimensions and locate the X HOLES dimension

The cursor popup will display a P-number. Add this number to the relations table

(for example: P208 = 6 or P208 = number\_of\_holes)

regenerate and pattern should now be using the variables.

## Draft Angle Greater Than 30 degrees

The draft angle is measured relative to the Draft Dir. If you insert a datum at an angle then use it as the direction, you can exceed the 30 degree limitation.

## Rename Parts in Assemblies

When renaming a part used in an assembly, have the assembly open and save after the rename.

If the assy isn't open, make sure it is in session (memory). Open and save before erasing from memory.

## Parts & Assemblies in the Search Path

If working with parts that are not in the working directory, set the config.pro search\_path

! list of directories to search path

search\_path c:\std\project1

search\_path c:\std\project2

search\_path c:\std\project3

## PreSelection Highlighting

Edit --> Select --> Preferences, clear check box

With Highlighting off, you have the option of using a Query Box

## System Color Map:

C:\Program Files\proeWildfire 2.0\graphic-library\appearances\appearance.dmt  
C:\Program Files\proeWildfire 2.0\graphic-library\appearances\DaveAppearance.dmt

### System Colors:

View -> Display Settings -> System Colors  
Scheme  
Use Pre-Wildfire Colors

### Save Color - Background

View --> Display Settings --> System Colors  
File --> Save (syscol.scl)  
{if you save to a different directory, change config.pro setting system\_color\_file}

### Black Edges (gray part)

View -> Color & Appearance  
Select "Black" color ball  
Assignment - Part  
Apply  
Clear  
Close

### Black Edges (colored part)

View -> Color & Appearance  
Select "Black" color ball  
Assignment - Part  
Apply  
Assignment - Surfaces  
Select surfaces  
OK (pop-up box)  
Pick Color Ball  
Apply  
Close

View, Display Settings, Model Display... Shade Tab, select "With Edges" box.

Step 1 : Checking the option "With Edges" by clicking #View #Display Settings #Model Display and in the Model display dialog box #Shade, the edge color will be the same as the model color.

Step 2 : Assign a designated color to the solid surfaces of the model by clicking #View; #Color and Appearances; Select a different color other than the model color; #Surfaces in the Assignment type; Select a surface #Hold the RMB; #Solid Surfaces; #Ok; #Apply. The color of the edges will be same as the original model color. The model color will be the new color assigned.

### Clear Surface Colors

View -> Color & Appearance  
Assignment - All Surfaces  
Clear  
Close

### UnHide ALL

View -> UnHide all

### Define Views

View -> View Manager  
Orient Tab  
RMB (right mouse button) **NEW**  
Enter name for new view  
RMB Redefine  
Note: Datum planes are yellow for the front, brown for the rear  
Select and define 2 datum planes for Front, Top, Right, etc as desired  
[OK]  
[Close]

### Define Cross Section View

View --> View Manager, Xsec, New, Enter, XSec Create  
Planar --> Single --> Done

### Display Cross Section

View --> Manager, Names  
Display --> Visibility

### ASSY VIEWS

FRONT = Front/Zo, Top/Yo  
RIGHT = Front/Xo, Top/Yo  
LEFT = Back/Xo, Top/Yo  
TOP = Front/Yo, Bottom/Zo  
BACK = Back/Zo, Top/Yo  
BOTTOM = Back/Yo, Bottom/Zo

### Select Query

Cursor over assembly - RMB - Query - Select from list window

### Hidden Datum in Group

Extrude - Pause - Define Datum - Resume  
Creates a hidden datum and the extrusion within a group

### Adding a CG Coordinate System

- 1) Insert->Model Datum->Analysis;
- 2) Check Model Analysis;
- 3) Give it a name in a specified field (press ENTER after) and click Next;
- 4) Select Model Mass Properties, click Compute and afterwards, Close;
- 5) In a Result params window, click Next;
- 6) In a Result datums window, select the datum desired (CSYS or point) and check Yes for each one;
- 7) Confirm and the datum is done.

### Display Message Log

Info --> Session Info --> Message Log

### Activating a Window Ctrl A

### GD&T - Adding a Datum Plane (fundamentals help file)

Create a Datum Plane and name it A, B, or C as required.  
Select Datum Plane and RMB --> Properties, Change from [ A ] to [ - A - ]

### GD&T - Adding a GTOL Symbol

Edit --> Setup --> Geom Tol --> Specify Tol  
Perp  
Model Refs Tab -- Reference to be selected, Datum, Select Entity (datum B)  
Datum Refs Tab -- Primary -- pull dn A  
[OK], Done/Return  
Done

tol\_display  
tol\_mode  
display\_tol\_by\_1000  
display\_dwg\_tol\_tags  
linear\_tol  
angular\_tol  
tolerance\_table\_dir

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## ASSEMBLIES

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## External Reference Control

Tools --> Assembly Settings --> Reference Control

Edit --> Setup --> (Part Setup) ↓ Ref Control

## Defining Assy Constraints

Placing a small part in a large assy avoid zooming in and out by creating empty constraints and editing them. Hit the green "+" (or middle mouse button) to leave the constraint incomplete and start another. Once they are defined, you can zoom to another area to add the other references. You just need to remember which ones to match up.

## Exploded-Move Multiple Parts

Preferences --> Move Many

## Assembly Relations

Original relations: D4=CWTDBG Current Relations: D4:1=CWTDBG

Since this is an assembly relation, the : colon is for the component ID. You could have several parts in your assembly that have a d4, for example, this is how you separate them.

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# SKETCHER

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## Turn OFF/ON constraints when inserting a line

Right Mouse Button while in line mode

## Line styles

Sketch-> [ ] Intent Manager (off) -> Geom Tools -> Cosm Font

or

Edit -> Line Style

## Ordinate Dimensions

Sketch -> Dimension -> Baseline (select geom to use as base) MMB

Insert Dimensions by selecting Base Dimension first (you can select more than one with ctrl)

(NOTE: Converting Driving dimensions to Ord in a drawing also converts the 3D model dims)

## Perimeter of an Arc

An arc can be controlled by dimensioning its length.

1 - Select the arc

2 - Edit --> Convert to --> Perimeter

3 - Select dimension that will become a variable dimension

BUG - you can only do this once in a sketch ?

## Dimension an Arc Angle

Sketch --> Dimension --> xxx --> Select one end, the other end, the arc, place the dimension

## Ordinate Dimensions

Sketch --> Dimension --> Baseline

Select geometry to us as a baseline (0.000), MMB to place the dimension.

Sketch --> Dimension --> Normal

Select the baseline dimension, then select entities to dimension, MMB to place the dimensions.

(More than one can be selected)

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# CONFIG FILE

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Tools --> Options

```
default_dec_places 5
param_dec_places 5
pattern_increment_dec_places 5
show_shaded_edges yes
sketcher_dec_places 5
prompt_on_exit yes
```



```
mapkey vf ~ Activate `main_dlg_cur` `ProCmdViewNamePick.view`1 ;\
mapkey(continued) ~ Select `nameviewlist` `nv_list`1 `FRONT` ;
mapkey vb ~ Activate `main_dlg_cur` `ProCmdViewNamePick.view`1 ;\
mapkey(continued) ~ Select `nameviewlist` `nv_list`1 `BACK` ;
mapkey vt ~ Activate `main_dlg_cur` `ProCmdViewNamePick.view`1 ;\
mapkey(continued) ~ Select `nameviewlist` `nv_list`1 `TOP` ;
mapkey vl ~ Activate `main_dlg_cur` `ProCmdViewNamePick.view`1 ;\
mapkey(continued) ~ Select `nameviewlist` `nv_list`1 `LEFT` ;
mapkey vr ~ Activate `main_dlg_cur` `ProCmdViewNamePick.view`1 ;\
mapkey(continued) ~ Select `nameviewlist` `nv_list`1 `RIGHT` ;
show_axes_for_extr_arcs yes
allow_anatomic_features yes
cmdmgr_trail_output yes
sketcher_refit_after_dim_modify no
hole_diameter_override yes
parenthesize_ref_dim yes
edge_display_quality very_high
prehighlight yes
ecad_comp_csys_def_name ecad_default
ecad_mapping_file D:\ProE\pro\mimu\BoardTransfer\sk_ecad_hint.map
ecad_area_default_import cosm_area
display_axis_tags no
model_note_display no
save_drawing_picture_file both
mass_property_calculate automatic
system_background_color 0 0 10
display_planes yes
display_axes no
display_coord_sys no
display_points no
pro_group_dir D:\ProE\UDF-Library
pro_material_dir D:\ProE\Materials-Library
pro_note_dir D:\ProE\Notes_Directory
start_model_dir D:\ProE\StartPart
system_colors_file D:\ProE\Colors\syscol.scl
trail_dir D:\ProE\Trail_Files
spin_with_part_entities yes
```

? maintain\_limit\_tol\_nominal

Shoe/Erase setting 'select to remove' or 'select to keep' defined by show\_preview\_default = keep

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## REFERENCE MATERIALS

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There are Wildfire 3.0 books available - just search on [www.amazon.com](http://www.amazon.com) for Pro/ENGINEER - Roger Toogood has written a few, plus others - I can't personally vouch for them as I've not seen them.

There are many other ways to learn Pro/ENGINEER:

<http://www.ptc.com/appserver/mkt/products/home.jsp?k=3303>

[http://www.ptc.com/community/resource\\_center/proengineer/](http://www.ptc.com/community/resource_center/proengineer/)

<http://www.ptc.com/products/tutorials/index.htm>

[http://www.ptc.com/community/express\\_archive/index.htm](http://www.ptc.com/community/express_archive/index.htm)

etc.

The Knowledge Base is also a fantastic resource - the first place I look if I don't know the answer, and there's excellent suggested techniques etc:

<http://www.ptc.com/appserver/cs/search/search.jsp>

Other forums/websites:

<http://www.eng-tips.com/threadminder.cfm?pid=554>

<http://www.mcadcentral.com/proe/>

<http://www.proengineertips.com/>

[http://www.cdcweb.co.uk/index.php?page=tutorials\\_wildfire.php](http://www.cdcweb.co.uk/index.php?page=tutorials_wildfire.php)

<http://www.mcaduser.com/>

<http://www.proesite.com/>  
<http://www.e-cognition.net/>

There are other good sites - my apologies to those I've missed.

In my early days of learning Pro/ENGINEER I read through the help manuals, and I think that's still a great idea: From Pro/ENGINEER:

Help > Help Center > Fundamentals > Pro/ENGINEER Fundamentals

Also Part Modeling, Assembly Design, etc will certainly improve your understanding.

Cadquest books are really good.

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Is there any way to create a hole callout in the draft environment which utilizes the &Screw\_Size, &Thread\_Depth, etc parameters? I know you can make a 3D annotation in the model, but I would like to be able to do it from the draft environment.

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>sure, find out the dimension names and use 'em in the note

I usually use the screw size dimension and add the depth and B.C. and any cbor dimensions and the like in this way:

```
(&d) THRU  
{cbor symbol} &D# {depth symbol} &D#  
&D#{degree symbol} APART ON A &D# B.C.
```

where &d is the screw size and all the '#' are referenced to whichever dimension ID this particular feature created

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???  
Pen Tables

I got all my info. from the help center. I did a global search for table.pnt. This file should be located in your loadpoint for pro-e.

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config.win file ? set window layout?

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## LICENSE SERVER

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### License Status

```
Open DOS Window shell Start > Run > cmd  
c:  
cd \EDS\I-Deas10\sec  
Imstat -a -c 7655@az18nt1971.honeywell.com  
Imstat -i -c "
```

```
C:\ptc\proiclient3.3\bin\ptcsetup.bat  
C:\Program Files\proeWildfire 2.0\bin
```

### Intralink - resetting workspace directory after reinstalling....

RMB the proIntralink desktop icon - Edit - add the following  
set pdm\_ldb\_path=D:\ProE

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## ProE Help Line

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1-800-477-6435

Service Contract Number 2A112296 (use 2 for A)

Phone sequence to get to mcad help

```
Mechanical cad support "1"  
ProE "2"  
ProE model "1"  
ProE drawing "2"
```

```
Data Management "2"  
Intralink "2"  
Intralink "1"  
Intralink "2"
```

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## Sheet Metal

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[http://www.me.cmu.edu/academics/courses/NSF\\_Edu\\_Proj/Wildfire\\_short\\_course/tutorial9.htm](http://www.me.cmu.edu/academics/courses/NSF_Edu_Proj/Wildfire_short_course/tutorial9.htm)

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## Castings

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see help file for Inheritance Features

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## Misc

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Digital Process Design - enables automatic model creation, updating and deployment without any re-work. Combined with Horizontal Modeling Techniques simplifies design changes. Eliminates downstream problems with reorder, suppression and feature deleting by eliminating interrelated parent/child build steps.

1-Carefully choose references. This includes references for sketching planes, sketch orientation, sketcher references, edges or surfaces for rounds, etc..

2-Choose references that follow the design intent. Though it's easy to make all your features reference only the base datum planes, the model won't follow when modifications are made. Choose references that allow the model "move" with the intent as changes are made.

(In Practice: Don't just accept Pro/E's automatic references (especially in Wildfire). Think about the design intent and choose references that are stable and reflect that intent. Wildfire is horrible about automatic references. Unless you're sketching on a default datum in the assumed orientation, automatic references are rarely applicable or appropriate. So much for minimizing mouse picks.)

3-Choose references that won't disappear. References like edges that disappear when rounded, are not usually the best choice. Datums and planner surfaces are typically better. References from base features are typically more stable than those of later ones.

(In Practice: Pro/E provides different methods of selecting references that give similar results, and some are more stable than others. For example, surface selection by Surf & Bnd, or Loop Surfs when selecting for draft (and other times). Select using Intent Chain (May 2002) for edges. These are just a few examples. Choose the best method for your application.)

4-Choose references sparingly. More references mean more feature interconnectivity which can make the model more difficult to work with. However, choose enough references to make the model follow design intent.

(In Practice: Some models are so tightly tied with references that the model fails with almost any modification. (August 2002) Users wonder why Pro/E is so hard to work with, but in fact, it's the way the model was constructed. Choices like Thru Next have no references.)

5-When several features are to reference the same thing (like a planner surface or axis), create datums for control. (June 2001)

(In Practice: Key datums (planes, points, axes) are easier to find and select when they are named. Named features also denote significance for someone changing the model later. Name your important datums.)

6-When several aspects of a part must interact (like cuts and protrusions to allow wall thickness and spacing) build control features like curve sets to manage the interactions, then reference them with the features.

7-Create relations that associate features when direct references are not practical.

(In Practice: When writing relations, use comments for "what" and "why". If other people use the model, they'll be more impressed if you are specific in your comments.)

8-Drafts and rounds are often best left to the end of the model. Although this is a good rule, there are times when they are required earlier. Care should be given to where the features are inserted in the model tree.

9-Rounds should normally be inserted as round features rather than put in a sketch. Again, this is a good rule of thumb, but there are times where a round can't be created or a dimensioning scheme (design intent) requires the radius in a sketch. As above, carefully consider when. (The same is true for Drafts, Chamfers, etc.)

(In Practice: It is often good to build models without drafts and rounds, then go back using Insert Mode to put them in -- where possible clustering them with the parent features.)

10-When the above guidelines don't make sense, carefully consider your options.

(In Practice: Good modeling practices are far more important early in the model than at the end when you're trying to put in the last round or draft.

### **Menu Buttons - Change Default**

In a part start a sketch feature and once you are in the sketch environment select Tools>Customize Screen. Under the Commands tab select Sketch to add sketcher tools. You can select items on the tool bar and drag them to the locations you want. You can also add new menus and flyouts from the New Menu selection at the bottom of the list. Make sure you have the check box Automatically save to selected and the file set.