

Tutorials for Pro/Engineer Wildfire 2.0

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1 About the Pro/Engineer Wildfire 2.0 Tutorial

1.1 What is Pro/ENGINEER®?

Pro/ENGINEER is a feature-based, parametric solid modeling system with many extended design and manufacturing applications. As a comprehensive CAD/CAE/CAM system, covering many aspects of mechanical design, analysis and manufacturing, Pro/ENGINEER represents the leading edge of CAD/CAE/CAM technology.

1.2 Conventions Used in this Tutorial

		Example
UPPERCASE	The name of a menu, window or dialogue box	FILE, FEAT, MODEL TREE
Boldface type	An item to be selected from a WINDOW or a MENU	open, save
[bracketed] text	information entered from the keyboard	[500]
<Text between these symbols>	Name of a keyboard key	<shift>, <Enter>, <Space>
<i>Italics</i>	A naming Convention	<i>partname.prt</i> indicates the name of a specific part will be substituted where partname.prt occurs
>	For a series of actions / commands performed in MENUS. The process of steps is linearly downwards with each new set starting on a new line	FILE > New
/	For a series of actions / commands performed in MENUS, a back slash is used when the actions are performed within the same MENU box	Protrusion / done / one side / done
<keystroke> + mouse	Simultaneously press and hold a specified key while selecting with a specified mouse button (default left)	<shift> + click

The following terms are used frequently in this tutorial:

Choose	Click the {left} mouse button on a MENU option, a pull down MENU, or a DIALOGUE BOX.
Select	Click the {left} mouse button on geometry in a model or drawing, or on an object in a database.
Pick	Click the {left} mouse button on a specific point or location.
Click	Single click the {left} mouse button on an icon, button, box, or hyperlink.
Object	An assembly, part, drawing, or set of metadata within a database.
Model	An assembly or part in a Pro/ENGINEER environment, or a graphical representation of a computer program.
Mode	A group of closely related functions in a Pro/ENGINEER environment.

1.3 About this Tutorial

This tutorial is introduced based on the collective efforts of many people over a number of years during the teaching and learning of CAD. Part of it came from previous versions of Pro/E Tutorials at UVic. Prof. Gary Wang of Department of Mechanical and Manufacturing Engineering, the University Manitoba rewrote many sections. Mr. Minh Ly also contributed to a number of sections. Their efforts are gratefully acknowledged.

Rather than competing with other comprehensive Pro/E tutorials available on the internet, we intend to provide a number of short tutorials that go over the basic functions of several basic Pro/E modes to allow a user to have a quick start. Pro/E on-line manual, other on-line tutorials, and reference books, listed at our website, provide more detailed explanations and practices on various Pro/E function modules.

2 Introduction to Pro/E WILDFIRE

This section is intended to briefly explain the Pro/E User Interface and get you started with a simple modeling task. The steps needed to start Pro/E and to generate a part model is discussed in the following tutorials.

2.1 Starting Pro/E

To start Pro/E on a Windows machine, there may be an icon on your desktop or you may have to look in the Start menu at the bottom left of the screen on the Windows taskbar. The program takes a while to load, so be patient. The start-up is complete when your screen looks like the following figure, which is a default Pro/E screen.

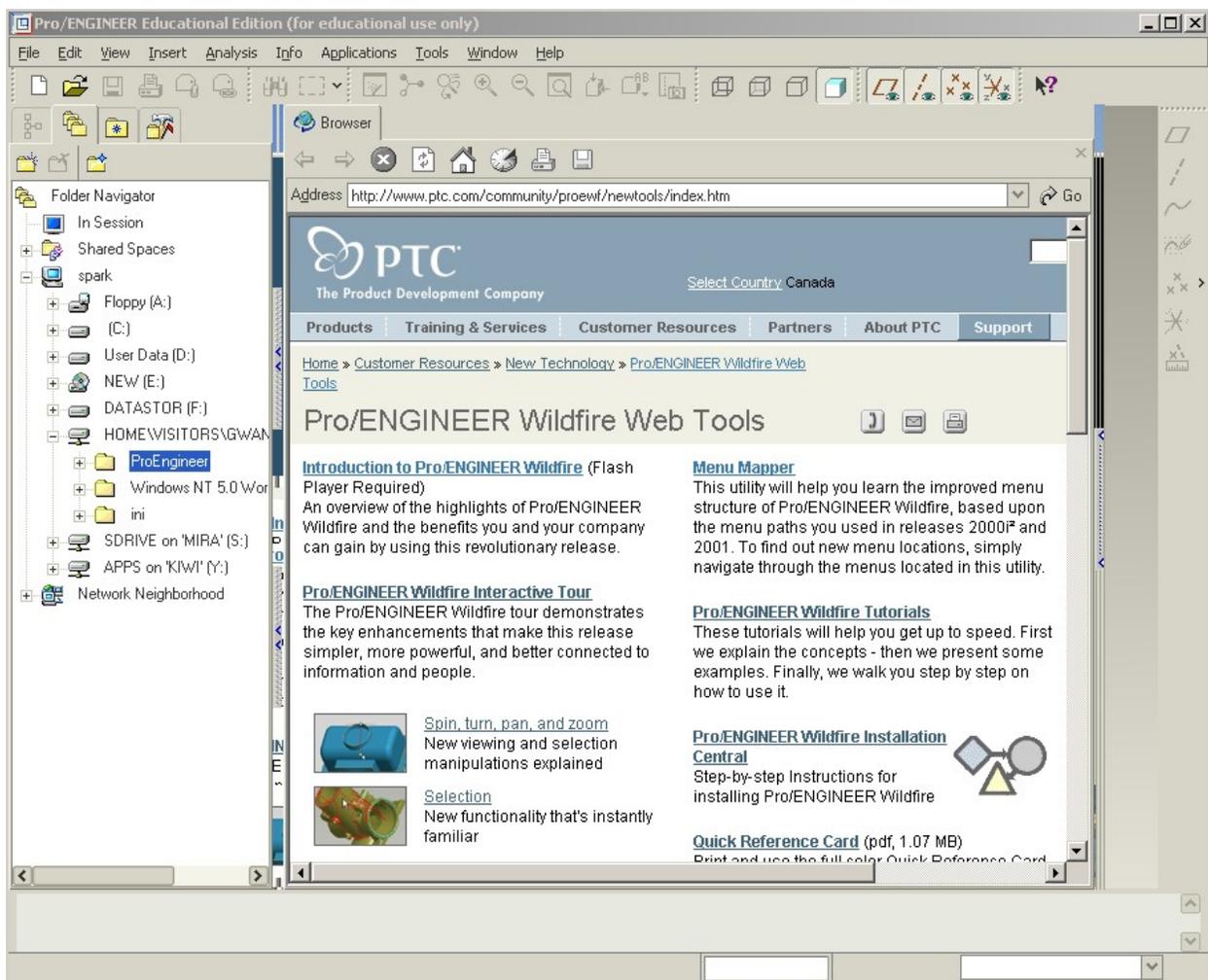


Figure 1 The default Pro/E Wildfire screen.

Now, look for the icon  under your menu to start a new application. Press the icon; or you may use the menu **FILE** > **New**. Either way, you should be able to launch the following window.

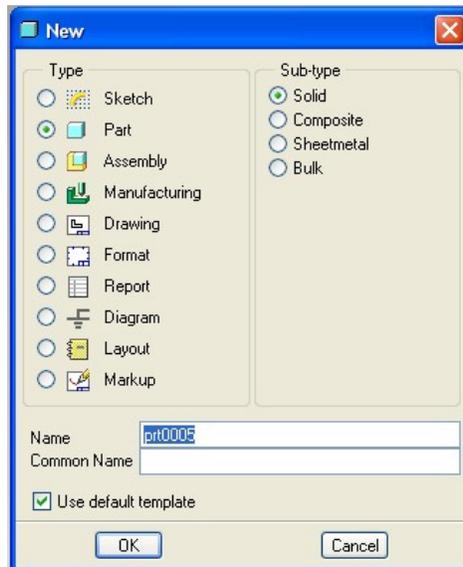


Figure 2 The pop-up window to start a new application.

You may type the name [housing] to replace the default name “prt0001”. In this section, we are going to create the first feature of a part called “housing”, which is one of the components of a disc brake assembly that we are going to create in the lab. The focus of this section, however, is on the introduction of Pro/E environment rather than the modeling techniques. More modeling techniques will be described in later sections.

After clicking the **OK** button, you should see the window shown in

Figure 3, which is pretty much self-explanatory. You are encouraged to move your mouse cursor on top of each shortcut button and read the description from the command description window. The filter setting selection is for the convenience of picking a feature on the main graphics screen. The default (or the lazy way) is to leave it as **Smart**.

HINT: DO NOT resize or move the main or menu window. If you start messing with the window size and placement, sooner or later you will bury a command menu behind other windows, then suddenly the computer seems frozen and you are stuck there! So before becoming an expert, you'd better let Pro/E do its own window management. This also tells you that if the computer seems frozen, try to move the windows around to see if some menus are hidden waiting for your mouse click.

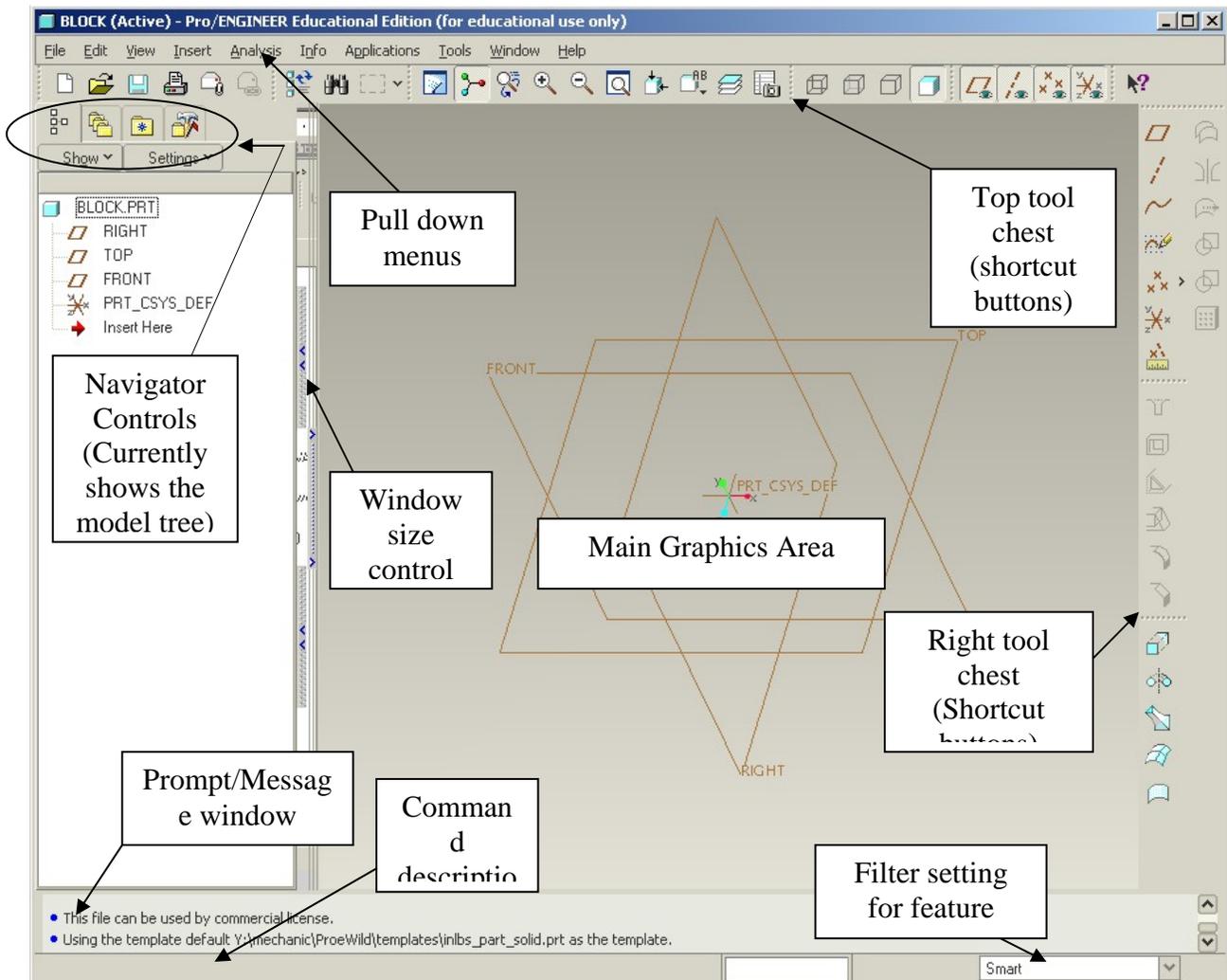


Figure 3 A description of the Pro/E screen.

2.2 Mouse Functions

Before we start with the hard job (modeling), you should know about some tricks of the mouse. Wildfire is meant to be used with a 3-button mouse. If it has a middle scroll, it is actually better and you are lucky. If your mouse is a 2-button one, try to use the <shift> key plus the left mouse button (LMB) simultaneously as an equivalent to the middle mouse button (MMB). If it doesn't work, talk to your system administrator.

Most selections of menu commands, shortcut buttons, and so on are performed by clicking the left mouse button (LMB). In this tutorial, whenever you “select”, “click”, or “pick” a command or entity, this is done with the LMB unless otherwise directed.

The functions controlling the view of the object in the graphics window are all associated with the MMB. These are the important Spin, Pan, and Zoom functions. The following table summarizes different uses of mouse buttons that can make your job easier and more fun. Note: if you know previous versions of Pro/E, you will find the mouse functions are quite different! Learn the new functions and don't let your experience frustrate you.

Table 1 Common mouse functions in Pro/E Wildfire.

Function	Operation	Action
Selection (click left button)	LMB	Entity or command under cursor selected
View Control (drag holding middle button down)	MMB	Spin
	<Shift>+MMB	Pan
	<Ctrl>+MMB (drag vertical)	Zoom
	<Ctrl>+MMB (drag horizontal)	Rotate around axis perpendicular to screen
	Roll MMB scroll wheel (if available)	Zoom
Pop-up Menu (click right button)	RMB with cursor over blank graphics window	Launch context-sensitive pop-up menus

HINT: If your mouse seems "dead", and so are the menus and toolbars, check the message window; Pro/E is probably waiting for you to answer its prompts.

How to Get On-line Help

Oops, there is one more thing to say. As any tutorial may not cover everything and some of the problems in the lab are very creative, both you and your TA/tutor will sometimes need to get the online-help. The Help function gets more important as you work on your own assignments and projects. OK, there are several ways to do this.

- ❑ Choose **Help > Help Center** to launch a browser, which lists many help items, including tutorials and step-by-step description of all the commands.
- ❑ Click the **Context Sensitive Help** button towards the right end of the top toolbar. Its equivalent is **Help > What is this?** Then click on any command or dialog window. (Can you find the button? If not, you didn't browse through the buttons. Please use your mouse cursor to go through those top toolbar buttons and read their description in the message window.)
- ❑ If your problem gets very tricky, you might need to register on-line at www.ptc.com as a user and get help from the knowledge base created by the Pro/E user group. Before you go through this route, talk to your TA as he/she may know the answer to your problem.

2.3 Begin to work in Pro/E

Now back to

Figure 3 where we left off. The left side of the main window shows the model tree of the empty part “housing”.

The main graphics windows shows three orthogonal planes, named TOP, FRONT and RIGHT, and a coordinate system. These planes are called datum planes, representing the 3-D world. These planes are very useful as reference planes when creating features and assembling components. Their advantages are not obvious when modeling simple parts, and in fact new users find these planes annoying. Whatever you feel now, my advice is to get yourself used to these “annoying” planes.

1) Prepare for sketching

Click the Extrusion button as shown in Figure 4.

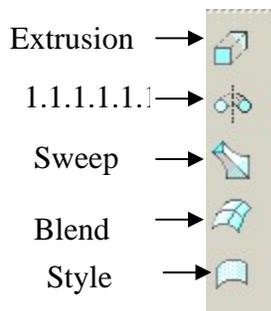


Figure 4 The Sketched Features toolbar.

Then you will see

Figure 5 at the bottom of the main window.

Figure 5 only explains the buttons that will be referred to in the tutorial. You should exercise moving the mouse cursor again to each button and read the description in the message window to find out about other buttons.

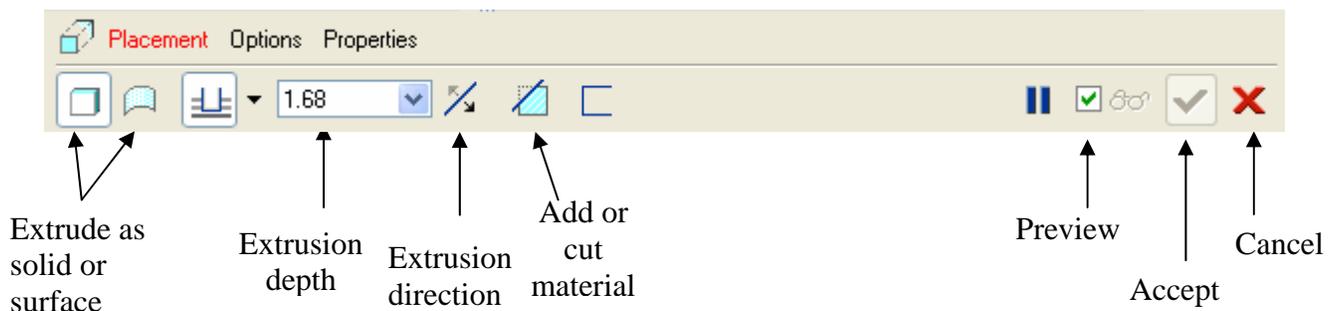


Figure 5 Extrusion dialog window.

Click the **Placement** button as shown in Figure 5, then click **Define**. A pop-up window will show up as Figure 6.

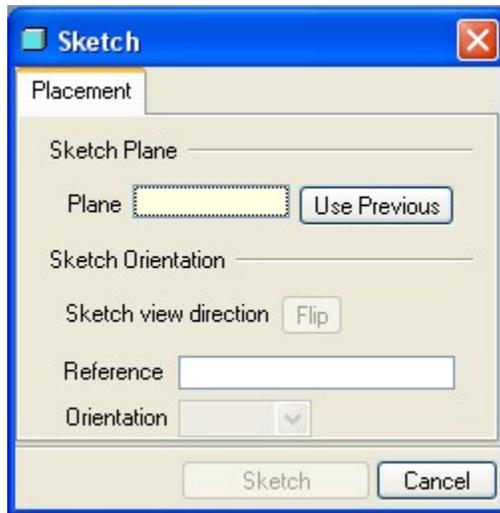


Figure 6 Sketch view set-up window.

Now go to the main graphics window, click the FRONT plane either on the word “FRONT” or any side of the plane. You will see the first blank in Figure 6 is filled with **FRONT** (ignore the words after **FRONT**; the same applies for other blanks in this window). This plane is chosen as your paper that you can sketch on. Image you are drawing a picture. After picking the paper, you have to place the paper in the right orientation so that you are either in a portrait or landscape view. That is why there is a reference plane as shown in the second blank in Figure 6. In this case, Pro/E should automatically fill in **RIGHT**, which means the **RIGHT** datum plane is chosen as the reference plane and it faces the right of your paper, which is filled in the third blank in the figure. Now, click **Sketch** button and you will be brought to a new window environment.

The Pop-up window named **References** appears and lists **RIGHT** and **TOP** datum planes as references. In the mean time, the two planes are shown in the main graphics window as two perpendicular lines and two brown infinite dotted lines override them. These two references are used as references for dimensions. As you may appreciate, no matter what you draw on the paper, you have to know the relative position of your drawing on the paper. This seems very obvious in a real drawing because human beings do all these things intuitively. But computer needs you to specify these. Of course, one may deliberately select a particular reference plane. As a starter, we just accept the default choice and simply click **Close** on the window.

2) Sketch the geometry

For the sketch, you pretty much work with two groups of buttons. The first group is to control the views of sketch, as shown in Figure 7. The second group is the sketch toolbar buttons, shown in Figure 9.

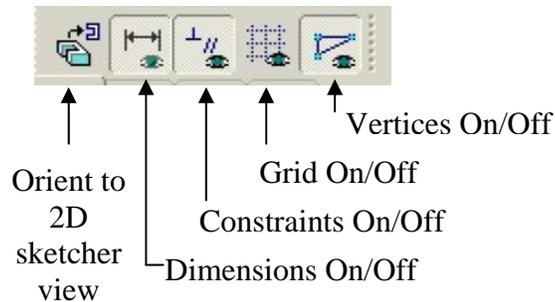


Figure 7 Control buttons for sketch views.

The view control buttons can help you set the proper view, clean the view, assist sketching, etc. I found the first button is very useful as I had the habit of using scroll ball to spin the geometry. So the first button can always bring me back to the paper (the sketch view). The **Grid On/Off** is often used as the grid can help the sketch. Now click on the button to turn the grid on. You should see Figure 8.

The second group of buttons shown in Figure 9 allows you draw different features. Some of these commands are very obvious, e.g., creating lines, circles, etc. Some are not. These commands may be explained later in this tutorial.

- Draw the profile

HINT (Sketch tips):

- a) *Keep sketches simple; try NOT to include rounds, chamfers, etc. in your sketch. This makes the final model flexible and helps regeneration.*
- b) *Do not sketch to scale*
 - *Firstly, concentrate on getting your geometry straight by sketching large*
 - *Secondly, resolve the sketch by modifying dimensions*
- c) *Use the grid as an aid*
 - *Create lines of equal length, parallel, or perpendicular*
 - *Align sketched entities*

Now click the right arrow beside the **Create Arc** button, as shown in Figure 9, choose the button  with the description “**Create an arc by picking its center and end points**”. Then click your cursor on the bottom side of the vertical reference (the dotted line) as the center point, then click the left side of the horizontal reference, and finish with the click on the upper part of the vertical reference. **NOW CLICK THE MIDDLE MOUSE BUTTON TO GET YOURSELF OUT OF THE CURRENT DRAWING MODE.** The MMB is used for canceling the current drawing mode for other commands as well. You should have drawn an arc. Don’t worry about the dimensions; just get the shape right at first.

Repeat the same step by clicking on the same center point but with different endpoints. The two endpoints should right above the end points of the first arc, respectively. You should have a concentric arc similar to the first one.

Draw two lines to connect the respective endpoints. You should have an enclosed profile with two concentric arcs and two vertical lines. The graphics window should look like that in Figure 10 (don't worry about the dimensions!).

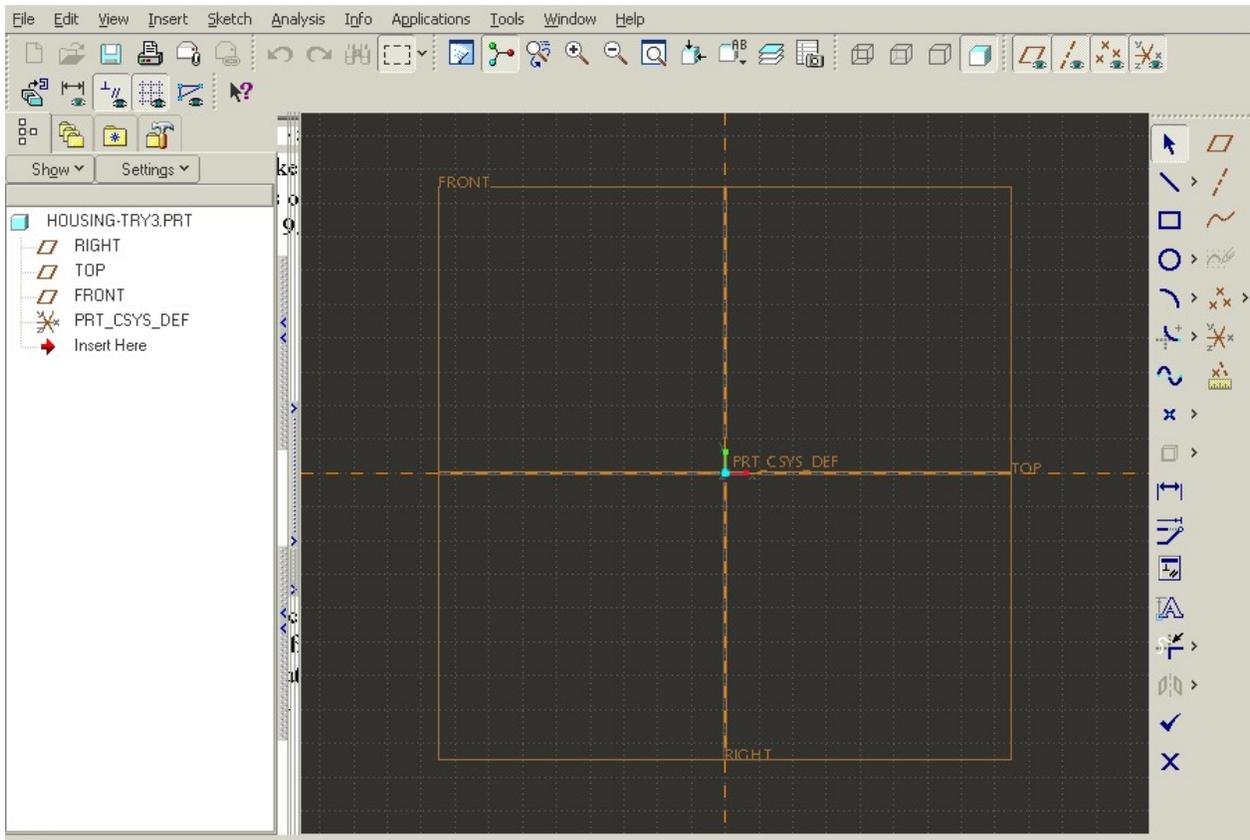


Figure 8 The sketch plane with grid on.

1) Re-dimension the geometry

In Figure 10, there are some gray dimensions. If you don't see these, click on the **Dimension On/Off** button, as shown in Figure 7. These dimensions are automatically added by Pro/E. Now, re-dimension the geometry and modify these dimensions to your desired ones.

Click on the **Dimension** button on your right toolbar as shown in Figure 9. We will specify two dimensions; one is the total horizontal length and the other is the height of the left vertical line. Click the two vertical lines using your LMB, and move your cursor to the middle of the two lines and click the MMB. You should see a horizontal dimension, indicating the length of the profile. Then click on the left vertical line using LMB, then click the MMB to place the second dimension.

Click on the **Modify Dimension** right below the **Dimension** button. Pick the horizontal dimension; you will then see a pop-up window. Deselect **Regenerate**; enter the value [45]. Then click the line length dimension, enter [10]. Pick the gray radius dimension for the arc, enter [210].

Then click the check mark button. The geometry will be regenerated with new dimensions, as shown in Figure 11. The message window will show “Dimension modifications successfully completed.”

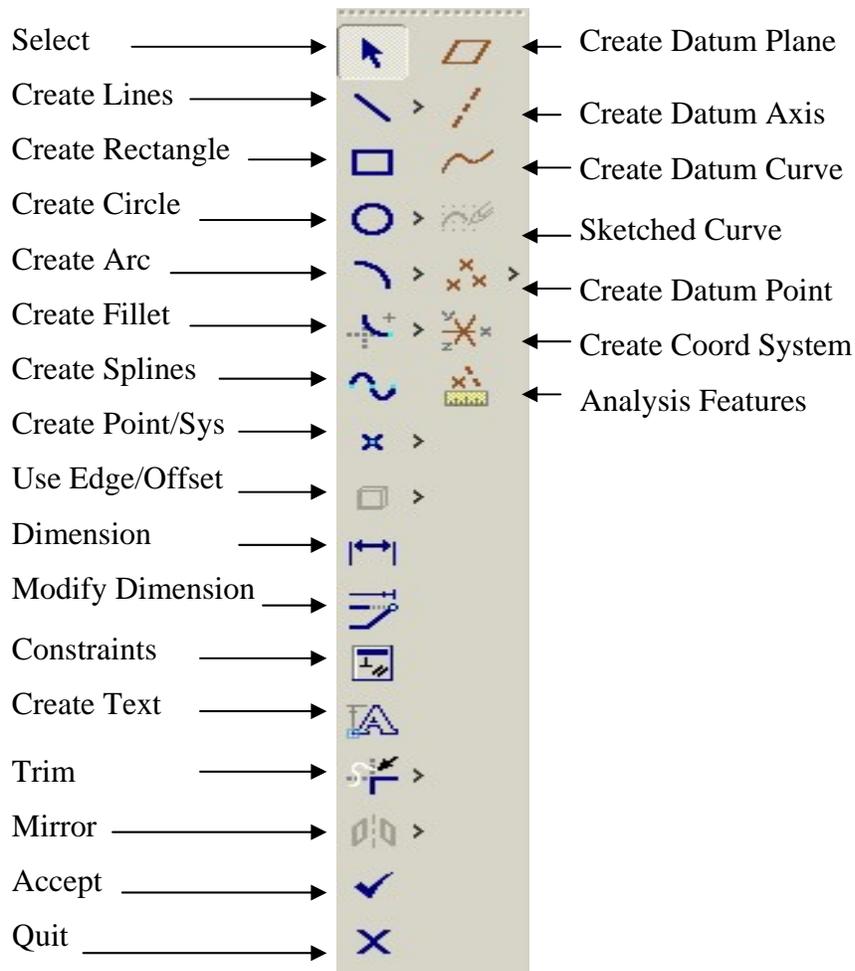


Figure 9 The sketch and datum toolbar.

Now, you can select the **Accept** button shown in Figure 9. The geometry turns pale yellow. Back to the buttons shown in

Figure 5, from the **Extrusion depth control** button, pick the alternative “Extrude on both sides”, then enter [31] in the blank besides the button. You then click the **Preview** button to see the geometry. Remember to practice your mouse functions to spin the geometry around! If everything is fine, you can then click the **Accept** button. And yeah, you are done! The final screen output should look like Figure 12. Though the geometry seems very simple, you should be very proud of yourself because you have just learnt to....

- ❑ Understand tricky datum plane conventions
- ❑ Understand the significance of references planes and the sketcher, which is sometimes

difficult for new Pro/E users

- ❑ Understand the parametric basis of Pro/E, i.e., the dimension drives the geometry change, and
- ❑ Create an extrusion, which is the most popular command in Pro/E.

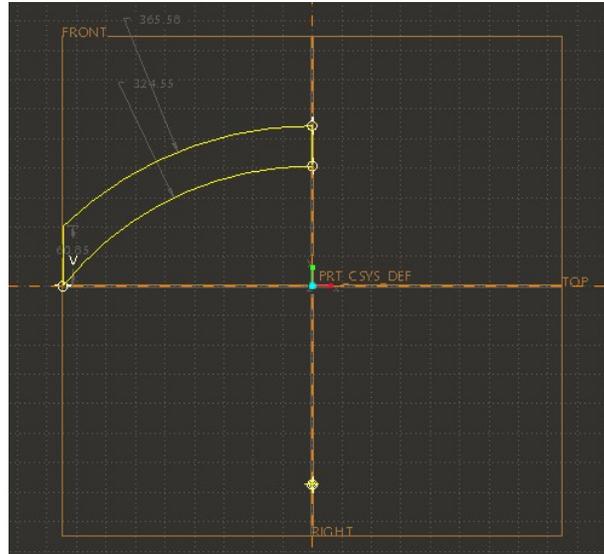


Figure 10 The profile.



Figure 11 The modified dimensions of the profile.



Figure 12 The first feature of the part Housing.

OK, you seem to be getting impatient. Well, fine, I may be very verbose when explaining the first feature. After that, this tutorial will become sketchy and sloppy. Please be patient with me

since the first is always the hardest, and you won't be able to enjoy this detailed information before long. Also if you want save time at the beginning, you might end up spending more later.

3) Redefine the feature

In case you messed up the part and cannot get the one shown in Figure 12. Don't panic. Click the **Extrude 1** feature (or even the sketch feature under this extrusion feature) in your Model tree window using the RMB. You will then see a bunch of commands including **Edit**, **Edit Definition**, etc. The **Edit** command allows you modify dimensions in 3D mode and the **Edit Definition** command brings you back to the sketch and the extrusion definition environment. You can then correct the steps that have been messed up with and follow the instructions in this section to get it right. Another way to modify a dimension is to double click a feature in the main graphics window; all the dimensions relevant to the feature will show up. You can double click the dimension you want to modify and enter a new number. Then click the **Regenerates Model** button (To use this function, make sure the Filter Setting at the right bottom corner of the window is turned to **Features**).

4) Save, view, and print the model

Pro/E, unlike other Windows applications, does not automatically save your work. You have to remember to do that. If you leave the program without saving your new work, it is basically gone! Anyone who says that they have never lost work this way is probably lying! Click **FILE> Set Working Directory** to change the default directory to a subdirectory under your home C:\25.353\start directory. By doing this, you can keep the default Pro/E directory tidy and avoid someone else accidentally deleting your file.

HINT: Save your model frequently to avoid loss of work.

Now, you should play with the buttons in the top tool chest.

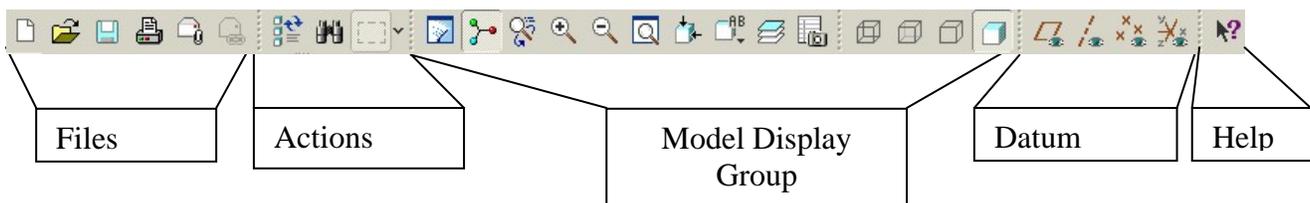


Figure 13 Groups in the top tool chest.

As shown in Figure 13, these buttons can be grouped to five groups. Buttons such as **Regenerates Model**, **Redraw the current view**, and **Refit object to fully display** are commonly used ones. Can you find them? Please note these toolbar buttons will change, depending which mode you are in. Examples of different modes are part modeling model, drawing mode, assembly mode, sketch mode, etc.

You can use **FILE > Print** to print your model, or **FILE > Save a Copy** to print it as a picture or

formats readable by other CAD tools. Or, you could simply use the <Print Scrn> key on your keyboard and then use Microsoft Paint to convert it into a picture file.

3 Modeling a Complete Part

OK, assuming you

- ❑ Have familiarized yourself with the Pro/E environment and you did either view and/or try all the buttons, icons, etc.
- ❑ have built the first feature alright, and
- ❑ can build the first feature again without reading the tutorial

If the answers to all the above are YES, then move on. Otherwise, go back to the previous section until the answers are YES. Because the rest of tutorial will be sketchy and, maybe, sloppy. You will be very frustrated if you didn't do the first section well.

The part, *housing*, that we are going to build is shown below:

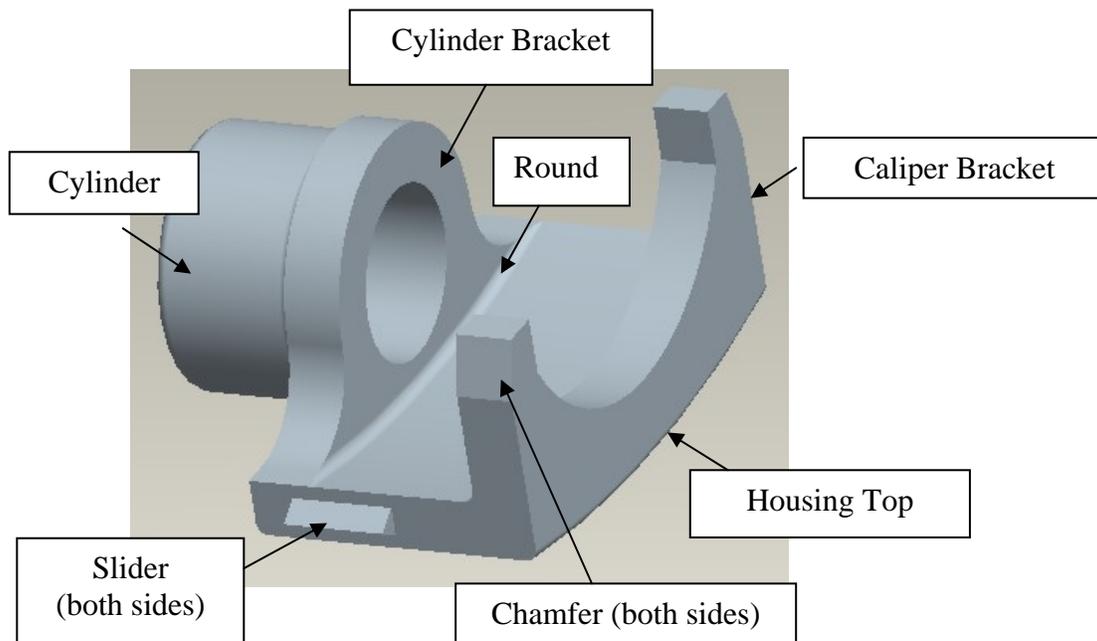


Figure 14 An illustration of the housing part.

This tutorial will guide you in modeling, one-by-one, the different features shown in Figure 14.

3.1 Complete the Housing top

In the last section, you have modeled only a half of the housing top. Please open this part called *housing*. Its file name suffix is “.prt”, which indicates that it is a part model.

We are going to model the other half of the feature by performing a command called “mirror”. The logic of the action is 1) pick the feature to be mirrored, and 2) pick the “mirror”.

Choose **EDIT > Feature Operations**. Pick **Copy** from the pop-up Menu Manager FEAT window. Click **Mirror / Select / Dependent / Done**. Pick the feature in the main graphics window, then **Done**. Then you see a pop-up window called SETUP PLANE window. You pick the RIGHT datum plane in the main graphics window. The complete housing top should be completed, as shown in Figure 15.

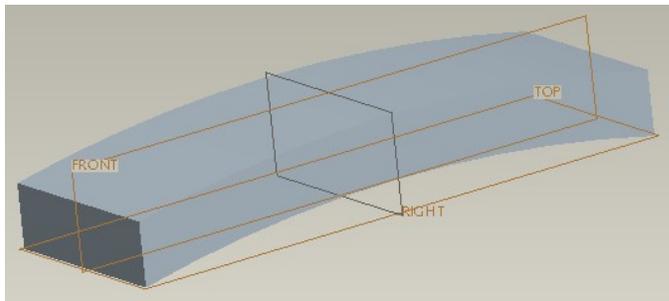


Figure 15 The housing top feature.

3.2 Build another extrusion: cylinder bracket

Prepare for sketch

Since it is another extrusion feature, please review the steps talked in detail in the previous section. Let's review it.

Choose the extrusion icon in your right tool chest. Click the “Create a section or redefine the existing section” icon in your bottom left tool chest. Now, pick the long side plane of your housing top as the sketch plan; accept the default reference plane (is it RIGHT plane? It should be).

Draw a sketched section

Click the “Create an arc by 3 points” button . Locate the two endpoints in a horizontal line below the horizontal dotted-line (turn on the Grids to help you position the endpoints). Locate the center at the vertical dotted-line below the two endpoints.

Use the same button to draw another arc tangent to the arc and the horizontal reference line at the left side. You may find that the arc is not shown tangent to the horizontal reference line (if they are tangent, there should be a small symbol “T” close to the tangent point). Click the  icon, then pick the . Select the new arc and the horizontal line. The small “T” should show up. This means the two entities are now tangent. Then dimension the two arcs as shown in Figure 16.

Draw a line to connect the tangent point with the left bottom point of the housing top feature.

In this exercise, we are going to practice using the “mirror” tool in sketch mode. First you should sketch a centreline which represents the “mirror” plane. Find and click the “Create 2 point centreline” button  and draw a line coinciding with the vertical dotted-line (remember what is this?).

Pick the new arc and the line (hold the <ctrl> key for multiple selections). Then click the mirror icon . These two entities should be copied to the right hand side.

Now, click the  button, and then pick the top curve of the housing top feature. You should see this curve turns yellow. Continue to pick the two sides of the housing top feature. Close the pop-up Type window. By now, you should have a closed sketch section. Dimension the section as shown in Figure 16.

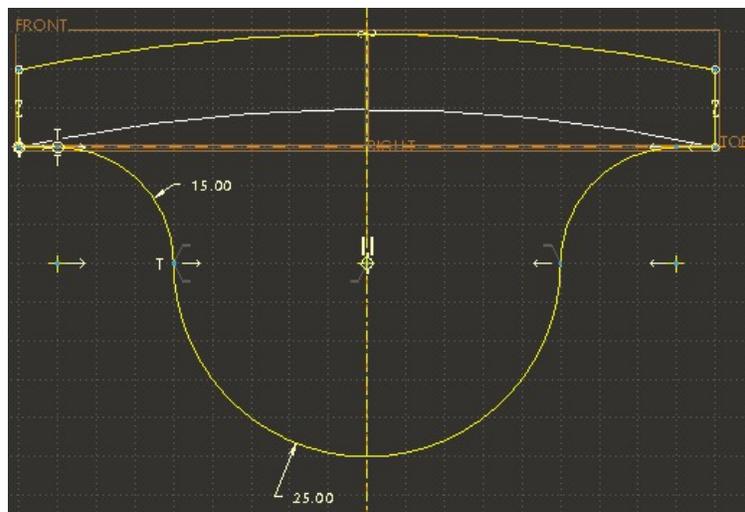


Figure 16 The complete sketch section for the cylinder bracket.

Click the “Accept” button to finish the sketch. Then go the toolbar shown in Figure 5, enter the extrusion depth [10]. Practice using the “Preview” button to preview the extrusion before accepting it; so you can correct any mistakes. Also play with the “Extrusion

direction button” and the “Add or cut material” button, and use the “Preview” button to get a feel what happens. If the preview looks fine, then click the Accept button.
The final model should look like

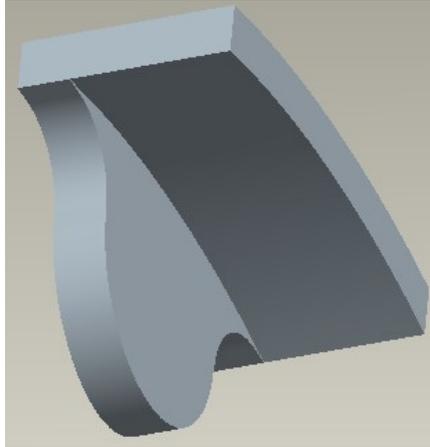


Figure 17 The housing top and the cylinder bracket.

3.3 Build another extrusion: caliper bracket

Now we repeat the same process to build another extrusion. Get ready for sketch. This time, pick the other side of the housing top surface as your “paper” (sketch plane). Remember to create a mirror centerline during Step 4.

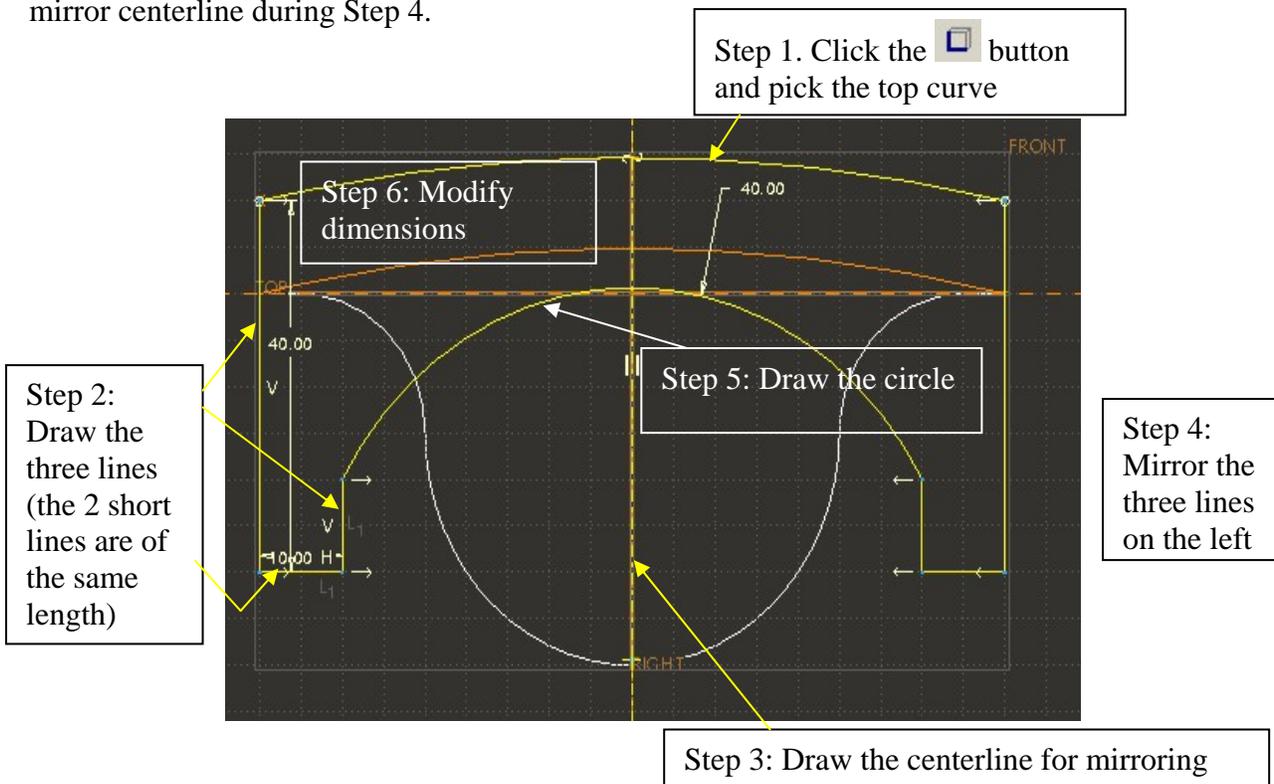


Figure 18 Steps for creating the caliper bracket sketch.

The steps in creating the sketch of the caliper bracket are shown in Figure 18. The extrusion depth is [10].

The model so far should look like Figure 19.

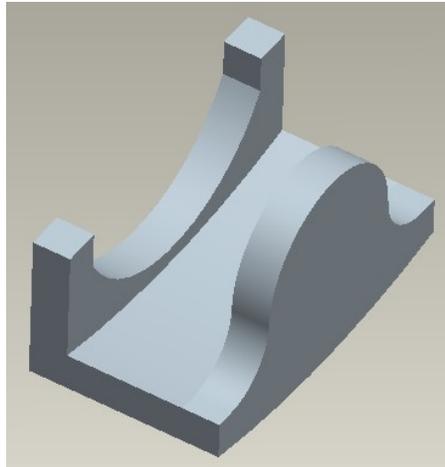


Figure 19 The housing top with the two brackets.

3.4 Create a cylinder

The cylinder is another extrusion created from the surface of the cylinder bracket extrusion. Too many extrusions, right? It is true that the extrusion tool is probably the most popular command. Get bored with my explanation on creating an extrusion? Fine, I will leave this to you to figure out (The diameter is 45 and depth is 25.)

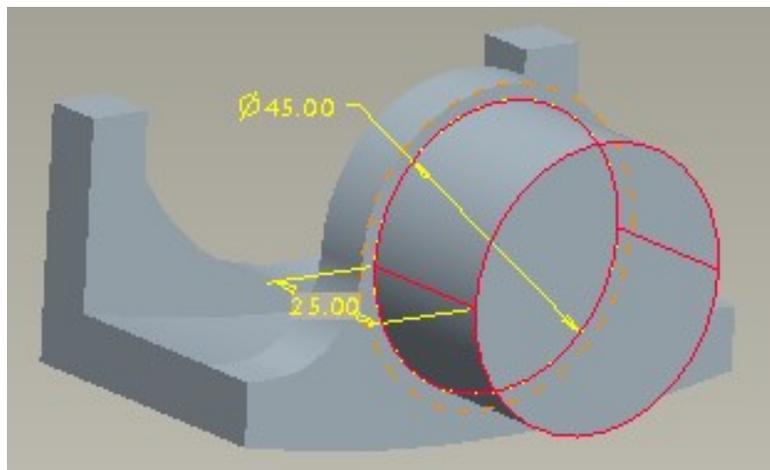


Figure 20 Dimensions of the cylinder.

HINT: Use the “Create concentric circle button” and pick the big circle on the cylinder bracket

to create the sketch section.

3.5 Create the hole

Now, you will do something different and have some fun. Click the  button. Then pick the axis of the cylinder that you have recently created (of course, you have to turn on the “Datum Axis On/Off” button). You should now see a hole in pale yellow. Wait, you need another reference to fully constraint the hole. Click the **Placement** button on your bottom left tool bar. Click the **Secondary References** blank, and then pick the starting surface of the hole, which is the other side of the cylinder bracket. Double click the dimensions, enter [30] for the diameter and [25] for the depth. You are done! Please refer to Figure 14 to see the hole.

This is in fact the so-called featured-based modeling. Fancy name, eh? It simply means that Pro/E allows you drag and play some simple features such as holes and chamfers to the model without getting into the datum planes → sketch → defining cycle, as in the extrusion definition.

3.6 Create the two chamfers

Since we are in the feature-based modeling mood, let’s finish the chamfers and rounds before modeling the last two slides. Referring to Figure 14, we are to create the two chamfers on the caliper bracket.

Click the Chamfer Tool button , a dialog window will appear at the left bottom window.



Figure 21 Dialog window for chamfering.

Pick the line on the caliper bracket to be chamfered. Choose the options and enter data as shown in

Figure 21. Note that the dimensions D1 and D2 might be interpreted differently by Pro/E than what you want. In this case, you’d change the value of D1 to [10] and the value of D2 to [3].

Repeat the same steps for the chamfer on the other side of the caliper bracket.

3.7 Create the rounds

There are in total 8 rounds to be created, namely, the four sides of the top surface of the housing, the intersection curves formed by the two brackets with the housing feature, the outer edge of the cylinder, and the intersection between the cylinder and the cylinder bracket.

Click the Round Tool button , enter the round radius [2] in the dialog window at the left bottom window. Then pick the eight curves. These rounds should be created accordingly. Refer to Figure 14 for illustration.

3.8 Create the two slides

Referring to Figure 14, the two slides, located at the two short sides of the “housing top” feature, are for assembling, which will be discussed later in the Assembling section.

Use the FRONT plane as the sketch plan. You will see that the FRONT datum plane is exactly in the middle. What a coincidence! (is it really a coincidence?) The section, an equilateral triangle, is shown below in Figure 22. The extrusion depth is [18].

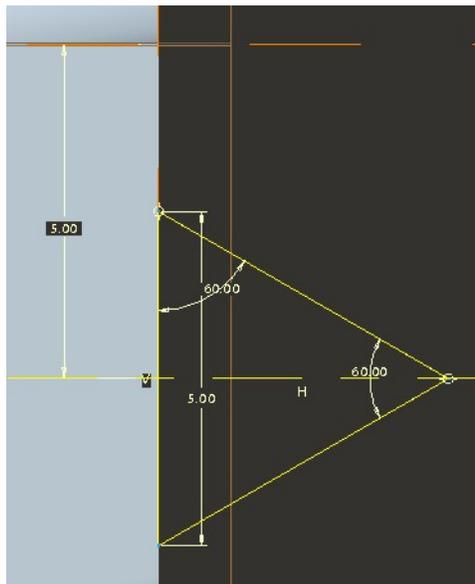


Figure 22 The sketch section of the slider.

Once the slide on one side is done. Use **EDIT > Feature Operations**. In the FEAT window, choose **Copy > Mirror / Select / Dependent / Done**. Pick the slider for mirroring. Then pick the RIGHT datum plane. The slider should be mirrored to the other side.

Congratulations!!! You’ve just finished your first complete part. Remember one thing: save your work.

3.9 Clean your directory

One more thing before you complete this section. Every time you save your work, Pro/E creates a separate file, be it a part, drawing etc. That is to say, if you saved your *housing.prt* 10 times during the modeling process, Pro/E should have created 10 files for you namely, *housing.prt.1*, *housing.prt.2*, ... *housing.prt.10*. This does have its advantage in version control. But you will find the files soon piling up. Therefore, at the end of your working session, you'd delete old versions of your files by clicking **FILE > Delete > Old Versions**.

In summary, you have learnt how to:

- ❑ Build features on existing features
- ❑ Use more advanced sketching skills such as mirror, constraints, use prev, etc.
- ❑ Use feature-based modeling tools such as hole, chamfer, and round
- ❑ Mirror a feature

Now, you should start from scratch, put aside the tutorial, and challenge yourself to see if you can build the part on your own. If you can do that, you are almost an expert on part modeling and you are ready to build the other components of the disc brake assembly (ask your instructor/TA/tutor about the other components). In the next section we will discuss how to generate a detailed engineering drawing for the housing part.

4 Creating a 2-D Engineering Drawing

In this section, we will turn the 3-D solid model of the component *housing* into a conventional 2-D engineering drawing.

Choose **FILE > New**, then select the radio button next to **Drawing** in the window. Enter the name [housing]. Uncheck the **Use default template** button.

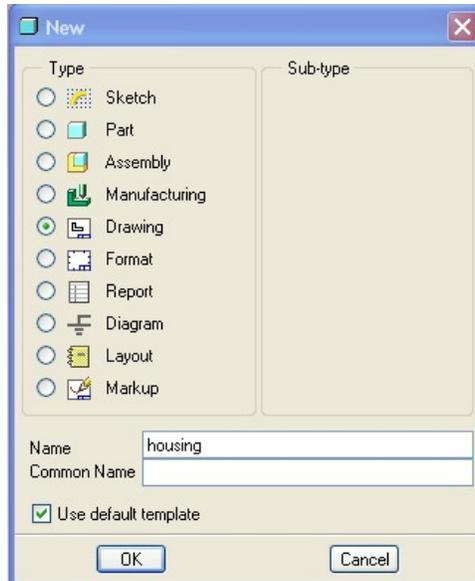


Figure 23 Creating a new drawing window.

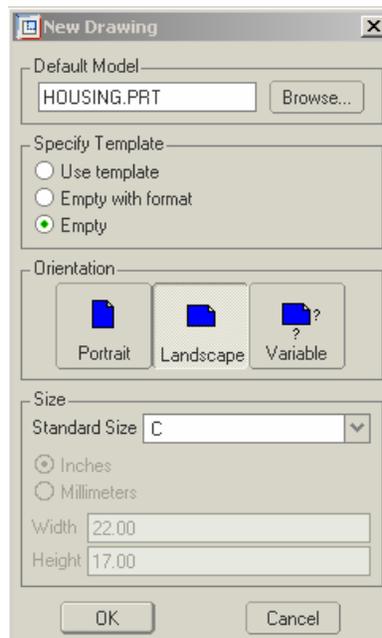


Figure 24 Drawing set-up window.

A dialog window will pop-up, shown in Figure 24. Pro/E automatically brings up the part model, as long as the filename is the same. The drawing file suffix is “.drw”, a part file suffix is “.prt”, and an assembly file suffix is “.asm”. Accept all the default settings in this window. Then you will face a black box for drawing. The size setting default should probably be changed to either A4 or A3 depending on the drawing requirements.

4.1 Insert views

Click the “Insert a drawing view”  button (or use the menu **Insert > Drawing View > General**). You will see in the message window “Select CENTER POINT for drawing view.” Click in the main graphics window to locate your first view (at the bottom left quadrant of the box). You will then see a dialog window as shown below:

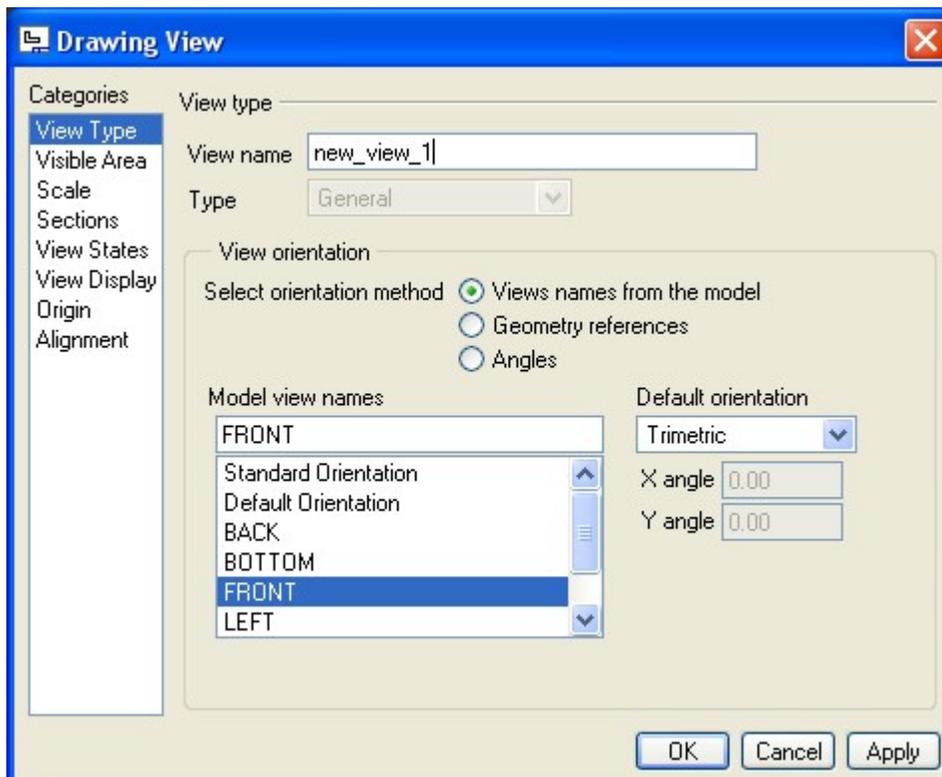


Figure 25 Dialog window for view control in the drawing mode.

In the dialog window, select **FRONT** as shown in Figure 25. Then click **OK**. You can double click the view to change the scale. You will then see the dialog window again. Select **Scale** in the left column, and enter the value [0.065] in the **Custom scale** blank. You should see a defined front view of the housing.

Use the menu **Insert > Drawing View > Projection**. Then click at a location right above the first view. You will see the top view is generated. Repeat the last step to create the right view of the model. (*Hint: This time you need to click the front view first to specify from which view the projection is created.*)

You will see now your views are pretty messy with many lines and datum features. You could press all the datum view buttons and then the **Redraw** button to clean the drawing a little bit.

Then Click TOOLS > Environment. In the last blank of the pop-up window, choose **No Display** for **Tangent Edges**. After performing a **Redraw**, all the tangent edges for rounds are cleaned up. The views look much better.

Last, we need to add an isometric view. This is done by clicking the  again. Click the upper right quadrant for location. Since the default view of the model hides a lot of the features, the model has to be re-oriented for a better view. Please refer to **Error! Reference source not found.** to select **Angles** from the view orientation section. In the **Rotation Reference** blank, pick **Horizontal**, and enter [180] degree in the **Angle value** blank.

Click **Apply** in the Orientation window, you should be able to see the isometric view. Change the scale to [0.065] in the same way as you did before on the front view. Then press the **OK** button.

Deselect the  button, you can select a view, press hold, and drag it to a desired location (using the LMB of course and following the exact action sequence please). You will notice that you cannot drag the views freely as they are inter-connected to satisfy their interrelationships.

The drawing at the current stage should look like that in Figure 26.

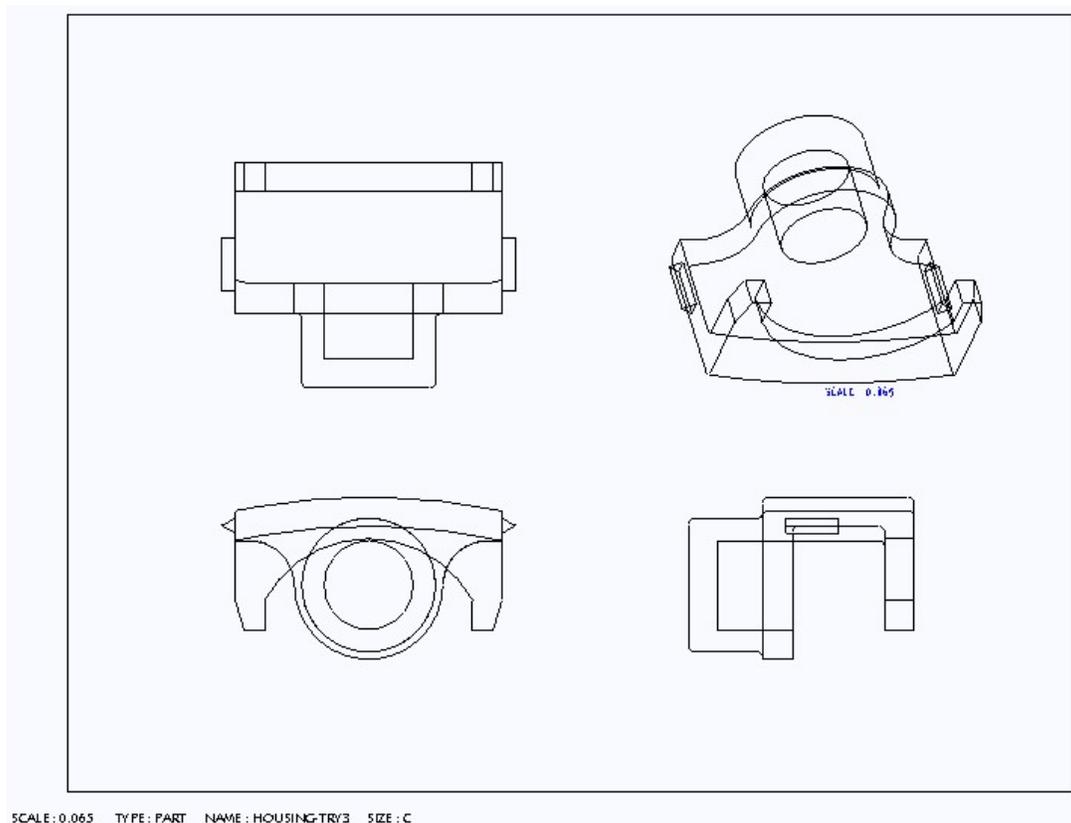


Figure 26 The drawing of Housing after the Inserting Views step.

4.2 Add dimensions

Click the  button. You'll see a window as shown in Figure 27. Pick the options as shown in the figure, and click any view in the main graphics window to select the part. You will see many dimensions shown on the drawing.

Click the dimension cleaning up button . Click and hold the LMB in the main graphics window to draw a virtual box to select all the dimensions shown on your drawing (finish the box by another LMB click). In the pop-up window, accept all the default settings and values about the spacing between dimensions (By the way, they are standards). Press **Apply** and **Close** button to close the window.

Now, as an engineer, you might find some of the default dimensions (which are created at the part modeling stage) are not appropriate. You might want to erase some of the dimensions and add some new dimensions. To do that, you have to use the  button and the Add new dimension button  for each detailed feature and dimension. You can also click each dimension using the LMB, then press the RMB, you will see a menu shown in Figure 28 which allows you move one dimension to another view (just choose the dimension and click the view of

destination), flip arrows, move the dimension text, etc. You will find it is very useful. Nevertheless, this dimension clean-up process is a little boring, but what can you do?

HINT: You should be careful when using the  button after you have cleaned up the dimensions. You might accidentally bring up all the erased dimensions up again to the views. The bad news is that you would not be able to Undo it. That means that you have to re-erase these dimensions one-by-one.

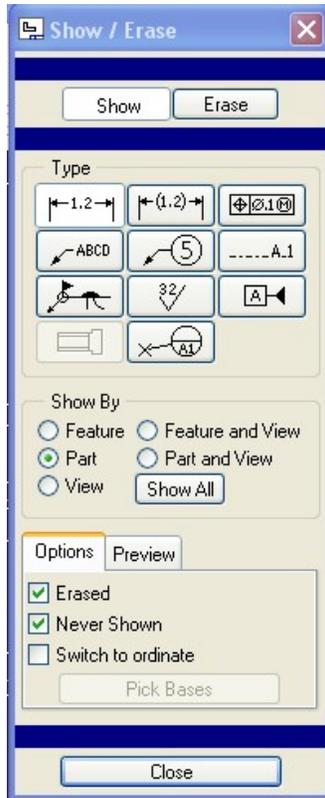


Figure 27 The show/erase dimension window.

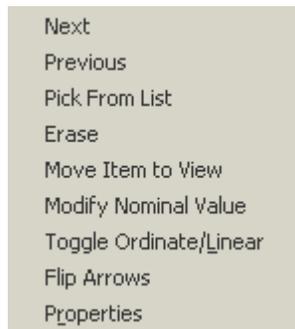


Figure 28 Right mouse button menu allows many detailed actions.

HINT: You might notice that the dimensions that you add to the drawing can be deleted but the dimensions shown automatically at the beginning can only be erased. Why? Because the

dimensions automatically shown are the ones you used to create the part model, which are called driving dimensions. Driving dimensions can be modified to change the part model, but cannot be deleted unless you redefine the part model. The dimensions that you add to the drawing are for the ease of understanding and are not driving dimensions. Thus they can be deleted.

4.3 Other Useful Features

Insert a Note

For all the small rounds, you can erase all the dimensions. Instead, you enter a note to the drawing. Choose **INSERT > Note**, select **No Leader / Enter / Horizontal / Standard / Default / Make Note**. After picking a point where to locate the notes, in the prompt window, enter [Small rounds are R2.00.] and press the enter key again to finish. Then click **Done/Return**.

Show Centerlines

If centerlines of circular features are not shown, you can press the  button, pick the **Axis** button to show and pick the three orthogonal views. If you see labels of the centerlines, deselect the **Datum axis on/off** button in the top tool chest.

Erase Snap Lines

The snap lines (dotted lines generated automatically when you show dimensions) are shown on the drawing. They will not be printed out when you create hard copies. However, if they bother you, click **TOOLS > Environment**, deselect the **Snap Lines** button in the **Display** window. Then click **Repaint** button. Those snap lines will disappear.

Modify Display

You could double click a view. You will see a **VIEW MODIFY** window. Click **View Disp**, then choose **Hidden Line / No Qlt HLR / No Disp Tan / Drawing Color / Done**. You can change the display of a view to the **Hidden Line** format, no matter what is the setting in the top toolbar.

HINT: The default view display (hidden line, wireframe, etc.) of the drawing is determined by the view display of the part in the model window. Once you used the view modify window to change a view's display as illustrated above, these display settings will become fixed and will not be affected by the top toolbar buttons.

HINT: Press and hold <Ctrl> key and you can click multiple views. So you can change their display settings all at once.

Change the Drawing Configuration

Pro/E defines many configurations such as arrow width, arrow length, etc. By changing those configurations, you can have more freedom in creating your drawing.

Now right click and hold RMB in the open space of the main graphics window (not one of the views). Select **Properties**, then **Drawing Options**. You will see a list of options. Choose **Sort > Alphabetical**, find the following parameters and change their settings to the values shown in Table 2.

Table 2 New values for the selected parameters.

Parameters	Values
drawing_text_height	0.1
draw_arrow_style	FILLED
draw_arrow_width	0.06
draw_arrow_length	0.16
tol_display	YES

After the setting change, you will see the arrows and texts are changed.

Display Tolerance

Just for exercise, you can now click the inner circle of the cylinder. Go to its **Properties**. Choose the options as shown in Figure 29.

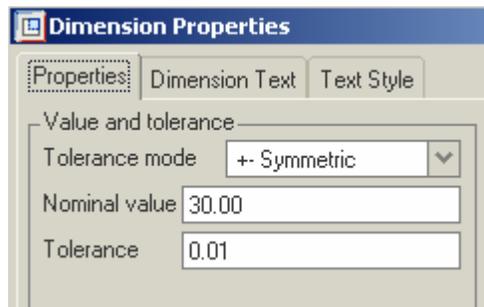


Figure 29 The dialog window for tolerance setting.

You should see the tolerance of the dimension showing up.

The final drawing looks like the one shown in Figure 30. Please note the scale has been modified. The display of dimensions are also modified a bit to allow zero decimal points except the diameter for the hole. You should be able to do all these now, right?

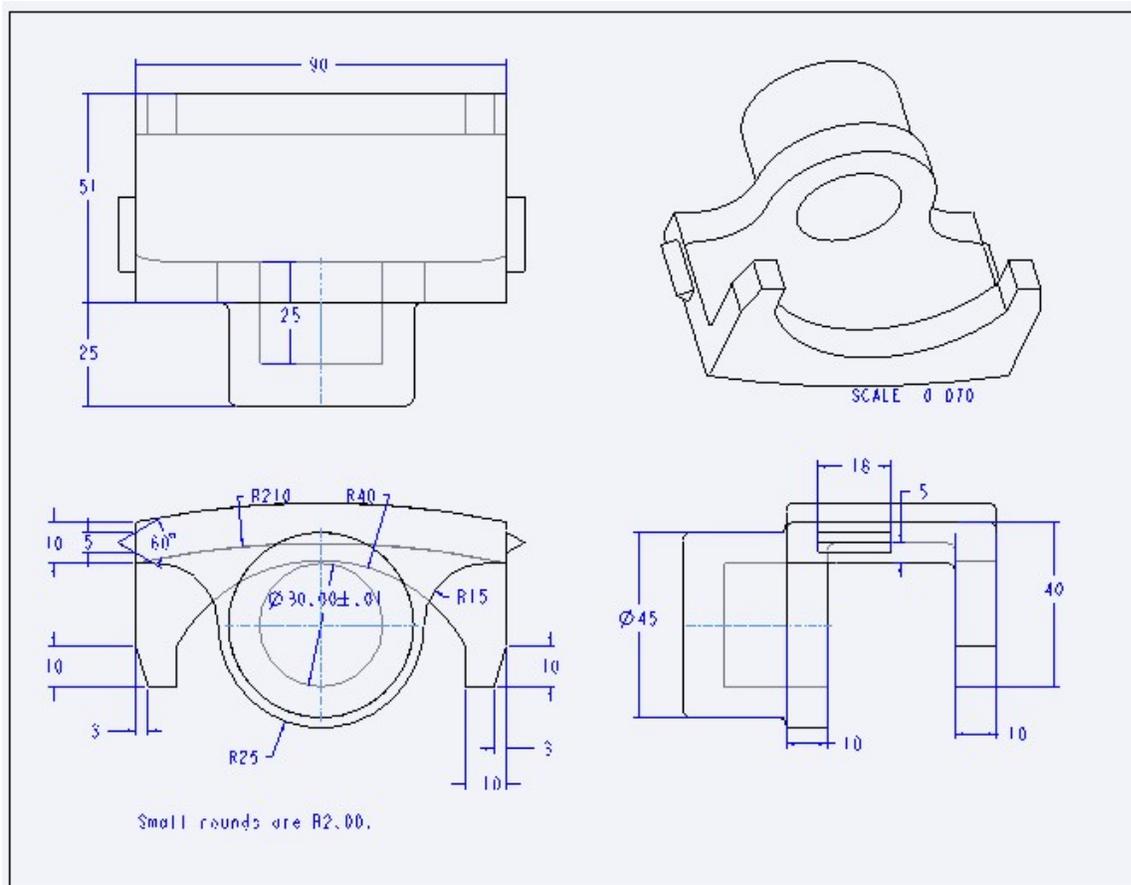


Figure 30 The drawing of the component Housing.

In summary, in this section, you should have learnt about:

- ❑ Inserting different views to the drawing
- ❑ Showing dimensions and adding new dimensions
- ❑ Cleaning-up dimensions by moving its position, switching views, erasing, etc.
- ❑ Inserting a note
- ❑ Modifying the model display
- ❑ Modifying drawing configuration
- ❑ Showing tolerances
- ❑ Showing centerlines

Now, put aside the tutorial and try to generate a drawing for *Housing* all from scratch. Repeat until you know how to do it all by yourself.

5 Creating the Disk-Brake Assembly

Creating an assembly is a fun task. Your main challenge will be display management as the screen gets messy with many features shown. However, you will not appreciate that until you get into your project.

To actually assemble components, we specify assembly constraints. As we know that the geometric relationship between any two parts has six degrees of freedom (DOF). To assemble two components is equivalent to constrain all 6 DOF's between the two. There are six types of common constraints that you should know. The rest should be easy to figure out on your own. Remember that the constraints must be used in combination in order to fully constrain the 6 DOF's.

5.1 Six Common Assembly Constraints

MATE (or MATE COINCIDENT)

Two planar surfaces or datums become coplanar and face in opposite directions.

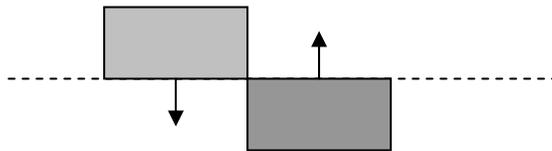


Figure 31 The MATE constraint [1].

MATE OFFSET

Two planar surfaces or datums are made parallel, with a specified offset distance, and face in opposite directions. The offset dimension can be negative.

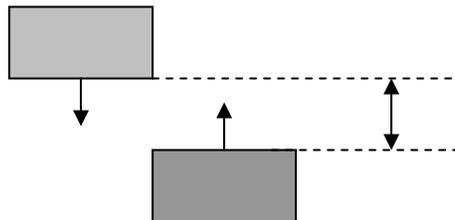


Figure 32 The MATE OFFSET constraint [1].

ALIGN (or ALIGN COINCIDENT)

This can be applied to planar surfaces, datums, revolved surfaces and axes. Planar surfaces become coplanar and face in the same direction.

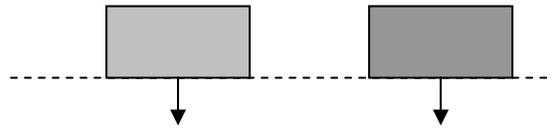


Figure 33 The ALIGN constraint with planar surfaces [1].

ALIGN OFFSET

This can be done only with planar surfaces: they become parallel with a specified offset and face the same direction.

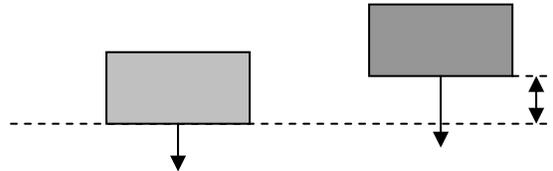


Figure 34 The ALIGN OFFSET constraint [1].

ALIGN OREINT

Two planar surfaces or datums are made parallel and face the same direction (similar to Align Offset except without the specified offset distance).

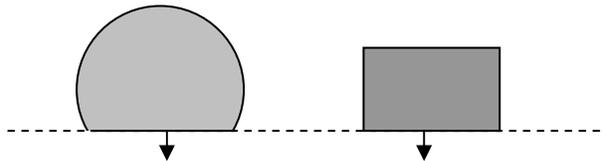


Figure 35 The ALIGN ORIENT constraint.

INSERT

This constraint can only be used with two surfaces of revolution in order to make them coaxial.

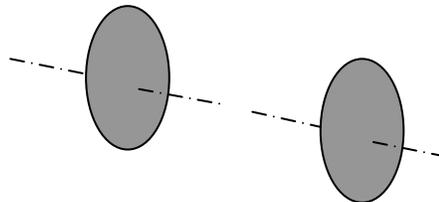


Figure 36 The INSERT constraint.

As you might already notice, for certain geometries and constraints, you could have more than one choices from the 6 basic types. For example, ALIGN can be used to make two axes coaxial, equivalent to INSERT, and so on.

Having understood the basic assembly constraints. Let's assemble the disc brake.

5.2 Build the disc-brake assembly

Use **FILE > New**, or click the  button to launch an **Assembly** application. Name it [DiscBrake], and uncheck the **Use default template** button. In the **New File Options** dialog window, choose **Empty**.

You should see an empty main graphics window with a few active buttons (comparatively). Click the **Add Component** button  to place the first component, which is the *Housing* part we created before.

Assemble the disc pad on the caliper side

Click the  button again to assemble. Choose the *disc_pad.prt* from the directory (if you cannot find them, that means that you have not created them yet. If you prefer, you could ask your TA/tutor for those components.)

*HINT: If you could not see the disc pad part on the screen, oops, you probably used a different unit system for the two parts. Open each part. For each part and the assembly to be created, choose **EDIT > Setup**. In the menu window, click **Units**. Make sure all the parts have the same unit system. If not, you can set them to be the same. When you do that, there are two options. You can either maintain the actual size so that all the dimensions will be translated to new numbers (the first radio button), or you can maintain the dimension numbers so that the size will either shrink or enlarge (the second radio button).*

The first two buttons at the top of the window shown in Figure 37 allow you either put the two components in the same window or in two separate windows.

HINT: If you want to assembly components as a mechanism, you'd have to assemble them within the same window. This seems unreasonable but...we have to live with it.

Continuing to refer to Figure 37, in the Constraints section, choose **Mate**. Then pick the inner side surface of the caliper bracket and the outer surface of the disc pad. Then pick **Align**, pick the small bottom surface of the caliper bracket and the bottom surface of the disc pad. The default Offset blank is **Coincident**. Click it and choose **0.0**. Change it to number [2]. You should now see the dialog window as shown in Figure 37. Down in the Placement Status section, the message says “**Partially Constrained**”. It indicates more constraints are to be added. The **Reference** section tells the user what are the features being picked for constraints. When you click any constraint in the **Constraints** section, the features being picked will be highlighted in the main graphics window. You can also add or delete a constraint by using the **Plus** or **Minus** sign button in the middle of the window. The window shown in Figure 37 is the one that you have to use again and again for assembling each component.

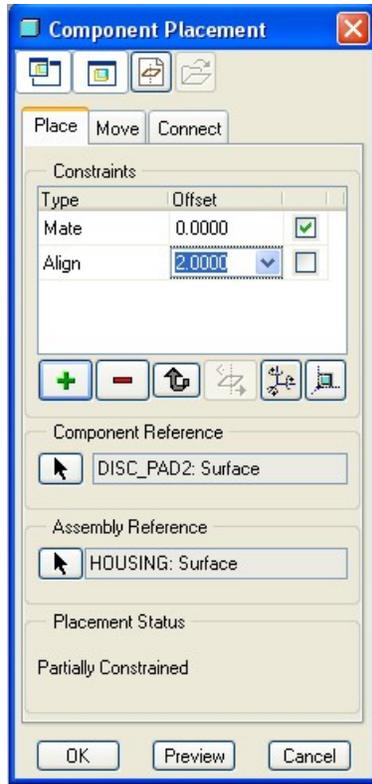


Figure 37 The main dialog window for assembly.

Please refer to Figure 42 for the third constraint to complete assembling this component.

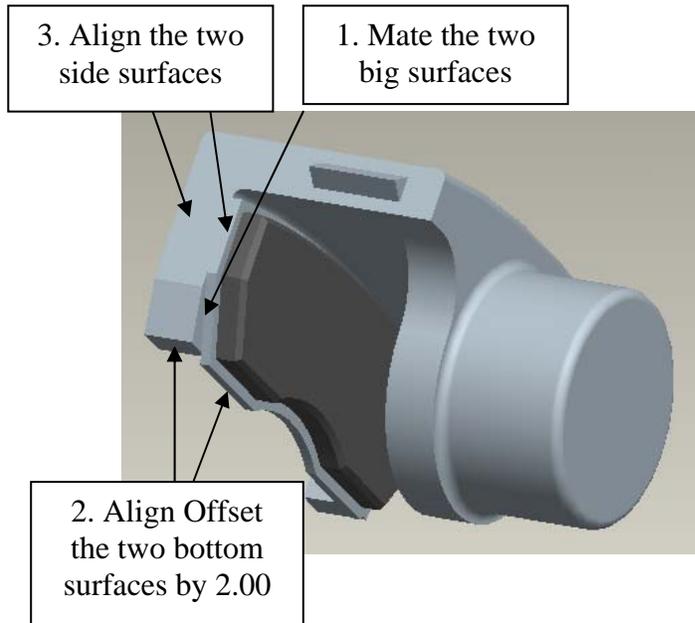


Figure 38 An illustration of assembling the first disc pad.

Having understood the first one, the rest assembling becomes easy. So the tutorial will only give you some guidelines and leave the details to you. Are you ready?

Assemble the piston

Click  and select *piston.prt*.

1. Use the **Insert** constraint and pick the outer surface of the piston and the inner surface of the hole in the part *housing*.
2. Use **Align**, pick the top surface (the open end) of the piston and the inner surface of the cylinder bracket of *housing*. Key in the offset number [2.0].

Assemble the pad housing

Click  and select *pad_housing.prt*. The assembly sequence is illustrated by Figure 39 (refer to Figure 40).

HINT: The sequence of constraints does matter. The tip is to bring the two components in the same window. After one constraint is specified, the component will move its relative position. A good sequence of constraints should bring the component closer (conceptually, not necessarily physically) to its final position after a constraint is added.

HINT: Turn on the datum plane display to view and select the datum planes.

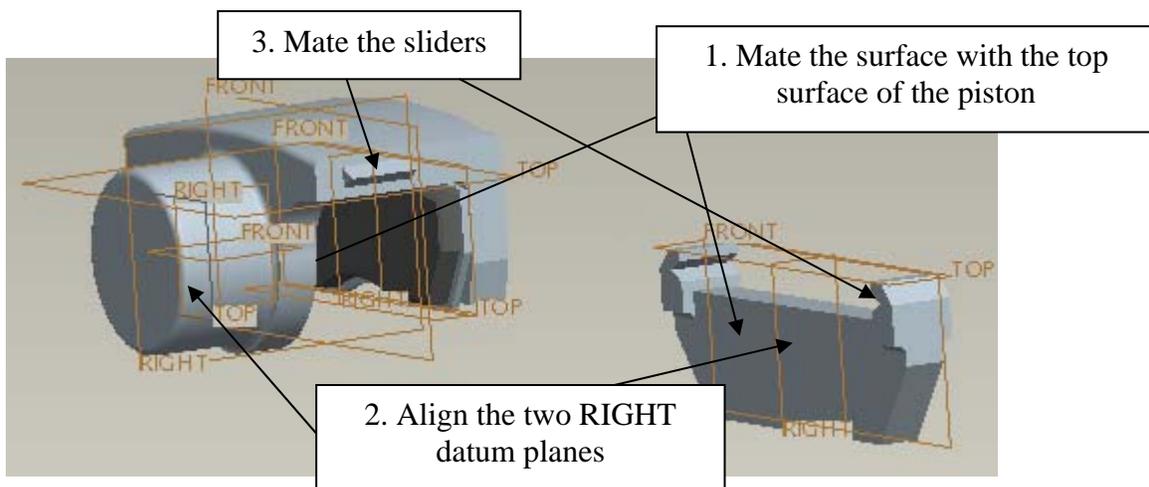


Figure 39 The procedure for assembling the pad housing.

Assemble the other disc pad

Procedure:

- 1) Mate the outer surface of the disc pad with the corresponding surface of the pad housing.
- 2) Align the RIGHT datum plane of the disc pad with that of the housing.
- 3) Align the bottom surface of the disc pad with that of the existing disc pad.

The final assembly show look like the following:

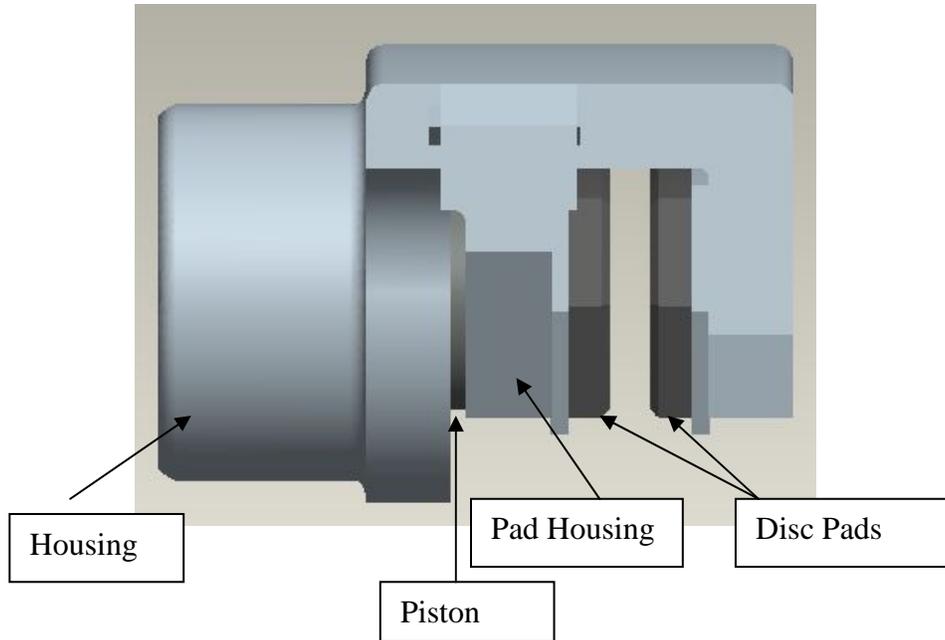


Figure 40 The disc-brake assembly.

5.3 Add Color and Create an Exploded View

Alright, since you have done the assembly, one common thing that engineers like to do is to add colors to different parts, though they are often not good at colors. You may see mine in the electronic version. Hopefully, there is at least one person who finds it pleasant.

Click **VIEW > Color and Appearance**. The rest should be straightforward. I will leave those to you.

An exploded view is useful when you create an assembly drawing. To create an exploded view, click **VIEW > Explode > Explode View**. The default exploded view will show, which often does not make sense. Then you should use **VIEW > Explode > Edit Position** to adjust the relative positions of the components. In the dialog window, you are asked to pick the motion reference, which is like a guide for your moving. Play with it until you get the position you like.

Figure 41 shows my exploded view and the colors assigned to the components.

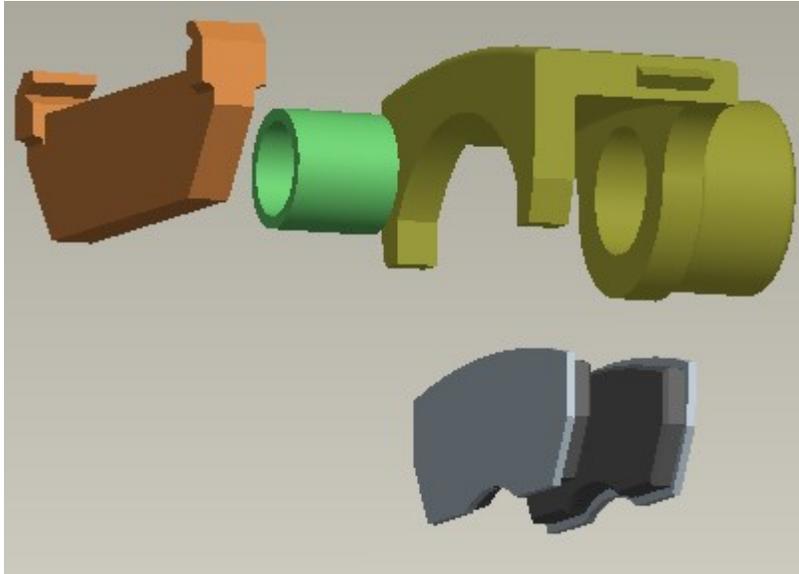


Figure 41 The exploded view of the disc brake assembly with colored components.

5.4 Create a Cutout View

You might notice that there are some modeling tools such as extrusion in the right tool bar. These tools allow you create assembly features on the spot. One example is that we can create a cutout view by creating a cut feature in the assembly.

HINT: To do that, you just need to create one Extrusion feature. The sketch section is just a line coinciding with one of the datum planes in the middle. The cut depth should be set to “Extrude to intersect with all surfaces”.

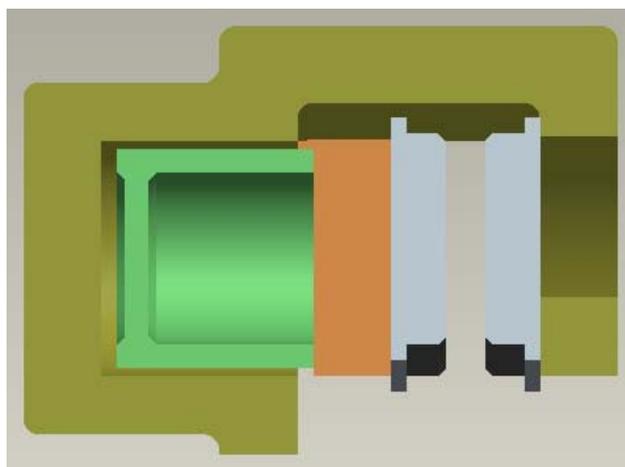


Figure 42 The cutout view of the disc brake assembly.

In summary, you have learnt:

- ❑ basic assembly constraints
- ❑ assembling components by assembly constraints
- ❑ adding colors to components
- ❑ creating an exploded view
- ❑ creating a cutout view

Again, put aside the tutorial, do it yourself!

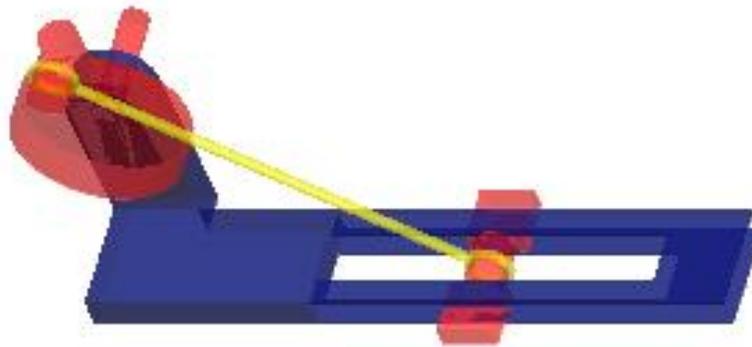
6 Animation in Pro/ENGINEER Wildfire 2.0

6.1 Background

Pro/E Animation is used to visualize the motion of a mechanism or a mechanical assembly in operation. **Pro/E Animation** is carried out by: (i) coordinating the components of an animation sequence, and (ii) playing back the animation.

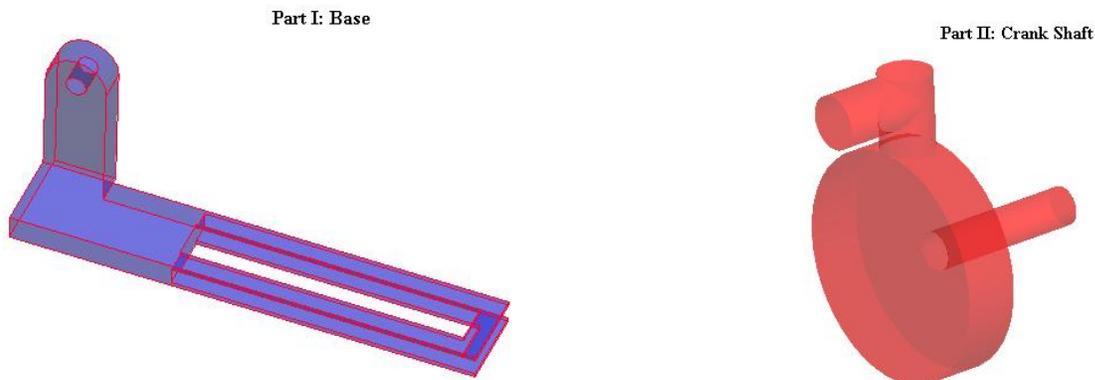
Different from ordinary Kinematics Motion function of Pro/E, the Design Animation function in the **Animation** module of Pro/E Wildfire 2.0 is more about showing the motion of a concept mechanism design. The motion can be illustrated by dragging the moving components, or by interpolating a number of positions of the mechanism. It may not honor all of the constraints defined in the design. One can carry out Animation before the detailed assembly is completed.

This tutorial demonstrates Pro/E Motion Design by creating the motion sequence of a simple four-bar linkage mechanism, as shown in the following figure.



6.2 Creating Assembly

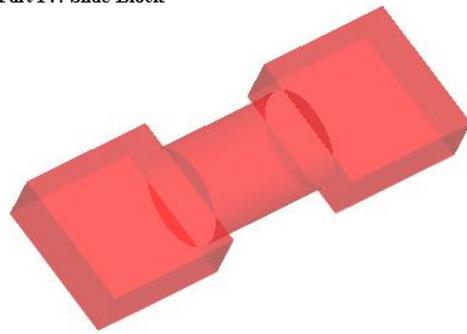
The mechanism to be animated consists of four components as shown in the following.



Part III: Link



Part IV: Slide Block



These components are modeled and their models are assembled in Pro/E. The assembly process could be a challenge. Uncheck the “Allow Assumption” in the Component Placement window tends to be helpful. The Pro/E model files of these components can be down loaded from the course website.

6.3 Creating the Motion Sequence

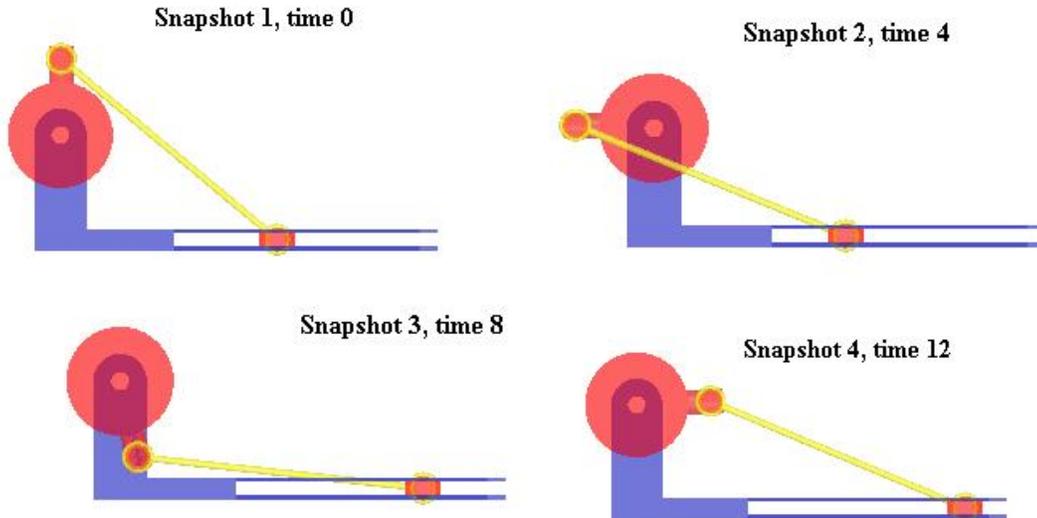
Following steps are needed to create an animation:

1. Create/open/bring up the assembly model.
2. Enter the Animation mode by clicking **Applications → Animation**.
3. The Animation toolbar (as left) and timeline appear on screen. You need to first get familiar with the meaning of the icons on the animation toolbar. An **Animation** menu is added to the Pro/ENGINEER menu bar. You can create your animation either by selecting commands from the **Animation** menu or by clicking the toolbar buttons.
4. You may practice on dragging one element of the mechanism to set the assembly moving. This can be done by clicking the “Drag model and create snap shot”  icon on the animation toolbar. In the Drag window, click on the “Drag model and create snap shot”  icon in the Drag window again, then selecting an object (left click on) and drag the part to move. Before the last step, you may also click the “snap shot”  icon in the Drag window first to record the new position after the dragged motion as a snap shot. This snap shot can be used later as a key frame of the animation.
5. Create a new animation by clicking **Animation → Animation** or click  on the Animation toolbar. The **Animation** dialog box opens with a default name for the animation. You may use the **Rename** command to reassign the name.
6. Check the body definition by selecting **One Part per Body**. This selection may empty the ground body of parts. You should edit the body named ground and reassign the base component as ground part.
7. Take a series of snapshots of the assembly at specific positions by using the drag function to move the moving bodies to several new positions. Animation will interpolate between these key frames to produce a smooth animation.
The series of repeated steps are:



Animation → Snapshot → Take Snapshot → Drag

For this example, drag the crank to four positions, i.e. 0° , 90° , 180° , 270° in the snapshot mode.



8. Define the animation motion sequence of a component (driver) to create the movement, such as **Key Frame Sequences**.

Click **Animation → Key Frame Sequence**; give a proper name, and make sure the reference is Ground. In the Key Frame box, sequentially select each of the snapshots and assigning corresponding time. For interpolation box, choose linear or smooth as you like. Then, Click ok and exit.

6.4 Playing the Motion

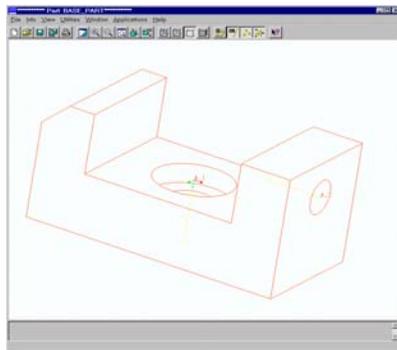
- 1) Start your animation by clicking **Application → Animation → Start the animation** or click on the  icon on the animation toolbar.
- 2) The **Run** dialog box opens. The model moves as specified by the animation components.
- 3) If you want to view the animation again, or to change the speed or direction, click **Animation → Playback** or click on the  icon on the animation toolbar.
- 4) Define views along your animation to view orientations and magnification of your model. You can also choose an interpolation method for your views.
- 5) Rerun the animation and view results.
- 6) Save your animation and results. You save your results by clicking  on the **Playbacks** dialog box. Animation saves your playback results to a .pba file. You can also export to a .fra file. Save your animation using the **File → Save** command. Animation saves your animation to the .asm file with your model.

7 Pro/Mechanica for Structural Analysis, Sensitivity Analysis, and Design Optimization

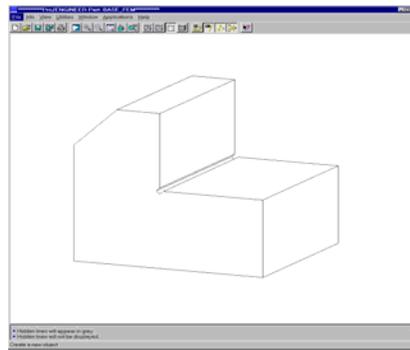
Pro/MECHANICA is a powerful finite element analysis (FEA) package developed for design engineers. There are three main functions provided by Pro/MECHANICA.

- structural, thermal, and motion analysis;
- design parameter sensitivity analysis; and
- design optimization.

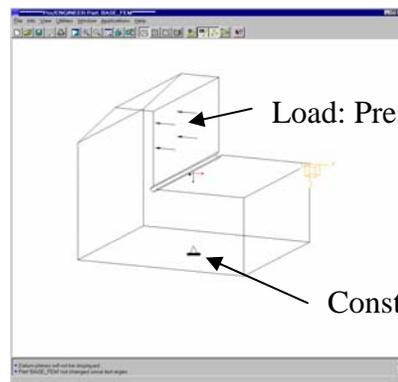
Pro/MECHANICA can work either as an independently FEA package, or as an integrated part of Pro/Engineering. This tutorial will outline its use in conjunction with Pro/E. The example part used in this tutorial is a fixture component for a milling machine. The part is tightened to the table of the mill through the center step hole using a bolt through the T slot. A block is pushed in horizontally to hold the part with the left arm of the dent (See Figure 43a). The left dent arm is then going to bear the holding force. The task of FEA is to calculate the maximum stress, verify the structural integrity, identify the weak area and design the part with a minimum volume (weight) while satisfying the strength and deformation requirements.



(a) Design Model



(b) Geometric Model for FEA



(c) FEA Model

Figure 43 Different models of the fixture component.

The FEA of the part is carried through the process as illustrated in Figure 44. A design model is usually simplified for the ease of analysis, which results in a simplified geometric model as

shown in Figure 43b. Based on the simplified model, material, loads, and constraints and others are defined for analysis. Figure 43c shows the model with loads and constraints.

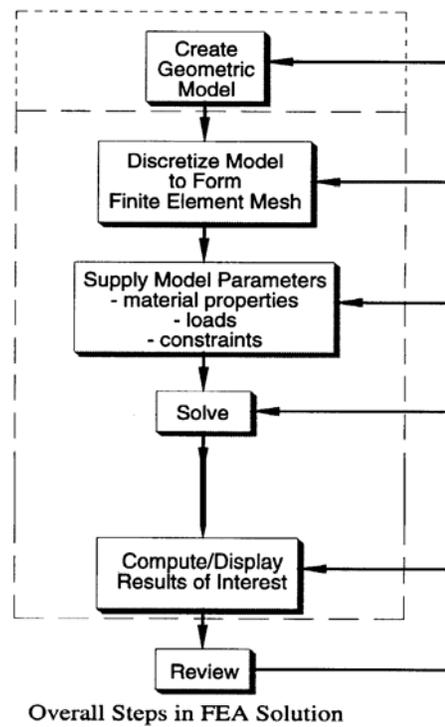


Figure 44 Finite Element Analysis process [4].

This tutorial will walk you through the structural analysis, parameter sensitivity analysis and the design optimization processes for the fixture component, named *base.prt*.

7.1 Prepare the Model

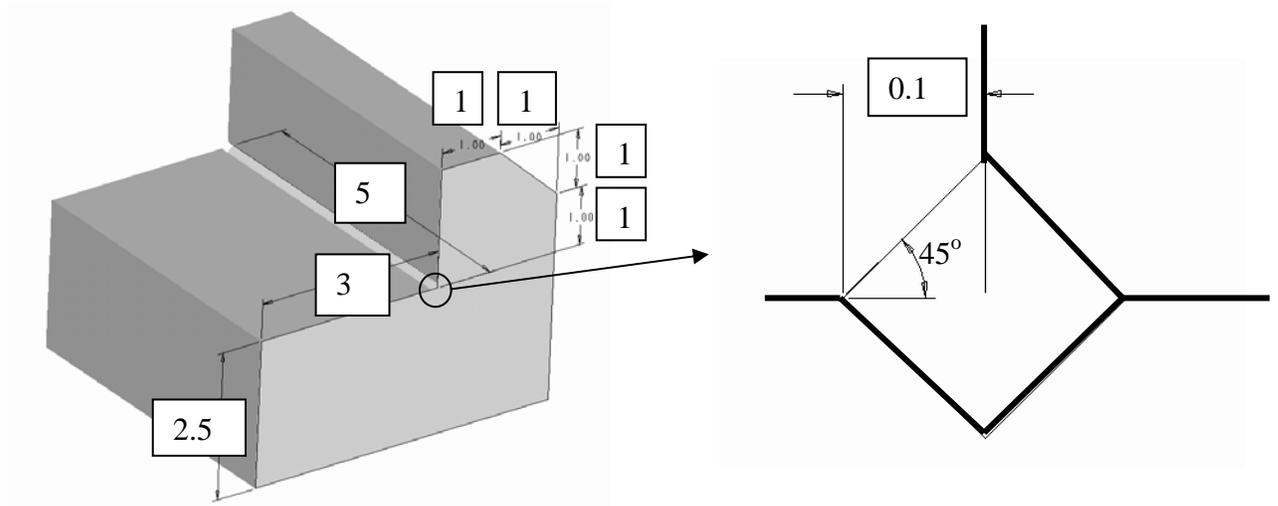


Figure 45 The simplified model of *base.prt*.

Figure 45 shows the simplified model with a scaled up illustration of the groove. This model can be created two simple extrusions within Pro/E. Please dimension the geometry using the exact scheme as shown in the figure. This is purely for the ease of carrying on later tutorials on parameter definition, sensitivity analysis and optimization.

HINT: For the groove cut, practice using geometric constraints  in the sketch model to define the square cut.

7.2 Start Pro/MECHANICA

Go to the Pro/E Pull-down Menu **Applications > Mechanica**. The pop-up window asks you about the unit system. Accept the default in-lbm-second unit system and continue. In the new pop-up window, choose “**structure**” and click **OK**. Now you will see the main menu for Pro/Mechanica with many icons. Scroll your mouse button on top of each one to learn their functions.

7.3 Define the FEA model

At least there are three basic elements to be specified to define a FEA model, i.e., material, loads and constraints.

Assign the material

Press the Define Materials button , you will see a window shows as below:

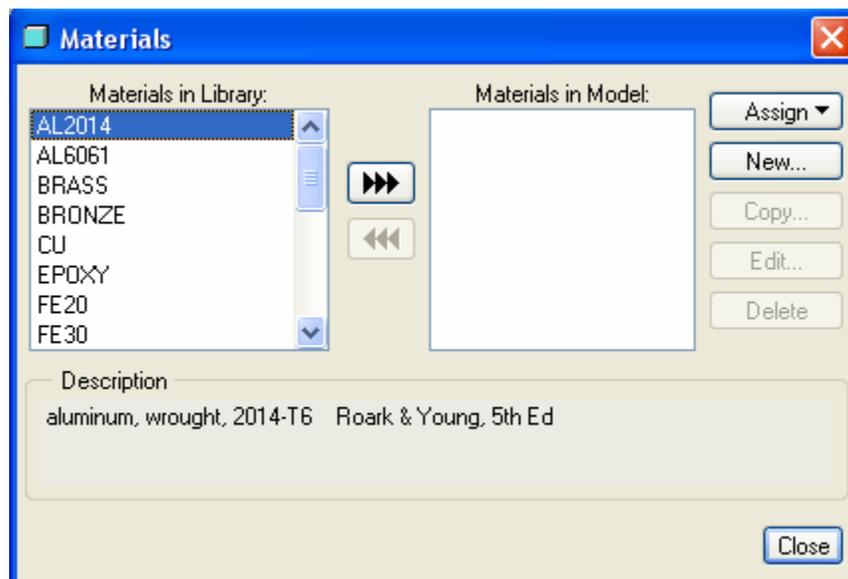


Figure 46 Material selection window.

This window lists the materials predefined in the library. One can scroll down the column and choose **STEEL**. Press the arrow and the **STEEL** will appear in the right column of the window. The **New** button allows you to define your own material. Or you can use the **Edit** button to edit

the material properties for the predefined materials. Press **Assign > Part**, then click the base part in the main window (press the middle button to accept the highlighted part). Then press **Close** to close the material selection window.

Define the loads and constraints

Press the New Pressure Load icon (where is it? If you cannot find it, scroll your mouse to read the description of each icon on the right hand side of the window). In the dialog window, accept all the default names. Choose the left inner surface as the load bearing surface and enter the value [5000]. You can use the Preview button to see the graphical indication of the pressure and select OK. *HINT: sometimes you may have to click the arrow below the word “Surface(s)” on the window to reselect your load bearing surface to proceed.*

Press the New Displacement Constraint icon. Following a similar process as in the load defining step, pick the bottom surface, and hit OK. You will see that all the six degree of freedom (DOF) of the surface are fixed.

The completed FEM model will look like below:

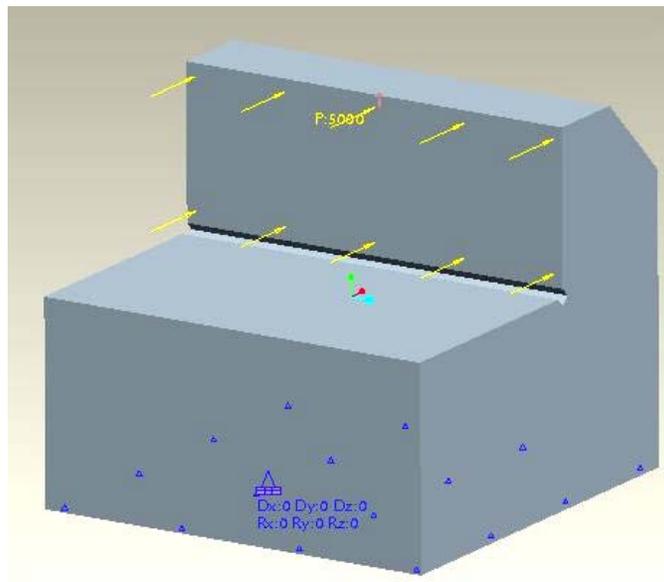


Figure 47 The defined FEA model with loads and constraints.

7.4 Run a static analysis

Now, you are almost ready to run your first static analysis. There are three important icons you should be very familiar with as shown in Figure 48.

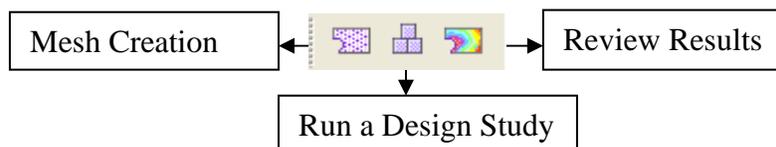


Figure 48 Important icons for performing FEA.

Pre-processing

As we know, the finite element method divides a finite element model to small elements. These small elements form a mesh of the model. The mesh generation is called pre-processing for finite element method. Different mesh generation scheme may result in considerably different analysis results. Pro/MECHANICA automatically generates finite element mesh. In advanced application of Pro/MECHANICA, one can specify important regions on the model, in which more detailed mesh can be generated.

Press the Mesh Creation icon shown in Figure 48, accept the default control parameters, you will see the elements on the model. These elements are rather coarse, because they are called P-element. What is the difference between a P-element and a N-element?

Perform the Analysis

Press the Run a Design Study icon, you will see a window shown in Figure 49.

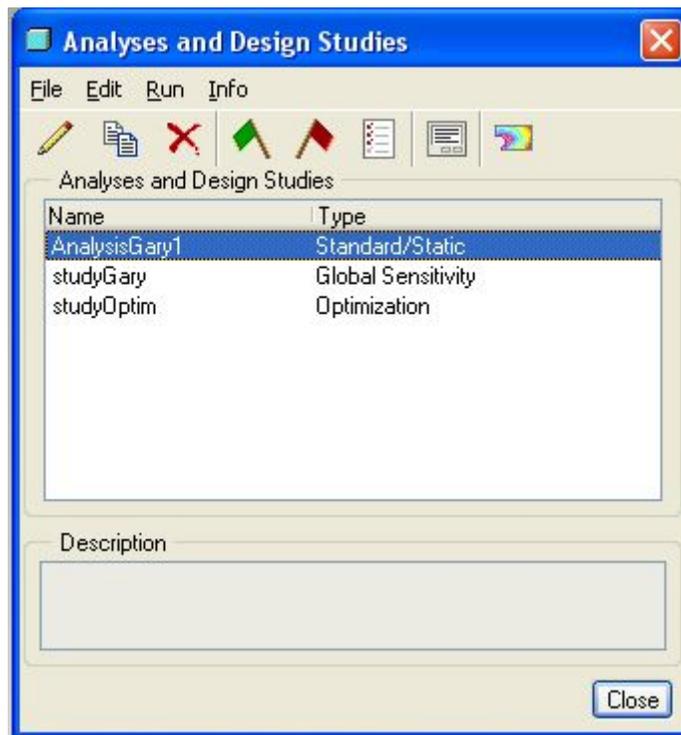


Figure 49 The design study definition and control window.

This window allows you define a simple analysis, a parameter sensitivity study, or an optimization study. We will refer to this window again and again in performing ensuing tasks. Please note there is an icon shown in the window identical to the Review Results icon in Figure 48. No surprise, they are of the same function as well, i.e., to view the FEA results after a analysis/study has been performed. Also to be noted here is that Pro/Mechanica calls a conventional mechanical analysis “analysis,” and the sensitivity analysis or optimization as “study.” Sometimes, it also uses “study” to refer to both “analysis” and “study”.

Choose **FILE > New Static**. In the pop-up window, you should see the load and constraint that you defined before. Since there is only one set for either load or constraint, you can leave all the default settings. To avoid override each other's analysis results, you may change the analysis name to something like "*analysisYourName*" instead of the default "analysis1".

Then you can hit the Start Run icon . Press the Display Study Status icon  to see the log file of the analysis.

If you successfully run the analysis and see the sentence "Run Completed", congratulations! It means that both your geometric and FEM have no error (It is modeling error only; it does not necessarily mean that your model prediction is accurate as compared to the real physical test.).

If you are careful enough, you will notice in the previous analysis definition window, there is a item called "method" with its default as Single-Pass Adaptive. Now, redefine the *analysisYourName* and change the method to Multi-Pass Adaptive. Run the analysis again. Hey, why do we have to do that? (You should have learned it in the classroom. If not yet, bear this in mind and ask it in the class.)

Post-processing

Press the Review Results icon from the Design study definition and control window. You will see a big blank window for displaying the analysis results. This step is called post-processing. You will have to tell Pro/Mechanica what to display by inserting display windows. Choose **Insert > Result Window**. In the pop-up window, press the icon below "Design Study". Then choose (or double click) the directory "*analysisYourName*." Accept the display quantity as "stress," component "von Mises." You should see the nice colored plot showing the maximum stress occurs in the groove. Repeat the same process to insert a new window. Choose the quantity as "displacement." Now you should see the results as in Figure 50.

You might have learned that the analysis method used in Pro/MECHANICA is called the p-element method. This method uses high order elements and gradually increases the order of elements based on the same mesh until the process converges. The best convergence criterion is the structure strain energy. The maximum von Mises stress of the structure is a value based on local information and thus may generate singular value ("freak" value). It might be fun to see the convergence history by using the two criteria to see if this is true.

Now, delete all the existing figures in the post-processing window. Repeat the inserting window process and choose the parameters shown in Figure 51. This window will display the convergence history for the maximum von Mises stress in the part. Following the similar process, but choose "strain_energy" instead of "max_stress_vm" to generate a new window. You should see the plots as shown in Figure 52.

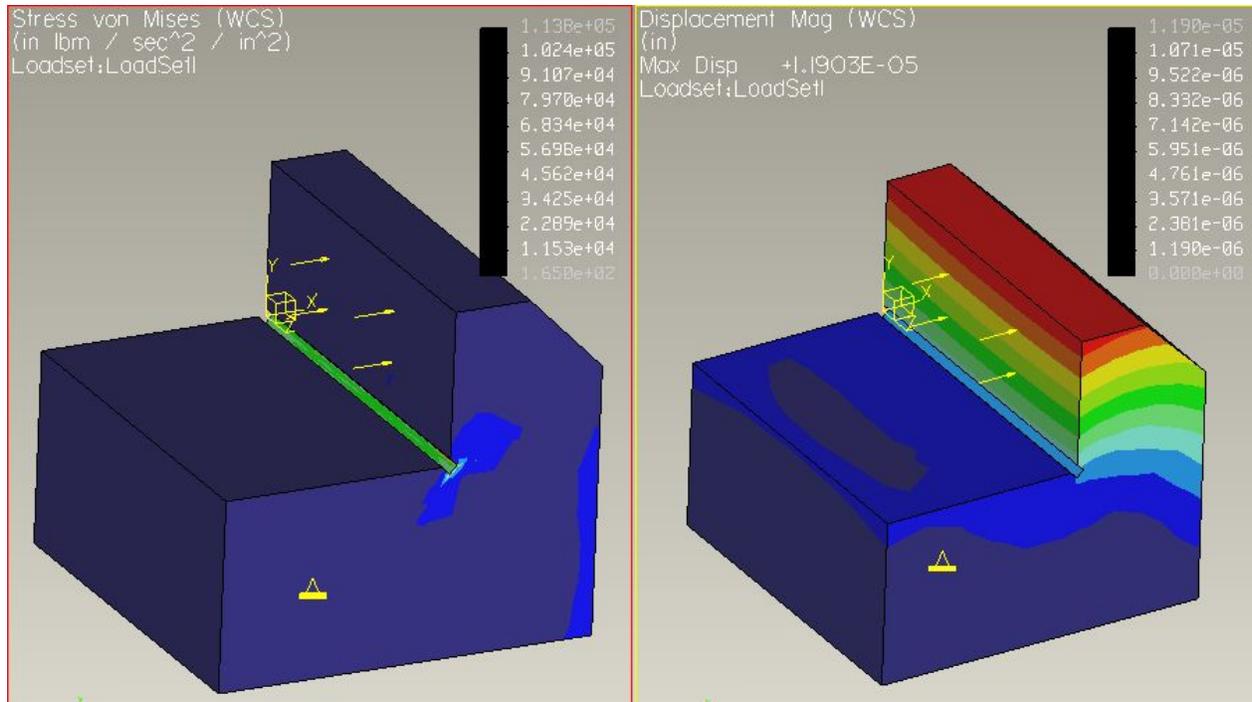


Figure 50 Stress and displacement analysis result.

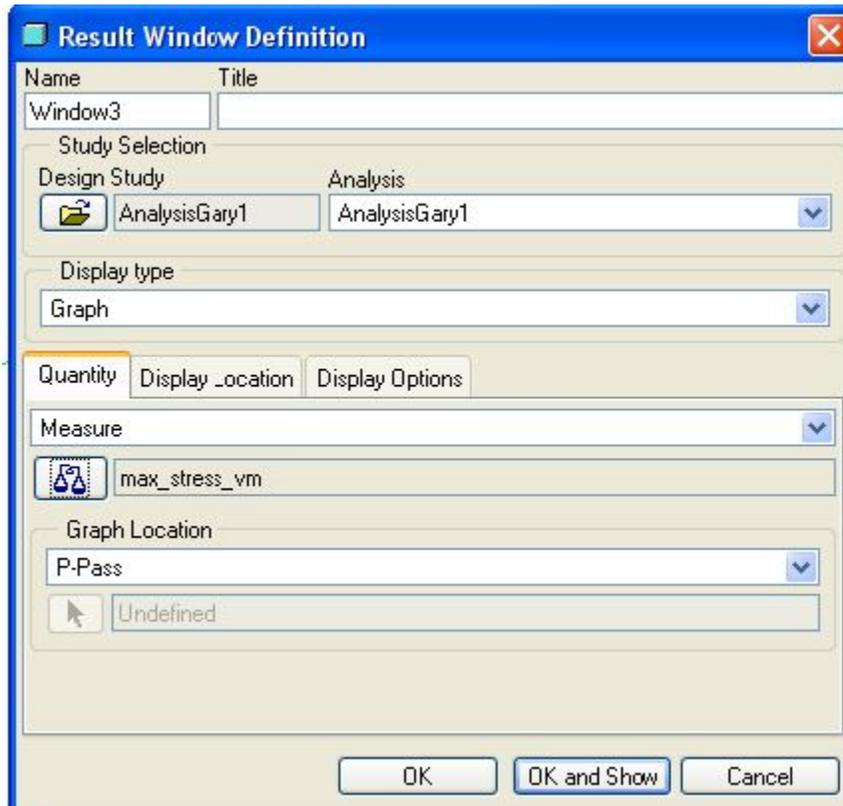


Figure 51 Parameters for generating a convergence plot.

The plots indicate that the strain energy well converges after six iterations while the maximum von Mises stress does not. (Note: you might get a different von Mises plot, that is fine as this stress does depend on your mesh which could be different from model to model. However your strain energy plot should be similar to the one shown below and should converge.)

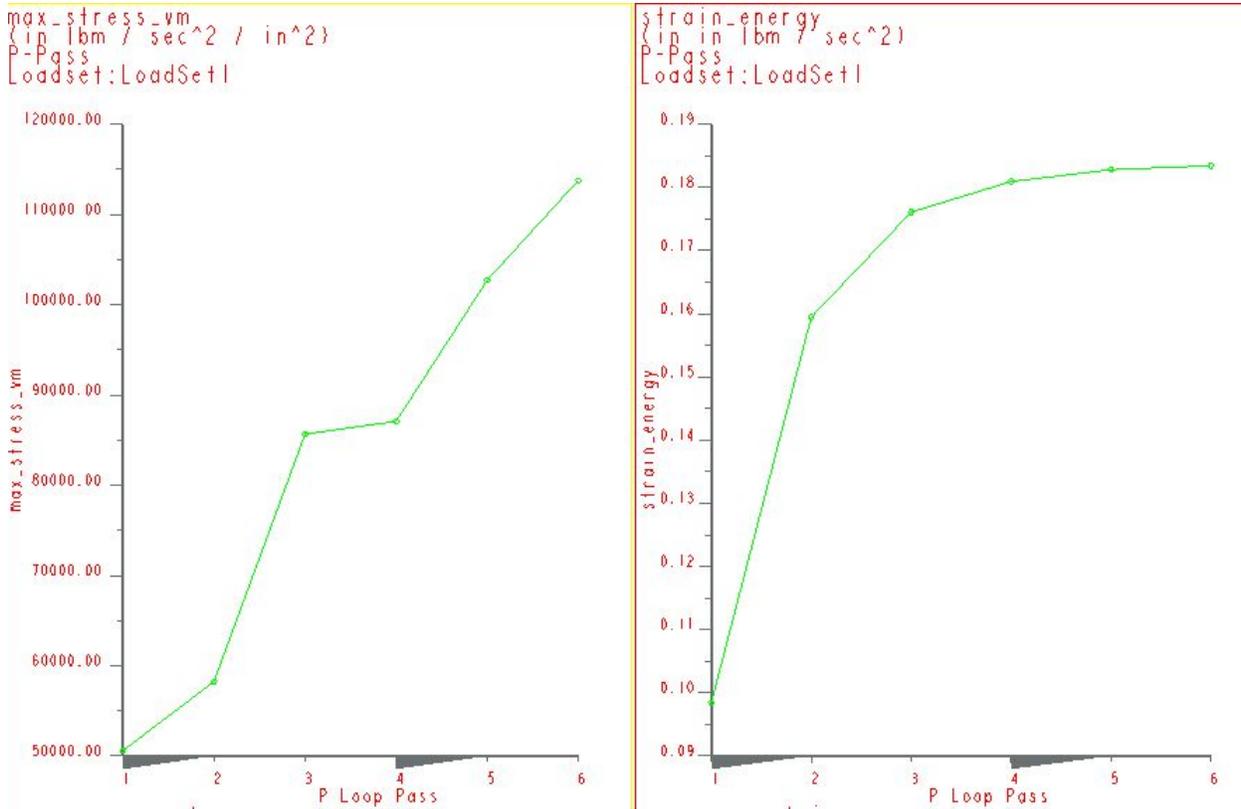


Figure 52 The convergence plots for the maximum von Mises stress and strain energy.

7.5 Design parameter sensitivity study

Pro/MECHANICA also can help a designer to study the parameter sensitivity to a certain performance criterion. In this example, we'd study the sensitivity of the groove size to the maximum von Mises stress and maximum displacement.

1) Define a design parameter

Choose **Analysis > Mechanics Design Controls > Design Params > Create**. In the dialog window, accept "dimension" as the parameter type, choose **Select**. Click the part, dimensions will show up on the screen, then pick the groove size dimension (**0.1** is the current value). The symbolic name of the dimension will then appear in the dialog window. Enter the minimum value as [**0.10**], maximum [**0.20**]. Then choose **Done**. Then you can use **Shape Review**, **Shape Animate** to check if the model remains feasible with a parameter varying in the given range. It is recommended for defining a new study. It is always a good idea to detect any potential problem at the beginning, isn't it?

2) Define a design study

Press the Run a Design Study icon shown in

Figure 48. Choose **File > New Design Study**. In the dialog window, enter the study name “*studyYourName*”. Choose “Global Sensitivity” as the type, and pick the analysis “*analysisYourName*” that you just performed. Select the design parameter; and its minimum and maximum should show instantly. Accept the default number of intervals as **10**. Make sure the Repeat P-loop Convergence is deselected to shorten the process time. Press **Accept**.

3) Perform the study

In the design study definition and control window shown in Figure 49, you should see the newly defined “*studyYourName*”. Hit the Run flag  to perform the study. You can also press the  icon to observe the progress.

4) See the results

After a while once the study is completed, you can review the results in a similar way as you did before in Section 7.4. Choose “**Measure**” for the Quantity option, select “**max_stress_vm,**” select the groove size dimension for the Location option, and accept the other default values. Choose Accept. Following the same procedure, create another window for “**max_displ_x**” (the direction of the pressure; depending on your coordinate system, it might not be the x direction.) The results should look like those in Figure 53. From the figure, one can see as the size increases, the maximum displacement increases along the x direction, while the maximum von Mises stress achieves the lowest when the size is about 0.13 (you may get a slightly different solution, it is perfectly fine as we know the von Mises stress is a local measure). Nevertheless, you should be able to see the convex curve which indicates there is an optimal value for the groove size leading to the minimum von Mises stress.

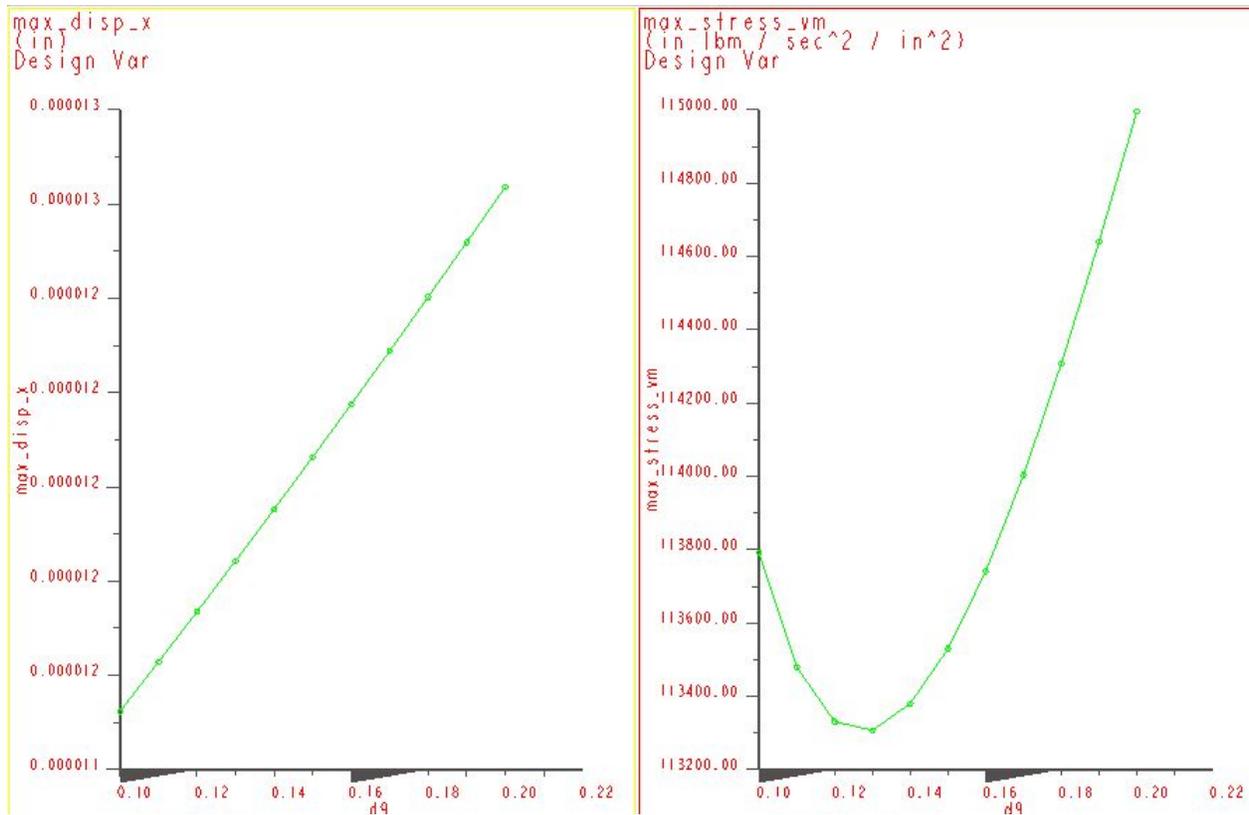


Figure 53 The sensitivities of the groove dimension to the maximum displacement along the pressure direction and the maximum von Mises stress

7.6 Design optimization

For the base part functioning as a fixture, the goal of the design optimization is to minimize the total mass, which usually associates with the cost. As a fixture, the part has to sustain certain load and the maximum deformation has to be controlled.

- 1) Define dimension relations and specify a new design parameter

Three basic elements in an optimization problem include the design variables (parameters), objective functions, and constraint functions (optional). At first, we define two design parameters. We had the groove size as the first parameter and define the height of the bottom piece as another parameter. We would like to associate some other dimensions with the height parameter, so that when the height changes, other parameters will change too. This is accomplished by defining relations. Remember you can define relations at anytime, e.g. part modeling, assembly, etc. It just happens that relations are introduced here in the design study. Relations are cool, which allows us to see the parametric feature of Pro/E.

Now back to the standard application by choosing **Applications > Standard**. Then choose **Tools > Relations**. You will see a window as shown in Figure 54. In the meantime, you will see the part dimensions are changed to symbols. (In case not, find the icon “Toggle between dimension

values and names” and click the part.) You will find the part is like somewhat shown in Figure 55.

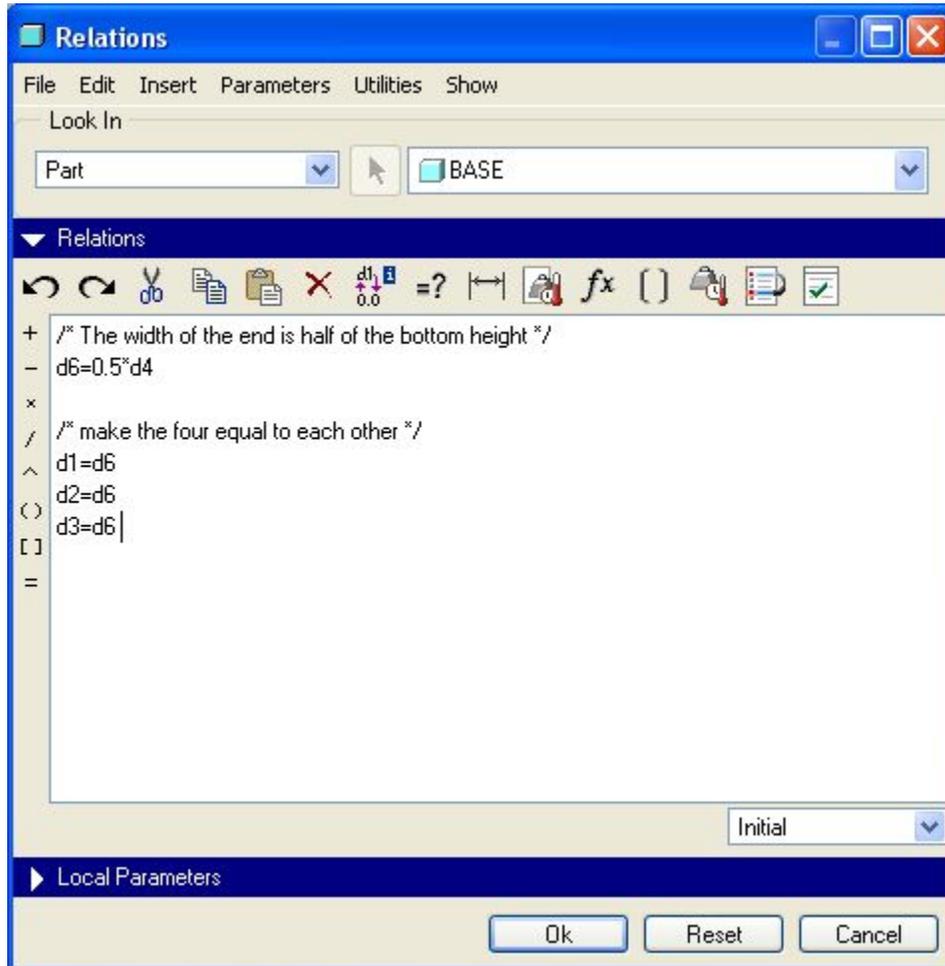


Figure 54 Relations entered in the window.

According to the symbolic names shown in Figure 55, enter the relations as shown in Figure 54. You can see the comments in the window, which should help you or others to understand the design intent, modify the part, or do other things later on. Then press OK. The relations are defined. Then press the Regeneration icon (remember which one?). You will see the dimensions for the corner are changed to 1.25. (why is that? It is because the current value for the height of the bottom piece is 2.5.) You can play with the relations by changing the value of the height and regenerating the model, you will see the dimensions of the corners change accordingly. One thing though, you can modify the corner dimensions any more using the conventional ways. Why? Because they become driven parameters and controlled by the bottom height dimension, which is also called driving dimension. Note the symbolic names are very likely different on your computer, do not copy my equations! Use your own dimension names!

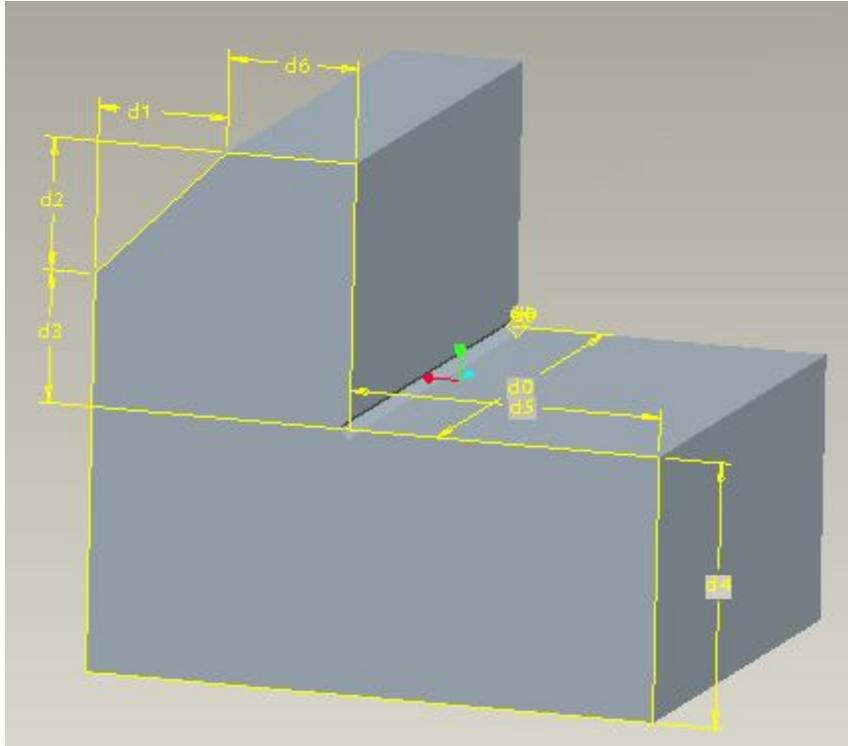


Figure 55 Symbolic names for dimensions.

Now define the dimension d4 as another design parameter as you did before for the groove size. (*HINT: you have to return to the Mechanics application*). Enter the minimum value as [1] and maximum [3].

2. Define a new design study

Thus far, we have defined two design parameters. The first is the groove dimension, d9, which is defined in the sensitivity study; the second is d4, the height of the bottom piece. The following step is to define an optimization design study to specify the optimization goal and constraints.

Choose **File > New Design Study**. Create a new study called “*StudyOptimYourName*”. Enter the **Type** as **Optimization**. Select Goal as “Minimize,” Measure: “total_mass.” Select Limits on Measures, Create, make two constraints: $\text{max_disp_x} < [1.25\text{e-}5]$ and $\text{max_stress_vm} < [1\text{e}5]$. Select “AnalysisYourName,” “loadset1,” and the two design parameters. The range of design parameters is from the minimum to maximum. The initial value for d9 is chosen as [0.18]; and the initial value for d4 is chosen as [1.5]. Accept the default **Optim Convergence** 1% and change the **Max Iterations** to [10]. Select Repeat P-Loop Convergence for a more accurate solution.

2) Perform the design optimization

Run the study. It will take a longer while than any studies before. The optimal design is found at **d9 = 0.2** inch and **d4 = 1** inch. The total mass is **6.81 lbm** with both constraints satisfied.

3) Results

Generate the plots for the maximum von Mises stress and displacement along x to check the constraints and the final analysis results as shown in (you'd pick the directory "studyOptimYourName" to generate the results.) Did the optimal design surprise you? Quite different from the original design, isn't it?

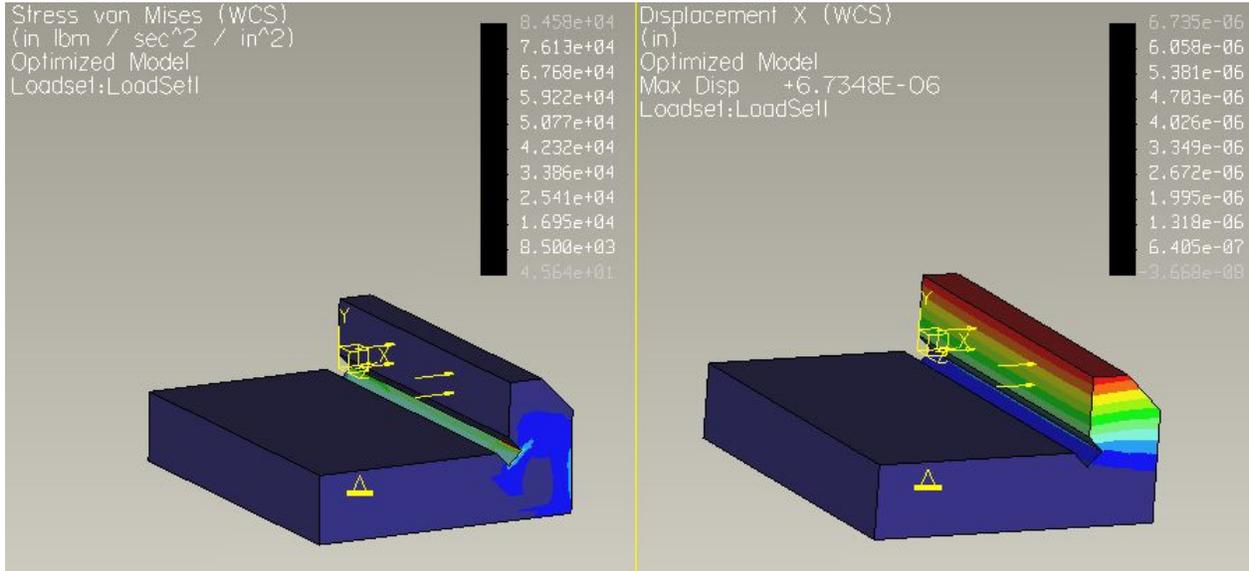


Figure 56 The maximum stress and displacement plots for the optimal design. One can generate the convergence plot as well to see the progress of the optimization.

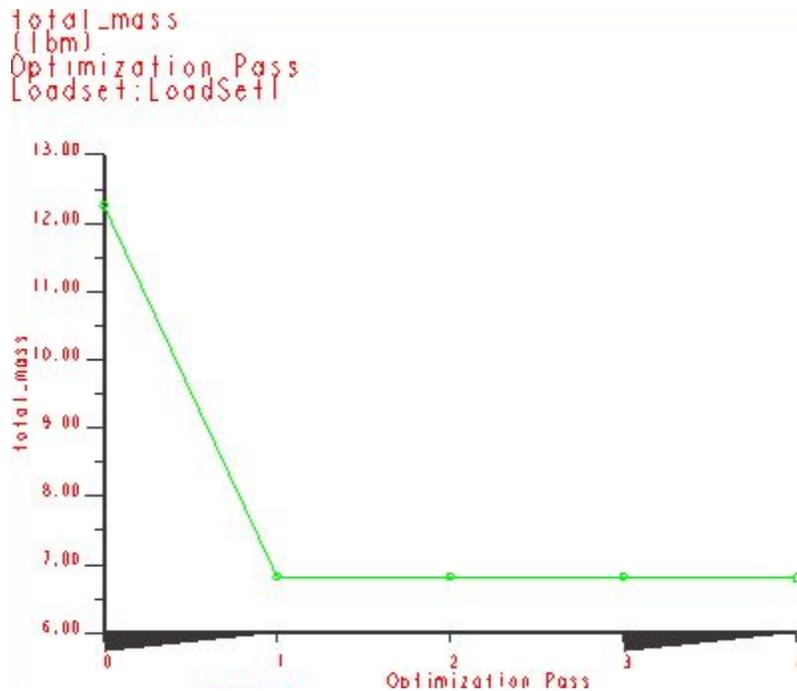


Figure 57 The convergence plot for the objective function.

The convergence plot Figure 57 tells us the history of optimization. To generate the plot, just create a graph plot, using the **total_mass** as the **Measure** and accept the other defaults. As one can see, the total mass converges to the minimum as the last iterations.

8 Pro/Mechanica – Standard Static Analysis

8.1 Objectives

To create a very simple solid model and perform a static stress analysis using Pro/MECHANICA.

8.2 Procedures

1. Creating the Geometry of the Model

- Download or Create a new solid part called bar using Pro/E with units of mm-N-s (millimeter-Newton-second) as shown in Figure 1.

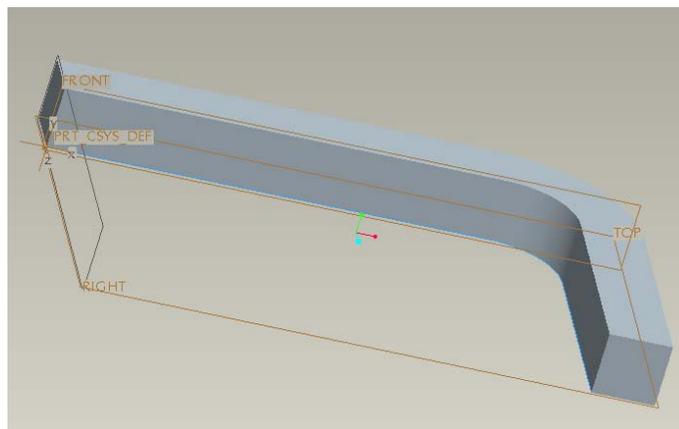


Figure 1 Bar

- Hints to create the part:

- a) Create curve on the TOP datum plane as shown in Figure 2

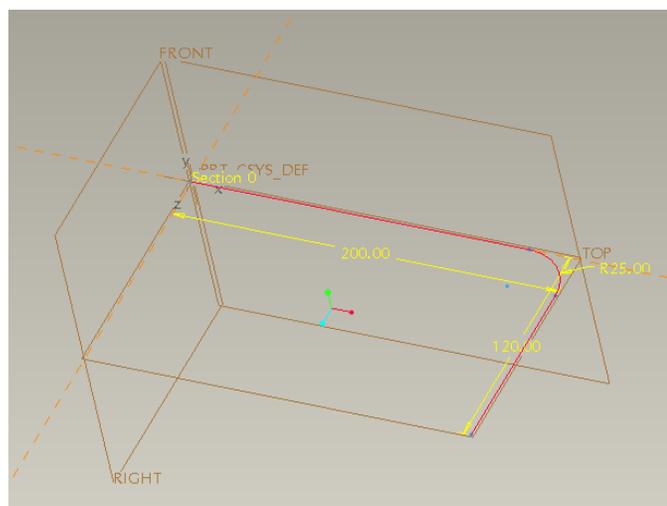


Figure 2 Curve

- b) Select Variable Section Sweep to create the square cross section as shown in Figure 3.

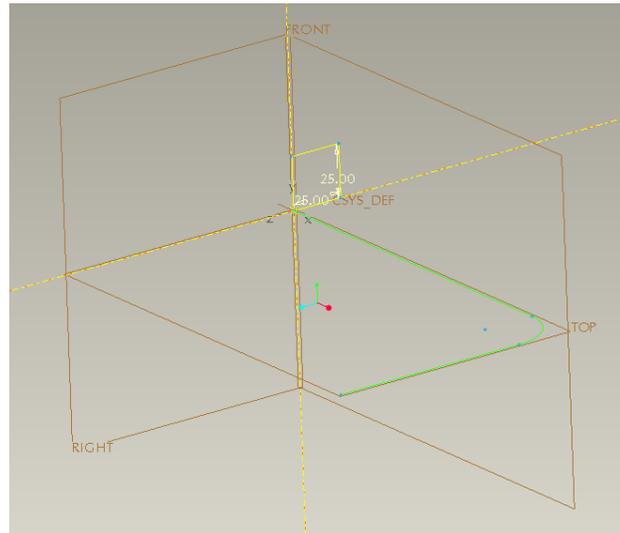


Figure 3 Square cross section

- c) The final part is created as shown in figure 4. Save the model

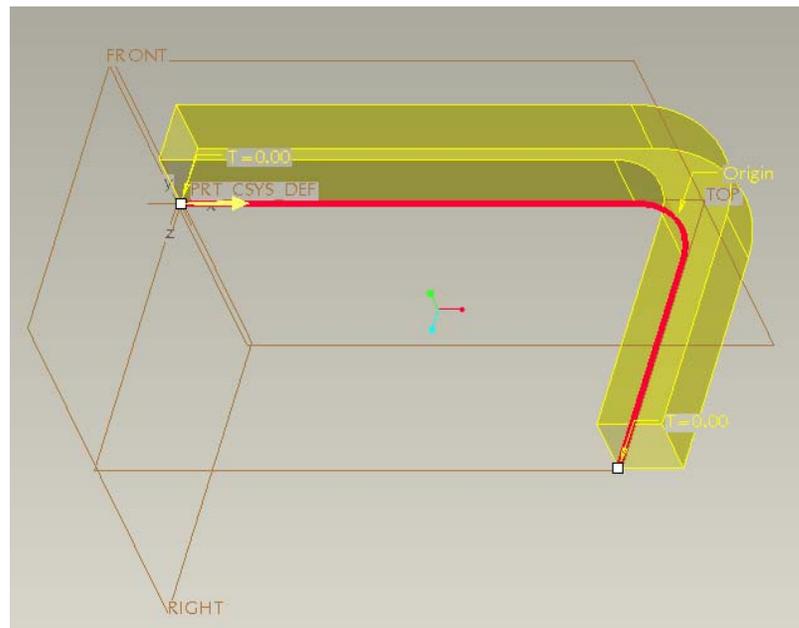


Figure 4 Bar

2. Setting up the FEA Model

- Start MECHANICA
Applications → Mechanics
Model Type → Structure. Window pops up as shown in Figure 5 → click OK.

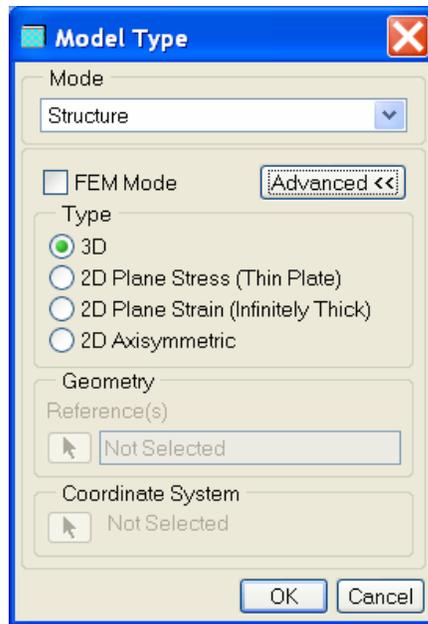


Figure 5 Modal type

- Apply the constraints
 Top menu → Insert → Displacement Constraint. Window pops up as shown in Figure 6. Enter name as fixed-face and pick the surface as shown in Figure 7.

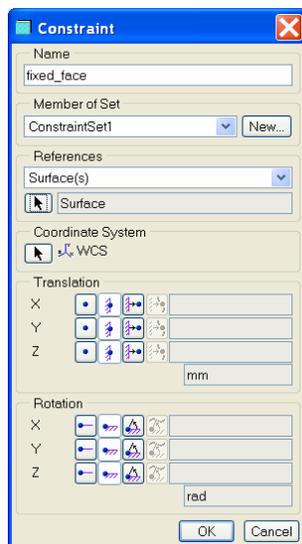


Figure 6

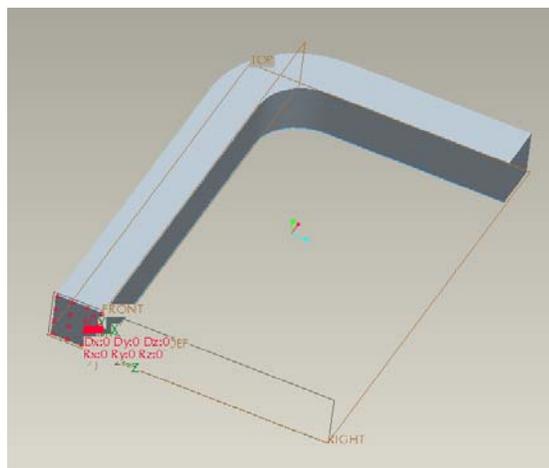


Figure 7

- Apply the loads
 Top menu → Insert → Force/Moment Loads. Window pops up as shown in Figure 8. Enter name as end load and pick the surface as shown in Figure 9. Enter Force components as shown in Figure 8.

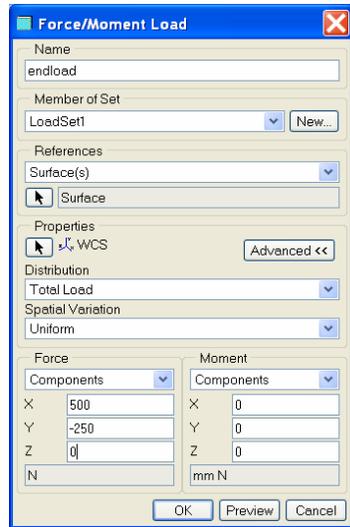


Figure 8

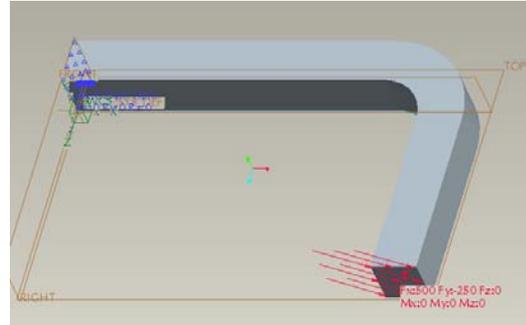


Figure 9

- Change the Load Arrows display to Arrow Tails Touching
 Top Menu → View → Simulation display → Settings. Uncheck Value and Arrows Scaled. Check Arrow Tails Touching as shown in Figure 10. The bar is shown in Figure 11.

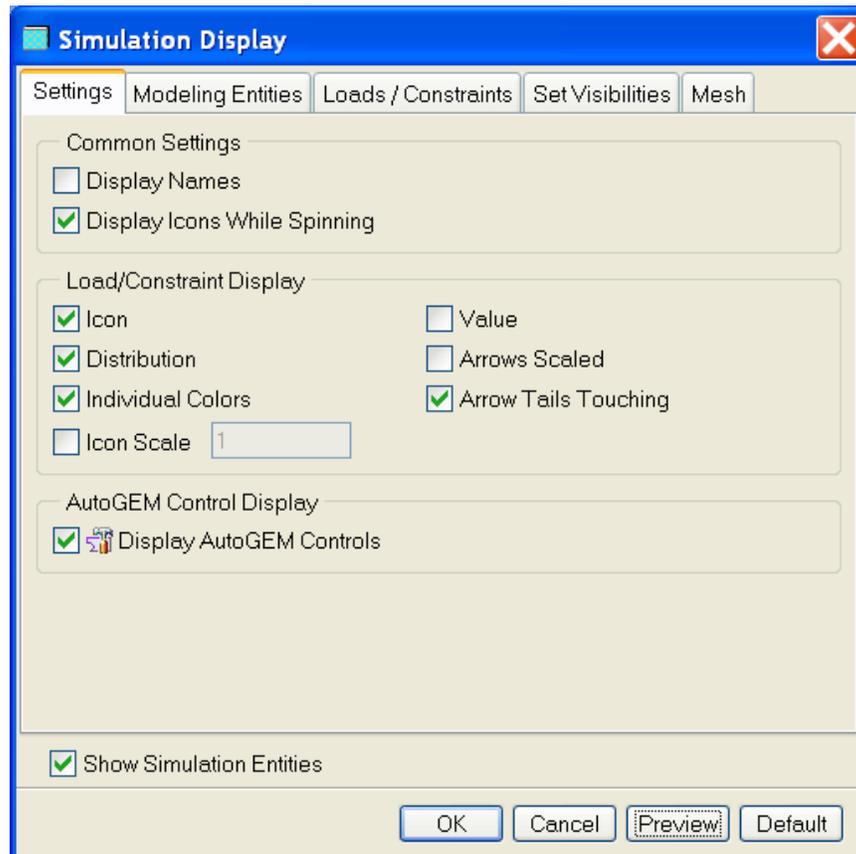


Figure 10

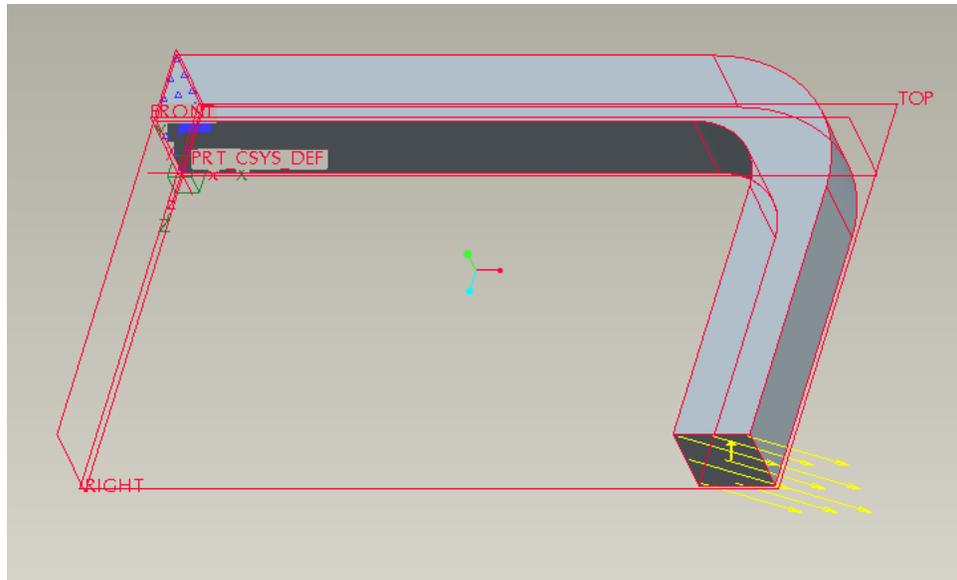


Figure 11

- Define materials

Top menu → Properties → materials. Window pops up as shown in Figure 12. Click on STEEL and Assign → Part. Pick the bar.

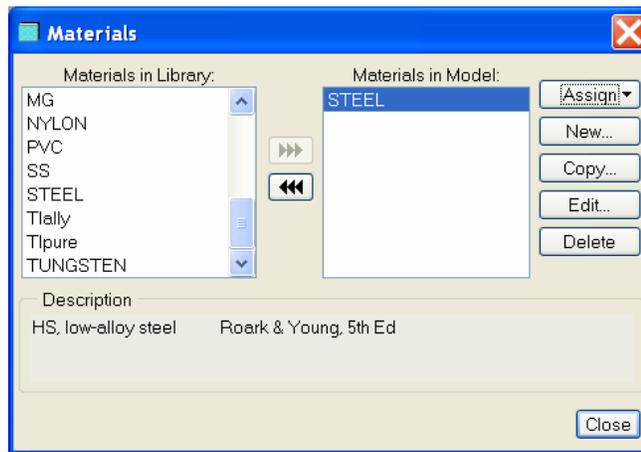


Figure 12

- Set up the Analysis

Top menu → Analysis → Mechanics Analyses/Studies. Window pops up as shown in Figure 13. File → New Static and another window pops up as shown in Figure 14. Enter a name for the analysis bar_1. Change method to Quick Check. Click OK to exit window. The Analyses and Design Studies windows as shown in Figure 15. In the pull-down menu select Info → Check model. There are no errors in the model.

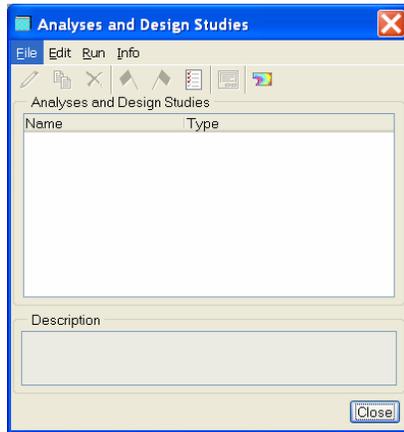


Figure 13

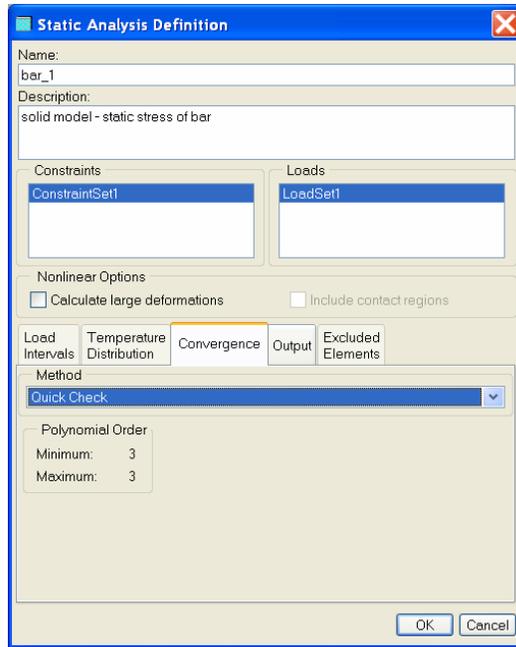


Figure 14

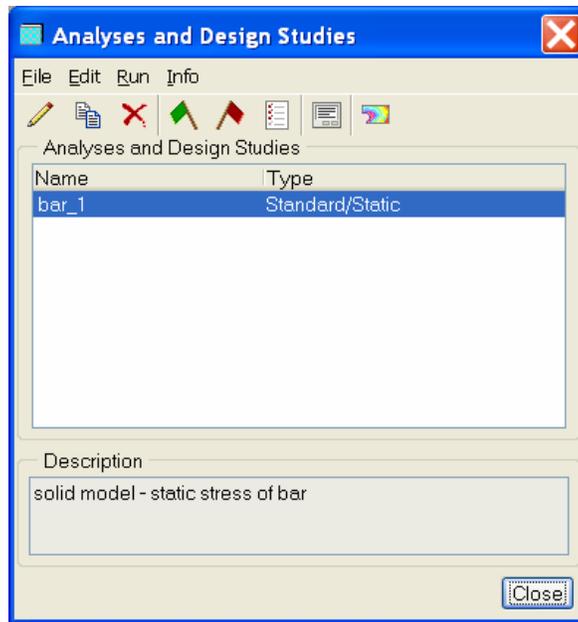


Figure 15

3. Setting up and Running the Analysis

- From Figure 15, select Run → Settings. Window pops up as shown in Figure 16. Check Memory Allocation to half the physical memory in the computer. For design60 – 70 enter 512MB and other to 256MB.

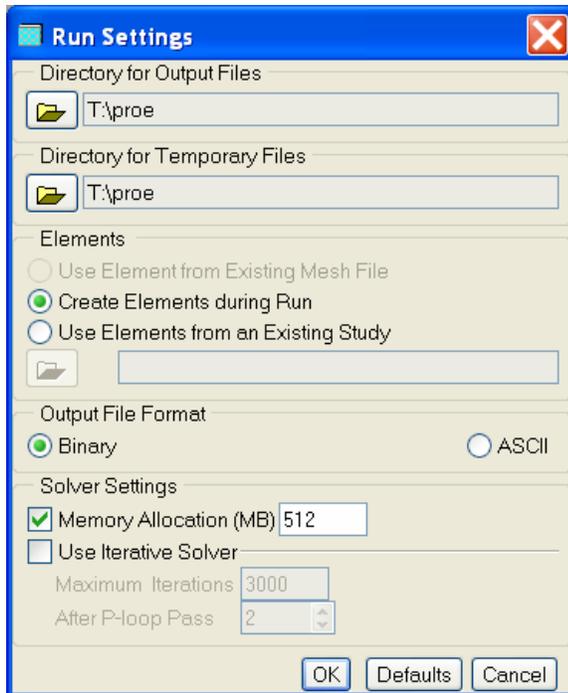


Figure 16

- From Figure 15, select Run → Start. You can follow its progress by selecting the Display Study Status button in the toolbar. The results is given below:
 max_disp_mag: 5.407248e-01 (mm)
 max_stress_vm: 4.434576e+01 (MPa)
- Change the Quick Check analysis to Multi-Pass Adaptive. From Figure 15, select Edit. Window pops up as shown in Figure 17. From the Method list, select Multi-Pass Adaptive. Set the maximum polynomial order to 9 and Convergence to 5. Select Start Run and open the Study Status windows to observe the iterations as they happen.

The run converges on pass 7. The result is giving below:

Run Settings

Memory allocation for block solver: 512.0

>> Pass 7 <<

Measures:

Name	Value	Convergence
max_disp_mag:	5.562450e-01	0.1%
max_stress_vm:	4.368421e+01	0.1%

RAM Allocation for Solver (megabytes): 512.0

Total Elapsed Time (seconds): 4.91

Total CPU Time (seconds): 4.47

This run was performed on design68 (3Ghz Pentium 4)

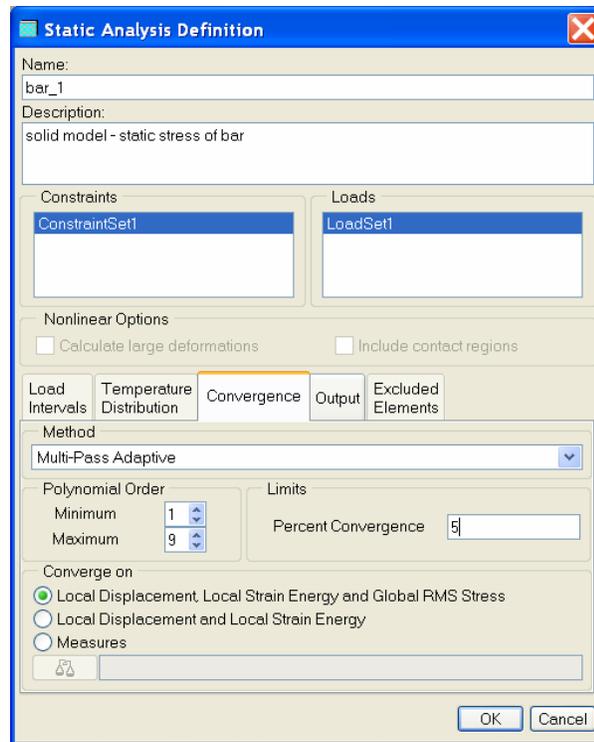


Figure 17

4. Display the results

- Top menu → Analysis → Results. The untitled window pops up as shown in Figure 18. From the pull-down menus, select Insert → Result Window. The Result Window Definition pops up as shown in Figure 19. Enter the name as shown in the figure. Click on the display option. The expended window pops up as shown in Figure 20 and check the box beside Show Element Edges. Click Ok and Shown. The result is shown in Figure 23.

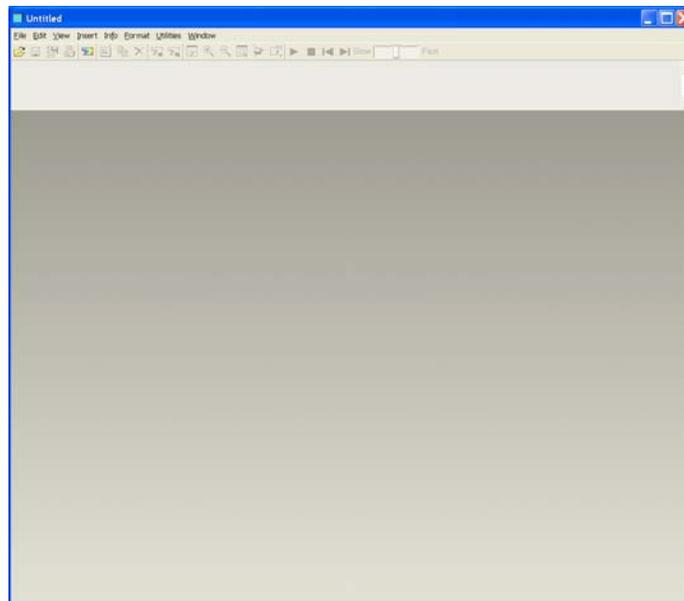


Figure 18

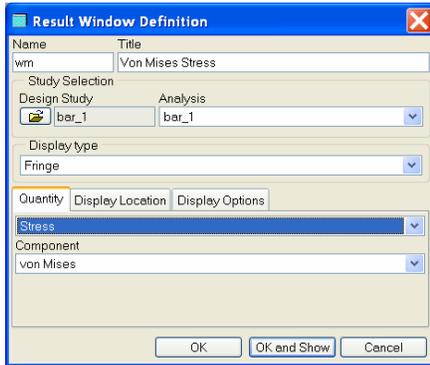


Figure 19

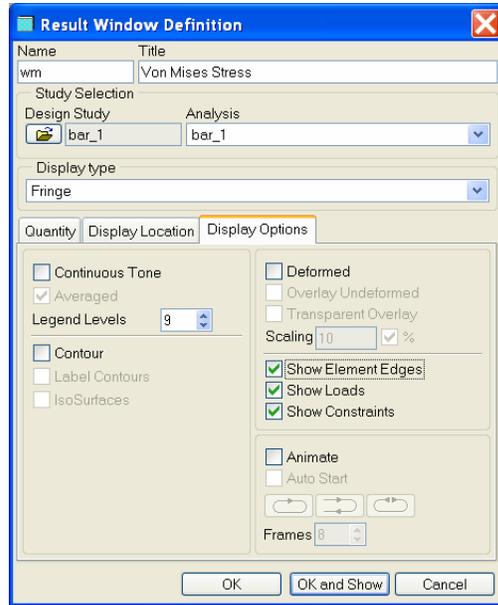


Figure 20

- From Figure 18 top menu -> Edit → copy and create deformation window. Enter name and display options as shown in Figure 21 and Figure 22. Click OK and Show. The result is shown in Figure 23.

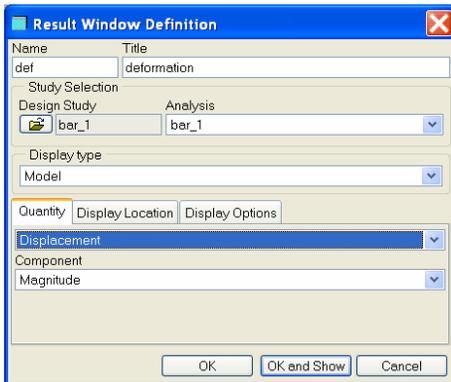


Figure 21

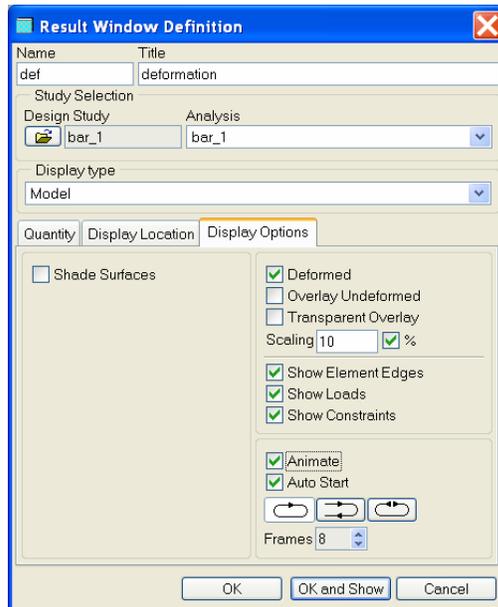


Figure 22

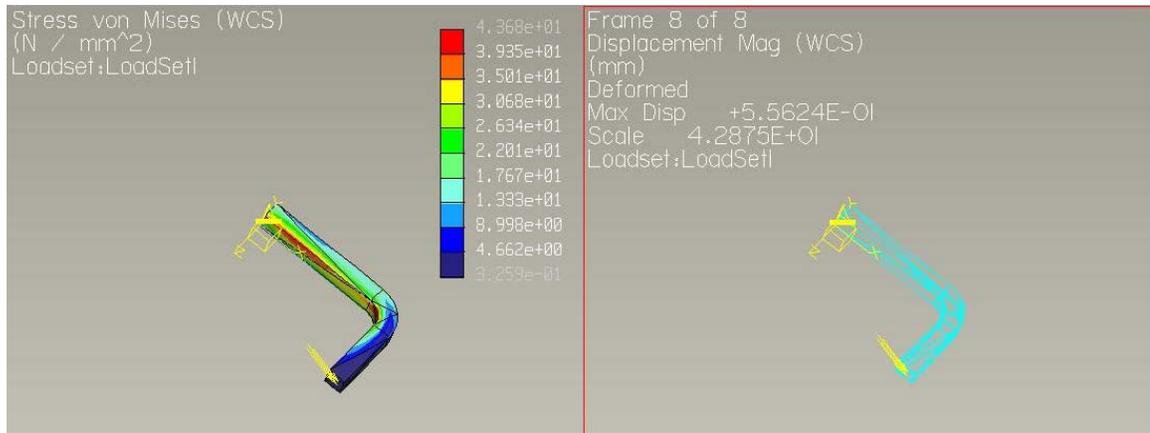


Figure 23

- Create the Multi-Pass Adaptive convergence behavior graph as shown in Figure 24. The Figure 25 display two results windows (von Mises stress and stress convergence) side by side

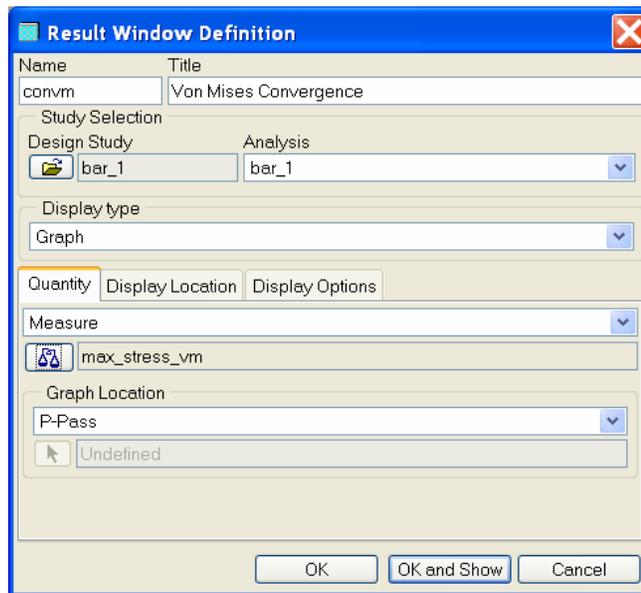


Figure 24

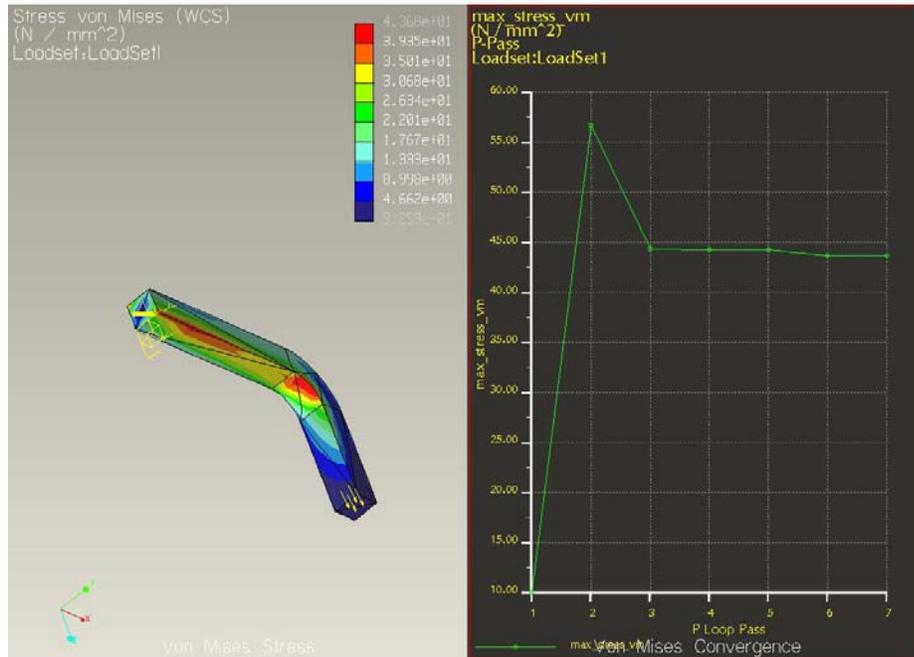


Figure 25

- Create Deformation Convergence graph as shown in Figure 26.

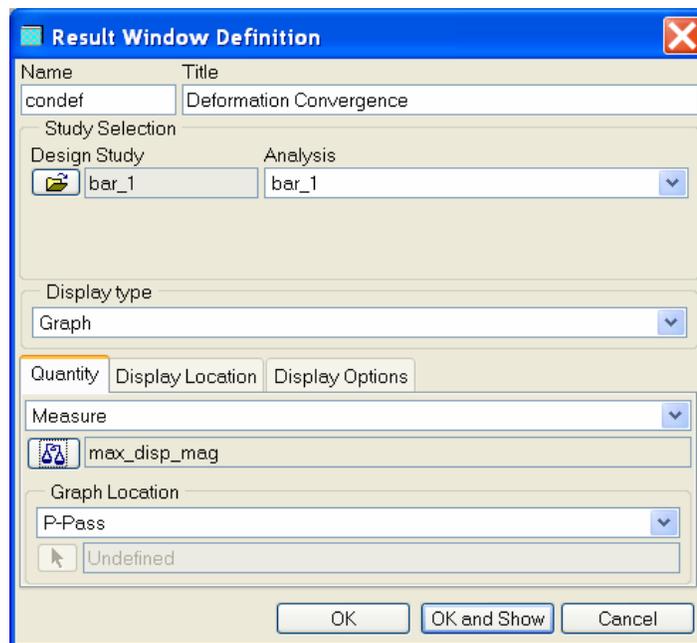


Figure 26

- Define another Strain Energy Convergence graph as shown in Figure 27.

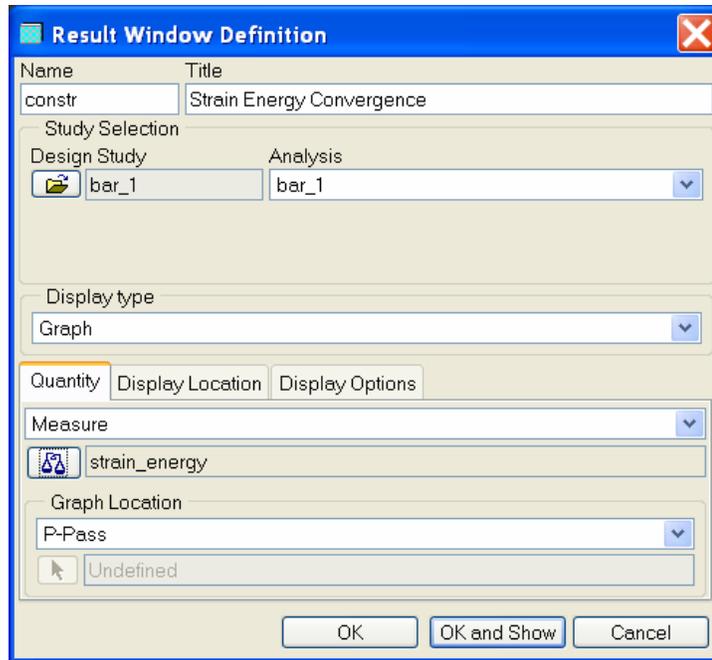


Figure 27

- We have now defined five results windows. To see a list of the defined windows, select View → Display This will open the windows shown in Figure 28.

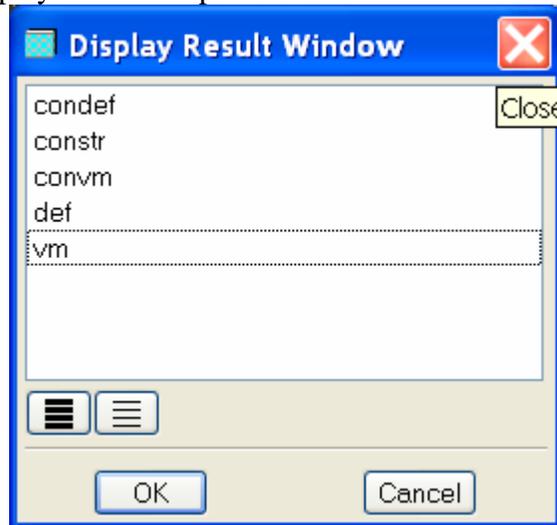


Figure 28

- Show the deformation animation in Figure 29. Select the entry def in the Figure 28 and then select OK.

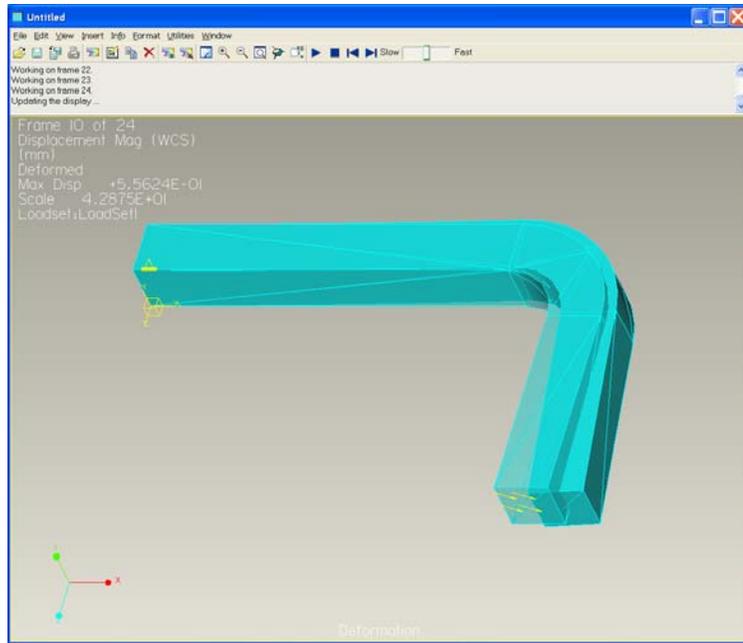


Figure 29

- Show the Von Mises Stress finger plots in Figure 30 and Figure 31. Figure 30 plots with contour level 8 and element edges. Figure 31 plots with contour level 9, continuous tone and no edges).

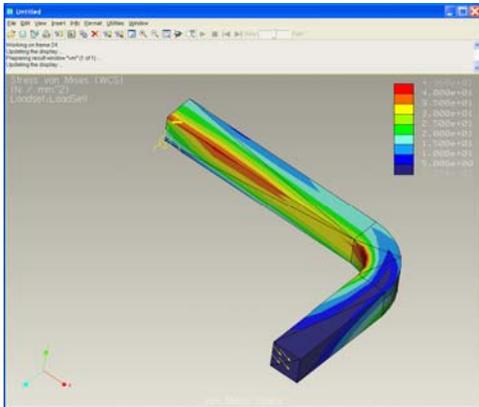


Figure 30

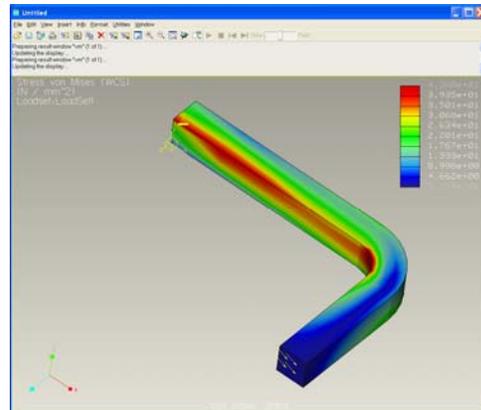


Figure 31

- Select Format → Result Window to change the background color and the visibility of various elements in the windows.
- Select Format → Legend to show the minimum and maximum values in the legend scale.
- Select Info → Model Max to locate the Maximum stress on the model.
- Select Info → Dynamic Query to display the stress level at the cursor location.
- Select Insert → Cutting/Capping Surfs. The window is shown in Figure 32. Select apply. The model is now sliced on the cutting plane as shown in Figure 33.

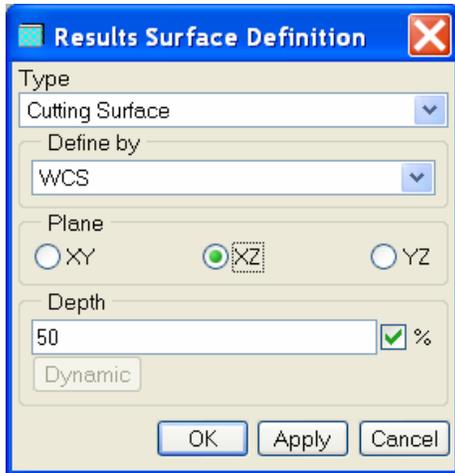


Figure 32

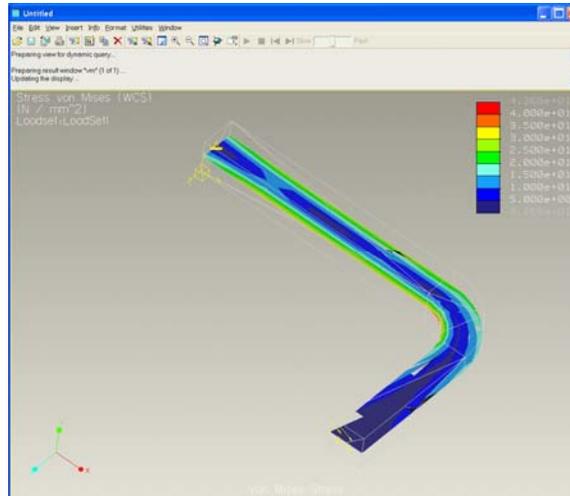


Figure 32

- The three convergence plots are shown in Figure 33, 34 and 35.

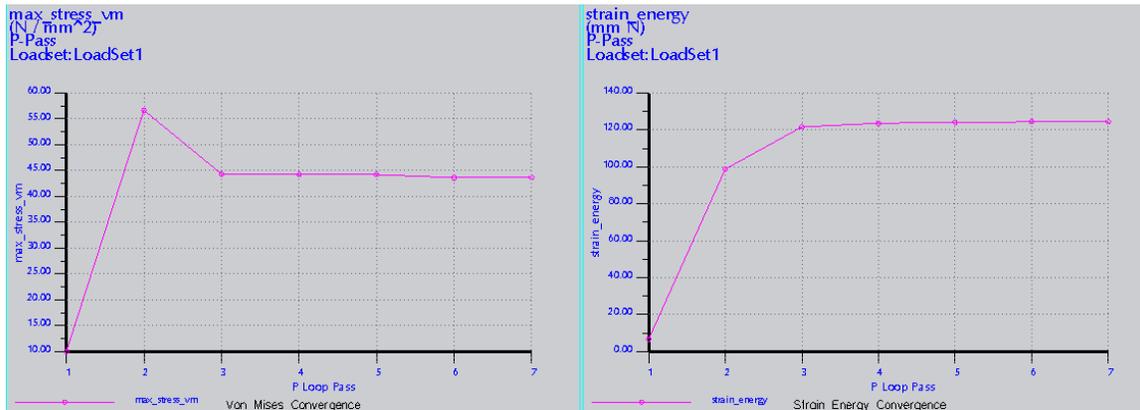


Figure 33 Convergence of Von Mises stress Figure 34 Convergence of total strain energy

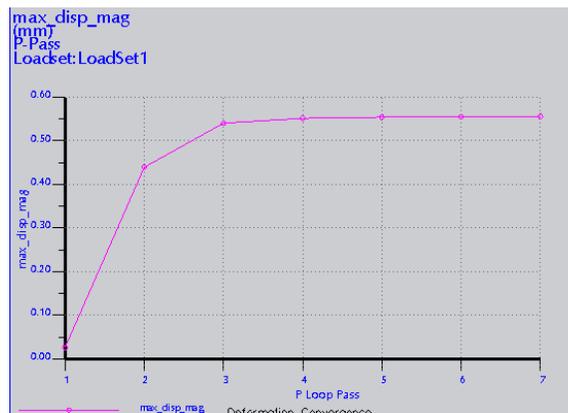


Figure 35 Convergence of maximum displacement.

- File → Save as to exit Pro/E.

9 Automated CNC Tool Path and G-Code Generation for Volume Milling

9.1 Objectives

- A. To create a solid part
- B. To create work piece
- C. To perform the machining operation setup
- D. To define the machining operation
- E. To View tool path simulation and cutter location file

9.2 Procedures

- A. Produce the part model. The part consists of two features, together forming a block (4" x 8" x 1.5") with raised letters "CAM" as shown in Figure 1.10.
 - 1) Start the Pro/E program. Windows menu items, Start → Programs → PTC25 → Pro ENGINEER→ Click on the icon  Pro ENGINEER.
 - 2) Set working directory. File → Set Working Directory, select working directory.
 - 3) Create the part name. Pro/E main menu, File → New, select Part in the New window, enter part name: CAM.
 - 4) Start of the part. The part contains some features already. The main graphics area shows 3 datum planes and a coordinate system as shown in Figure 1.1.

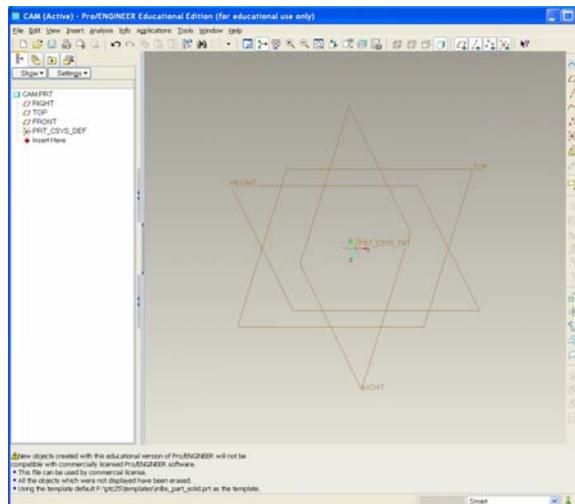


Figure 1.1 Start of the Part

- 5) Create a 4" x 8" x 1.5" rectangular block as shown in Figure 1.8.
 - Choose INSERT → EXTRUDE from the menu. You should see a new toolbar

called dashboard appear as shown in Figure 1.2.

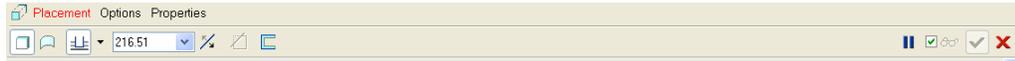


Figure 1.2 The Dashboard

- Click on the placement on the extrude dashboard (Figure 1.3) and select define. The sketch dialog window appears as shown in Figure 1.4.

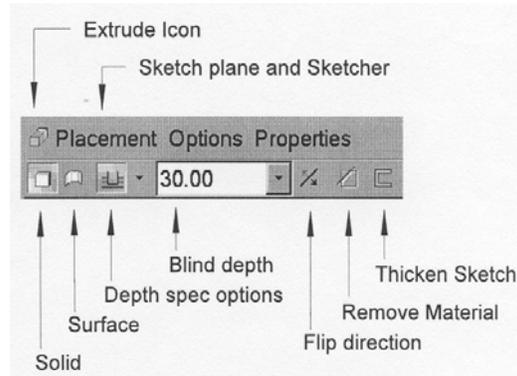


Figure 1.3 The Extrude Dashboard



Figure 1.4 Sketch dialog

Choose the datum place FRONT by clicking on it in the graphics window. Accept the default on the sketch dialog window and just click on the Sketch button. References sub-window pop up as shown in Figure 1.5 and click close to start sketch.

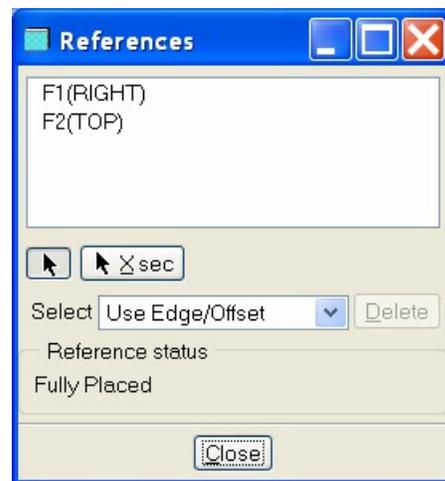


Figure 1.5 References dialog

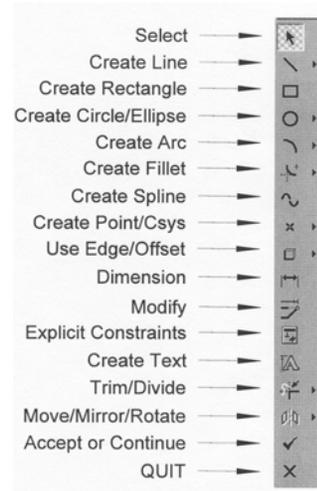


Figure 1.6 The Sketcher toolbar

- Choose Create Rectangle icon on Sketcher Toolbar (Figure 1.6) to sketch a rectangle for extrusion in plane FRONT by clicking on the bottom-right and top-left corners of the rectangle in the drawing windows. Click middle mouse to

finish drawing the rectangle.

- Modify the dimensions to 4 x 8 by double clicking on the dimensions on the rectangle. To end sketching choose Accept icon on Sketcher Toolbar (Figure 1.6) and click OK in the Section dialog.
- Extrude the rectangle to form the solid block. Enter depth value as 1.5 into the depth field of the extrude dashboard (Figure 1.3) and click the Accept tick to finish (Figure 1.7).
- View the rectangular block as shown in Figure 1.8. View → Orientation → Standard Orientation.

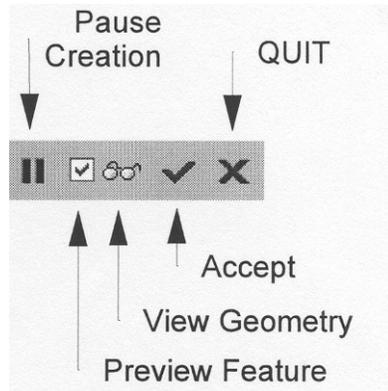


Figure 1.7 Dashboard controls

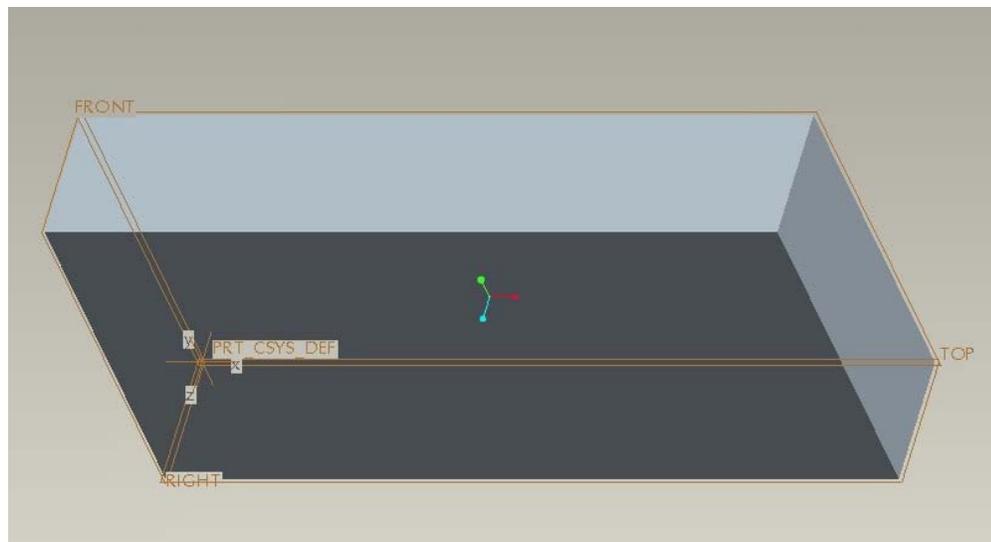


Figure 1.8 Block

- 6) Create 2" x 6" x 0.5" letters "CAM" on top of the block as shown in Figure 1.10.
 - INSERT → EXTRUDE from the menu. The dashboard appears as shown in Figure 1.2.
 - Click on the placement on the extrude dashboard (Figure 1.2) and select define. The sketch dialog window appears as shown in Figure 1.9. Choose top surface of the block as sketch plane and accept the default on the sketch dialog window and

click on the Sketch button.

- Produce letters. Sketch → Text from the menu and pick two points on the sketch plan to determine the height of the text and enter letters CAM as shown in the figure 1.10. Modify the size of the text to 2 x 6 and locate the text to the center of the block as shown in figure 1.11.

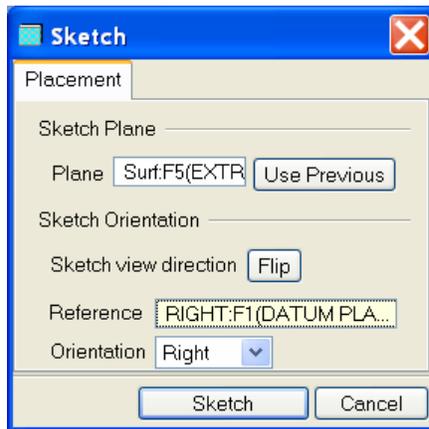


Figure 1.9 Sketch dialog

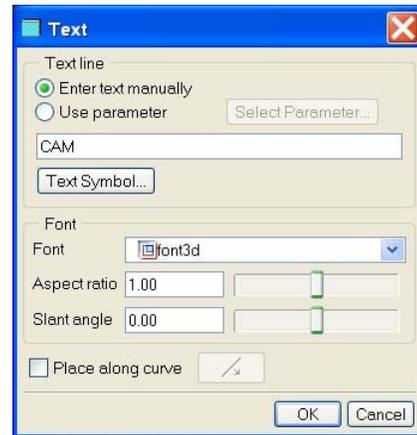


Figure 1.10 The Text dialog

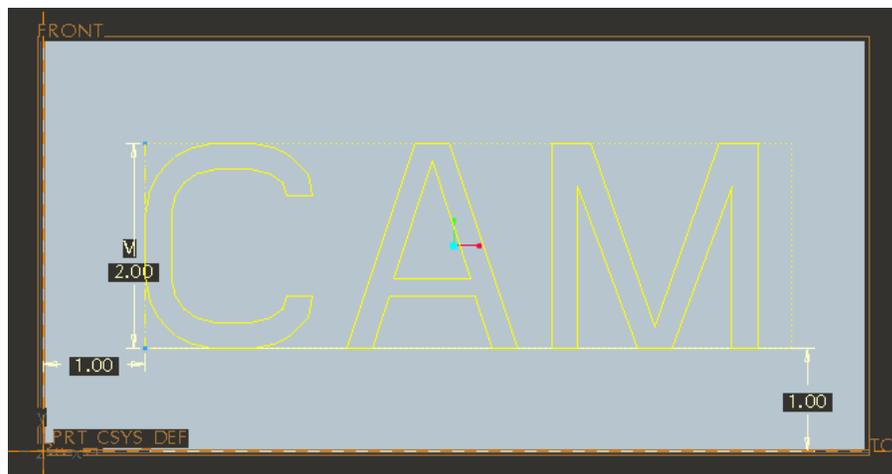


Figure 1.11 Letter CAM

- Extrude the text to 0.5” out of the block.
- View the created part (Figure 1.12). View → Orientation → Standard Orientation.
- Save the part. File → Save and Exit or Close to continue.

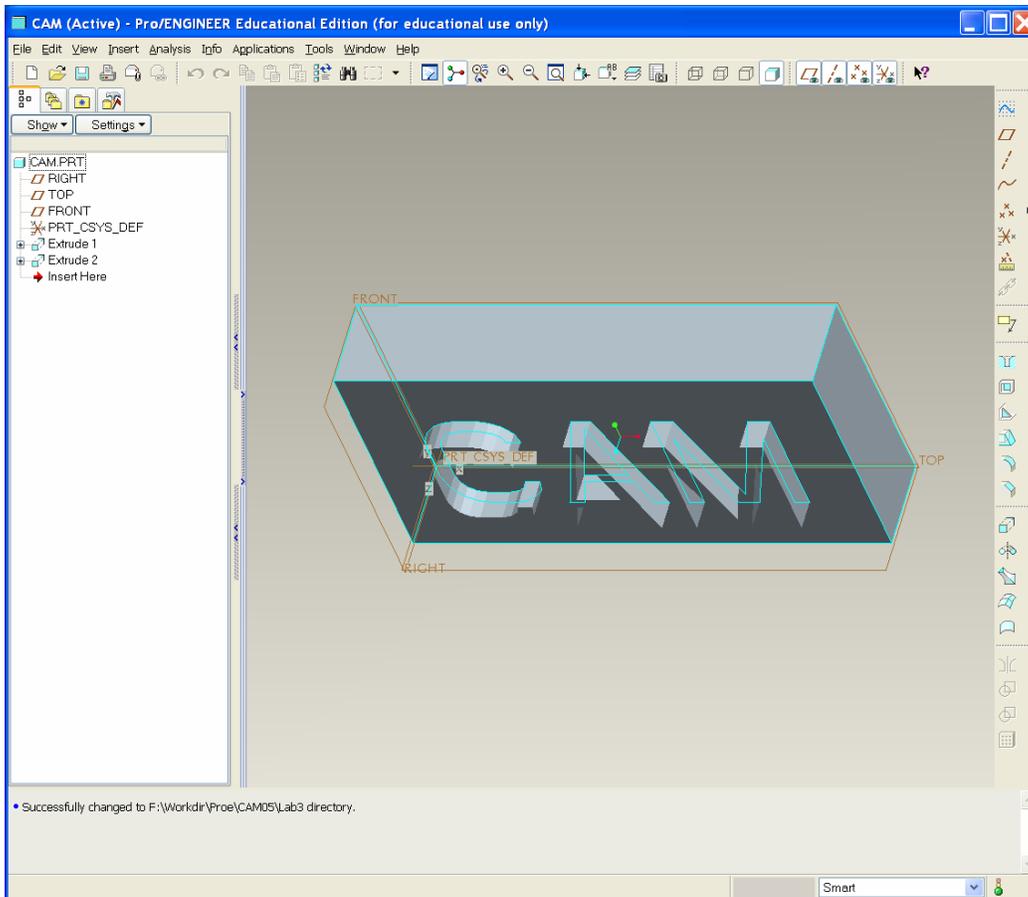


Figure 1.12 Part model

B. Create the workpiece. The workpiece represents the raw stock of material from which the part will be machined. Pro/E refers to this procedure as an assembly operation.

- 1) Start the Pro/E program. Windows menu items, Start → Programs → PTC25 → Pro ENGINEER → Click on the icon  Pro ENGINEER.
- 2) Set working directory. File → Set Working Directory, select working directory.
- 3) Create the part name. Pro/E main menu, File → New, select Manufacturing in Type window and NC Assembly in Sub-type window, enter name: MCAM as shown in Figure 1.13

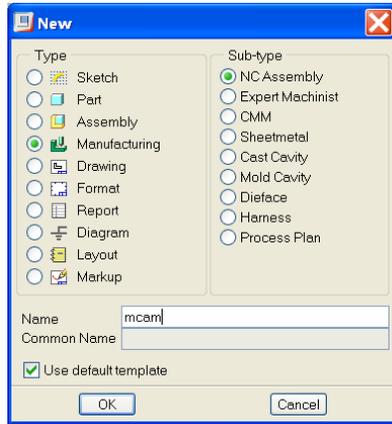


Figure 1.13 Creating a new file



Figure 1.14



Figure 1.15

- 4) Load the part. From MANUFACTURE menu, select Mfg Model → Assemble → Ref. Model. Select CAM.prt in the open window. The component Placement window pops up. Select  to place the part at default location as shown in Figure 1.16. Click OK to close Component Placement windows as shown in Figure 1.17.

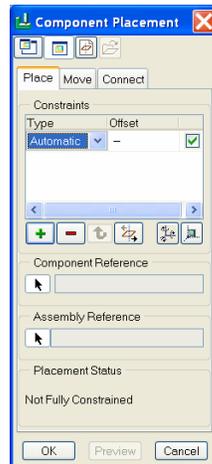


Figure 1.16

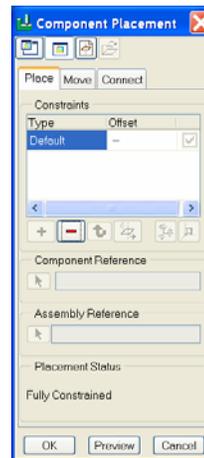


Figure 1.17

- 5) Create a workpiece 6" x 10" x 2.25" . From MANUFACTURE menu, select Mfg Model → Create → workpiece. Enter a name for the workpiece: BLOCK. Select Solid → Protrusion → Extrude → Solid → Done. Select NC_ASM_FRONT as sketch plane and NC_ASM_RIGHT as reference plane in Figure 1.19. Create a rectangle 6" x 10" surrounding the CAM part as shown in Figure 1.20 and extrude it to 2.25" as shown in Figure 1.21.

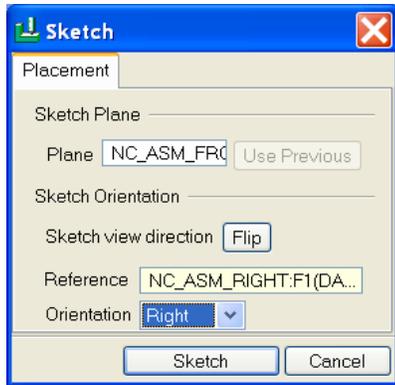


Figure 1.19

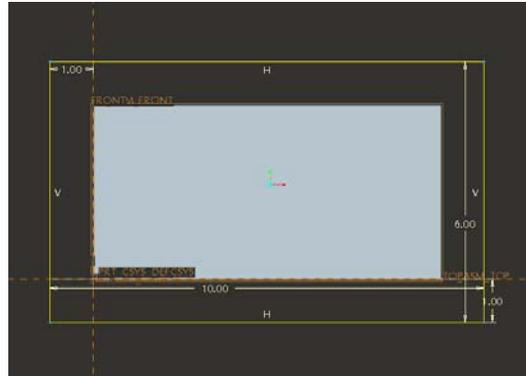


Figure 1.20

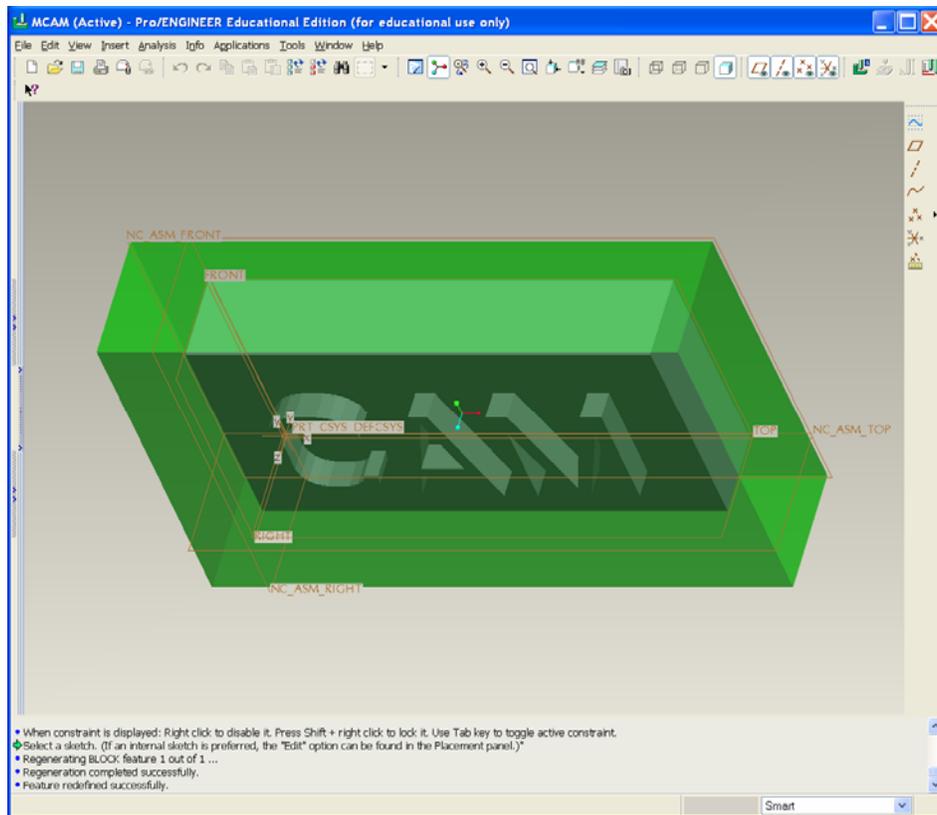


Figure 1.21 Reference Model and Workpiece

C. Perform the machining operation setup. The setup consists of defining the type of machine to use. It also requires defining a coordinate system if one does not already exist and a retraction plane for the cutting tool. The coordinate system must match the mill orientation and the part zero.

- 1) MFG Setup → Operation. Operation Setup window pops up automatically as shown in Figure 1.22.
- 2) Define NC machine. Click NC machine icon  in Figure 1.22, Machine Tool Setup windows pop up as shown in Figure 1.23. Enter the parameters as shown below
 Machine name: Victor
 Machine type: Mill
 Number of Axis: 3
 CNC control: FANUC
- 3) Define Machine Zero. Click Reference Machine Zero icon  in Figure 1.22. Create MACH CSYS as Figure 1.24. Pick workpiece to create coordinate system in. Pick 3 reference planes as Figure 1.27 (click two sides and top planes while holding down Ctrl key) to place origin as shown in Figure 1.25. Orient X, Y axes as shown in Figure 1.26.
- 4) Define retract plane. Click Retract Surface icon  in Figure 1.22. Retract Selection window pops up (Figure 1.28). Choose Along Z Axis, and in the panel of Enter Z Depth, input 0.5, click OK to close the window. The retract plane is shown in Figure 1.29.
- 5) Complete Operation setup. Click OK to close operation setup as Figure 1.30. MFG Setup → Done → Return.

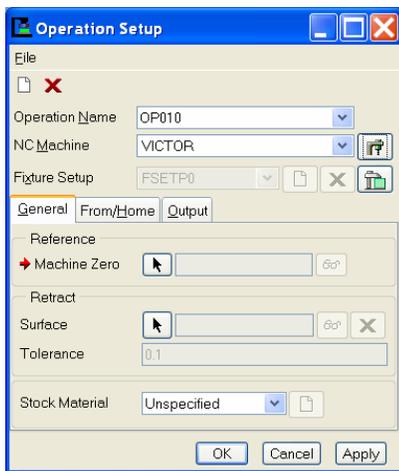


Figure 1.22

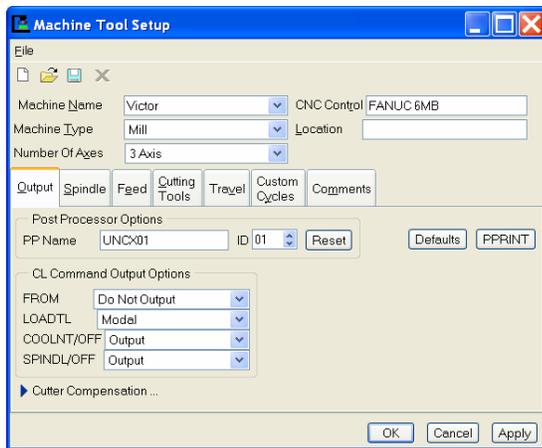


Figure 1.23



Figure 1.24

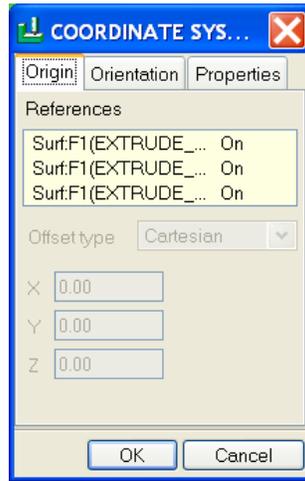


Figure 1.25

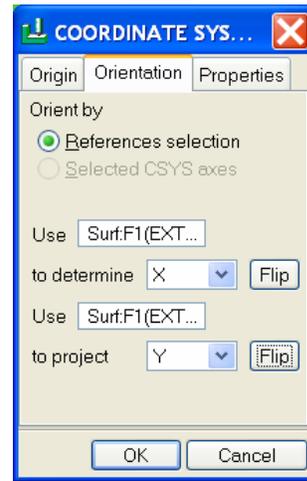


Figure 1.26

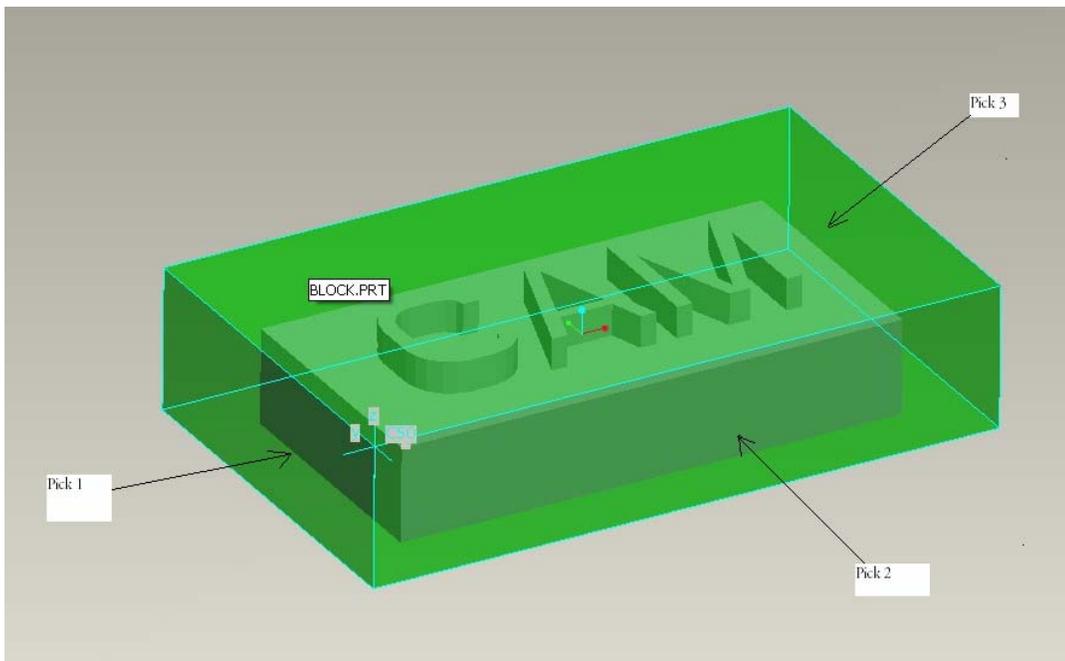


Figure 1.27 Defining the Coordinate System

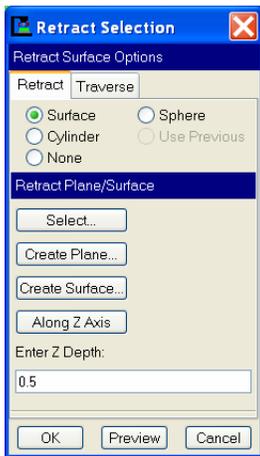


Figure 1.28

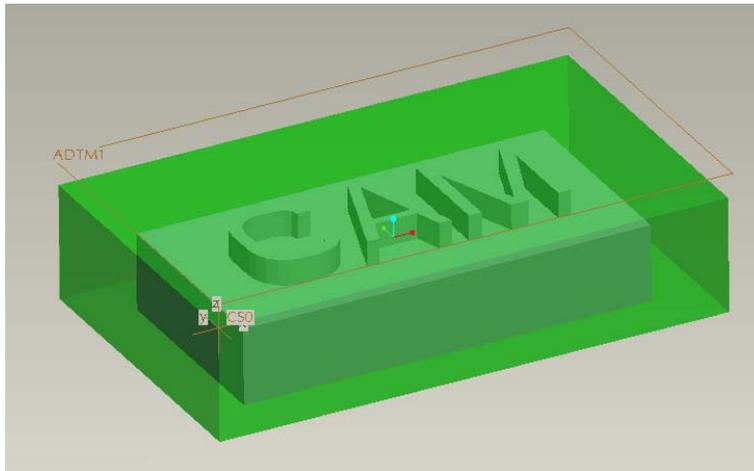


Figure 1.29 Defining the Retract Surface

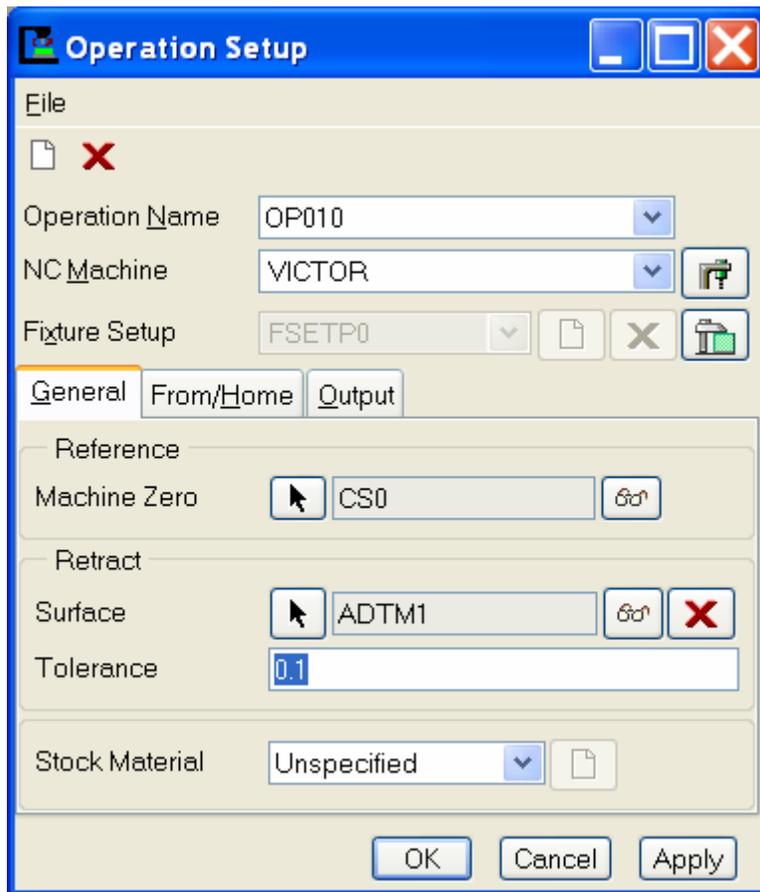


Figure 1.30 Completed Operation Setup

D. Define the machining operations. The setup consists of defining the type of tool to use and machining parameters (tools size, cutting speed, etc.), and specify the volume of material to be removed.

- 1) Machining → NC Sequence → Volume → 3 Axis → Done.
- 2) SEQ SETUP window pops up. Ensure that tool, parameters, retract and volume are checked and then choose DONE. (Figure 1.31).

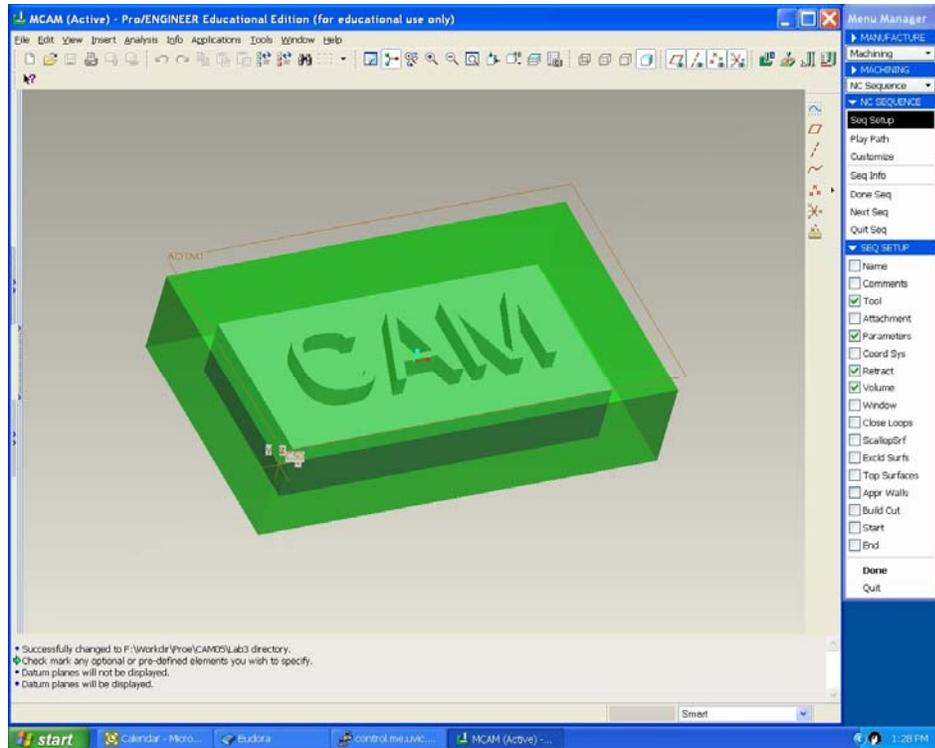


Figure 1.31

- 3) Tool setup table pops up. Enter the tool values as shown in Figure 1.32 and APPLY OK.

Cutter_Diam	.25
Length	4

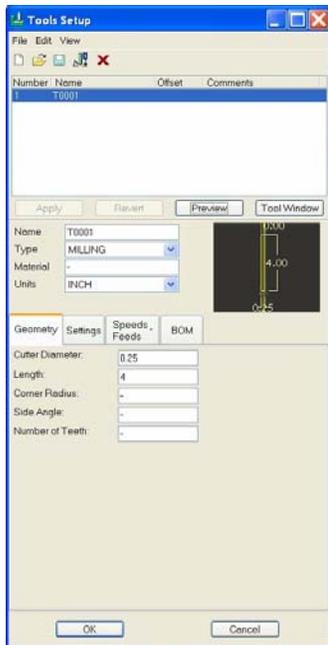


Figure 1.32

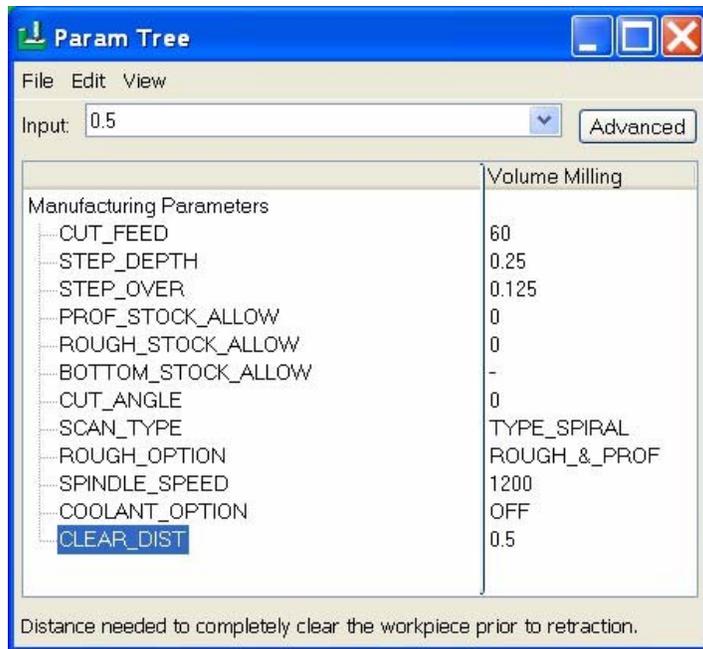


Figure 1.33

- 4) MFG PARAMS → Set. Param Tree window pops up. Input or change the values as shown in Figure 1.33. Select Advance button to change MACHINE → CIRC_INTERPOLATION → POINTS_ONLY as shown in Figure 1.34.
- 5) Retract Plane → Retract Selection window pops up. Select ADTM1 created in previous operation setup (Figure 1.35) and click OK.

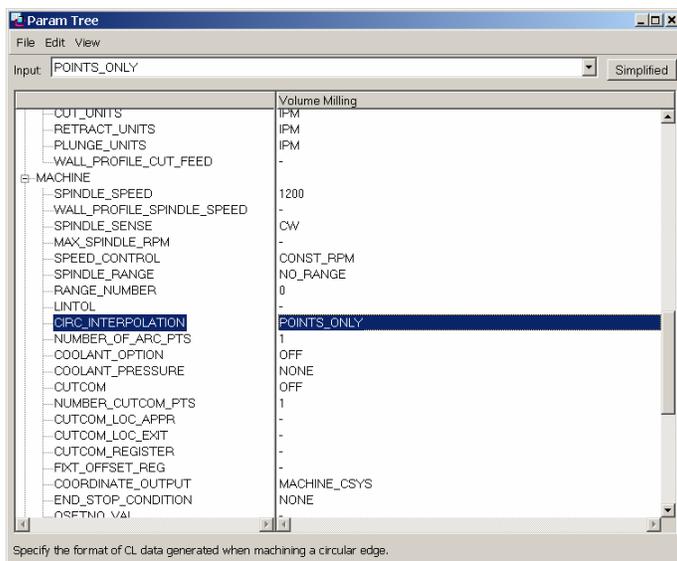


Figure 1.34



Figure 1.35

6) Create Mill Volume. To specify the volume of material to be removed.

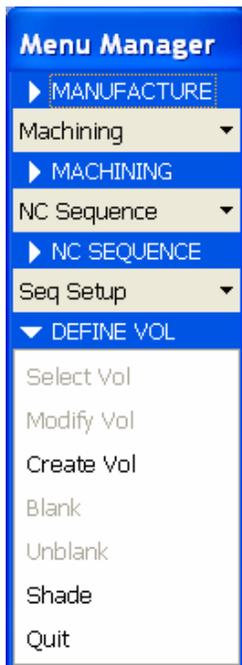


Figure 1.36



Figure 1.37



Figure 1.38



Figure 1.39

- NC Sequence → Define Vol → Create Vol (Figure 1.36) and Enter a name: mv1. We will use the sketch command to create the volume of material to be removed from our workpiece. We will remove all the workpiece that lies outside the part.
- Create Vol → Sketch → Done (Figure 1.37).
- Solid Opts → Extrude → Solid → Done (Figure 1.38).
- Attributes → One side → Done (Figure 1.39).
- Select NC_ASM_FRONT as sketch plane (Figure 1.40). Flip the arrow and select Okay. Select NC_ASM_RIGHT as Sketch View Right (Figure 1.41).

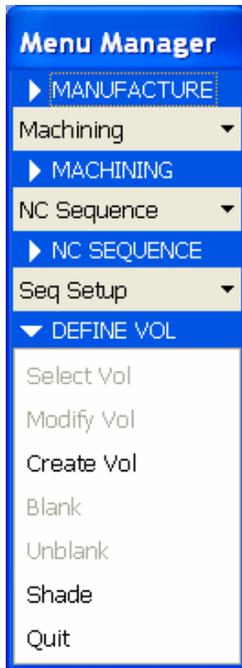


Figure 1.40



Figure 1.41

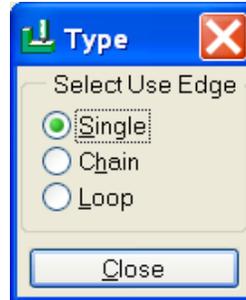


Figure 1.42

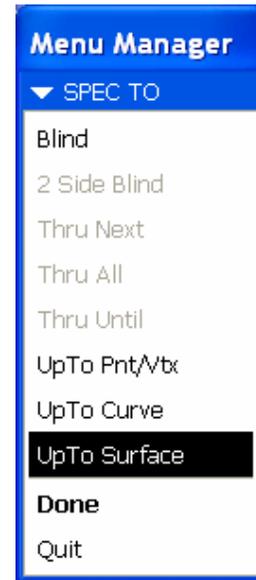


Figure 1.43

- From Pro/E pull down menu, select Sketch → Edge → Use and select all four outer edges of the workpiece (not the part). Click Close (Figure 1.42) and to complete sketch. Up to Surface (Figure 1.43) → Done and pick the top surface of the workpiece. Select OK (Figure 1.44) to complete the protrusion.
- We have sketched our entire workpiece as the mill volume. But we need to leave the material that represents our part. At this point Pro/E provides a Trim function that will “trim” the part from mill volume. Select **Trim** and pick the part we wish to be “trimmed” out of the mill volume. Done/ Return.

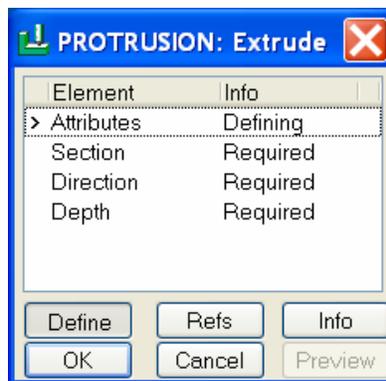


Figure 1.44

E. View tool path simulation and create cutter location file.

- 1) Machining → NC Sequence → NC Check → Run. The simulation of machining process is shown in Figure 1.45.

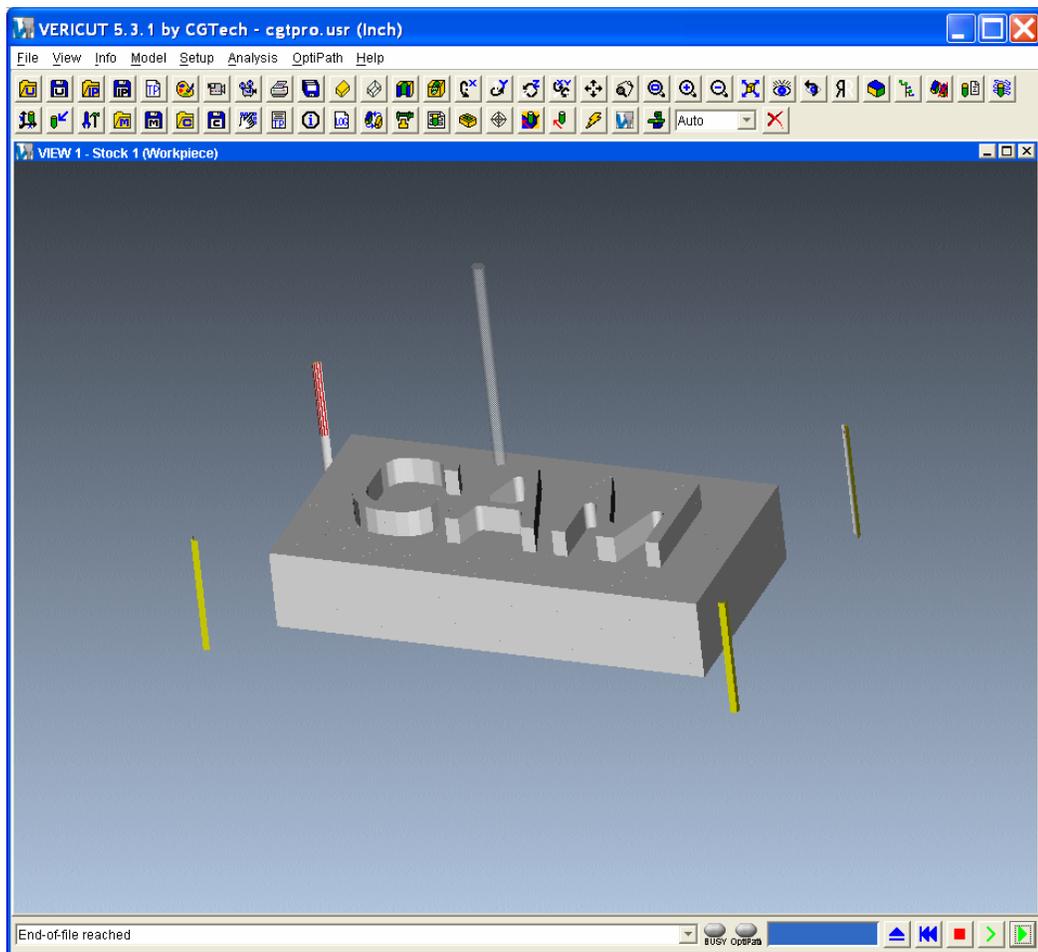


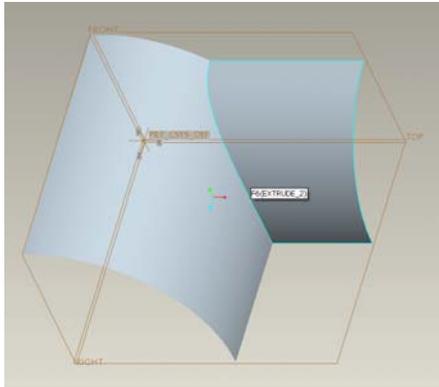
Figure 1.45

- 2) Machining → CL Data → Output → Select Feature → NC Sequence → Volume Milling. PATH → FILE → Done. Enter name: cam in the Save a Copy window. Pro/E will save the file as cam.ncl.1.
- 3) The cutter location file can be converted to G-code file through post processor program written for specify CNC milling machine.

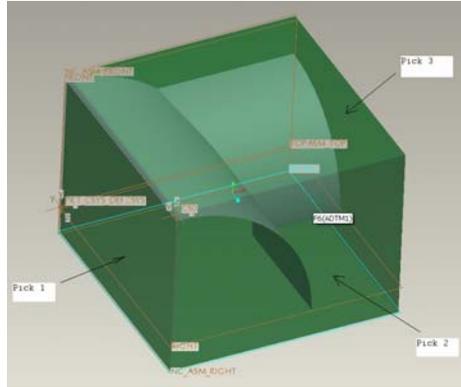
9.3 Advanced Features

Pro/Manufacture supports more advanced functions in automated CNC tool path generation from CAD model; G-code generation; simulation of CNC machining or tool path verification; and surface quality verification for given tool path and machining parameters. These advanced topics are covered in our Computer Aided Manufacture (CAM) course, MECH460. The following figures from the Advanced Pro/E tutorials of the CAM course illustrate these applications.

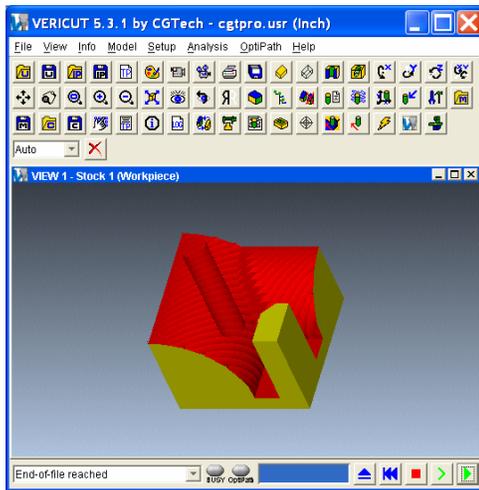
Design Surface



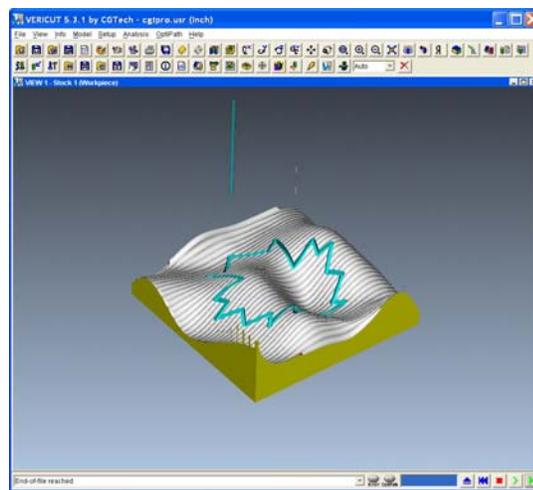
Part and Stock



NC Tool Path Verification



Surface Engraving



10 Definition and Machining of Free-form Surfaces in Pro/ENGINEER

10.1 Introduction

This document explores the advanced functions provided by Pro/Engineer 2004 Wildfire to generate a solid with a free form curved surface boundary. The curved surface is defined using B-splines; the generated solid is later produced on a CNC machine center using 3-axis milling. The machining program is automatically generated using the Pro/Manufacture module integrated with Pro/E. *CNC Tool Path* and *NC-Check* are studied to make necessary adjustment on the machining parameters. The CNC Machine Center in the Advanced Manufacturing Lab in B127, a 4-Axis Victor CNC machine, is used for the machining.

Over the years, Pro/ENGINEER CAD/CAE/CAM system has been steadily improved. Some advanced features have been introduced and improved in the Pro/ENGINEER Wildfire, especially in the area of surface modeling and manipulations. The “variable sweep” function in Pro/ENGINEER Wildfire was recently improved to generate a solid with true sculptured surfaces defined using B-spline curves. The design modeling and automated tool path generation for CNC machining of the designed sculptured surface using Pro/ENGINEER and Pro/MANUFACTURE are illustrated in this overview.

10.2 Variable Sweep Creation of Free Form Surface

The sculptured surface that serves as a boundary surface of the solid part can be defined using trajectories along the u and v directions. The solid is created by extruding this surface along one specified direction.

Step 1. Creation of u Direction Trajectories

One can use any number of trajectories along the u direction the surface. However, one of these boundaries is the *Origin* that must be a straight line defining the general direction of u , as shown in Fig. 1. Here Chains 1, 2 and 3 are three B-spline trajectories.

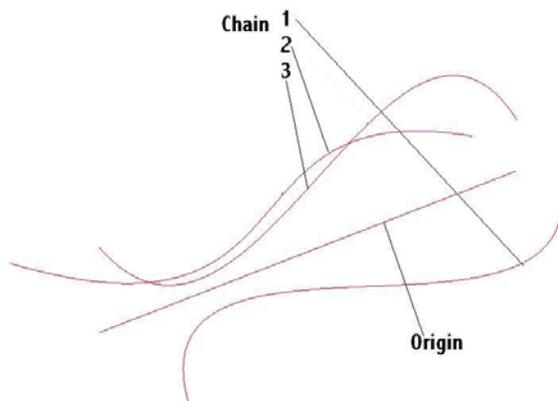


Figure 58 Trajectories in u Direction

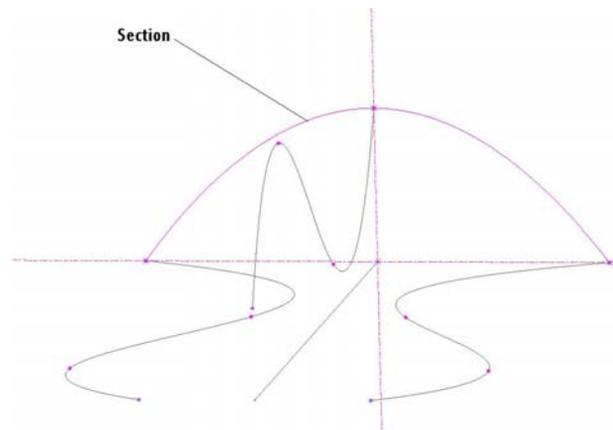


Figure 59 Creation of Section in v Direction

Step 2. Creation of Section

The only curve that defines the v direction of the surface is named as *Section* that is made of a curve on the plane perpendicular to the *Origin*, as shown in Fig. 2. This curve will follow the u direction trajectories to define the sculptured surface. The surface created is a result of this extended “ruled surface”. When only one straight line trajectory is used, the surface will be a ruled surface.

Step3. Surface Generation by Variable Sweep

A preview, as shown in Fig. 3(a), can be carried out before the variable sweep is created. The final sculptured surface produced is shown in Fig. 3(b).

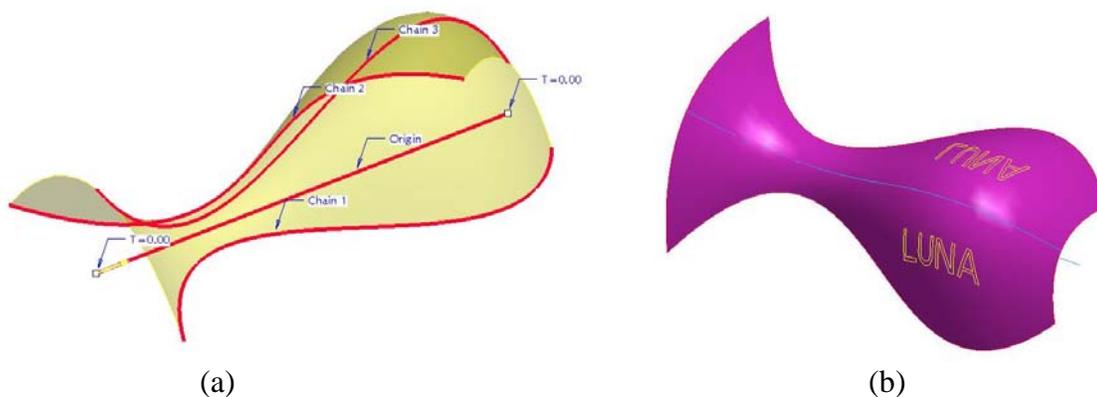


Figure 60 Variable Sweep Surface

10.3 More Complex Surface and Part Model

The “variable sweep” function is an advanced feature of Pro/ENGINEER Wildfire. Previous versions of Pro/ENGINEER only support the modeling of ruled surface using a single trajectory. In Pro/ENGINEER Wildfire, with multiple trajectories, the “variable sweep” can form more complex shapes. It can produce a curved surface with as many curves as one would like to have using different parameter values of u .

The illustration in Fig 4 shows a sculptured surface that is generated using multiple trajectories of different u values ($u = 0, 1/8, 1/4, 3/8, 1/2, 5/8, 6/8, 7/8, 1$). All of these trajectories are B-splines that interpolate given confining points. The *Section* in v direction is also defined using a B-spline curve. Together they define the surface.

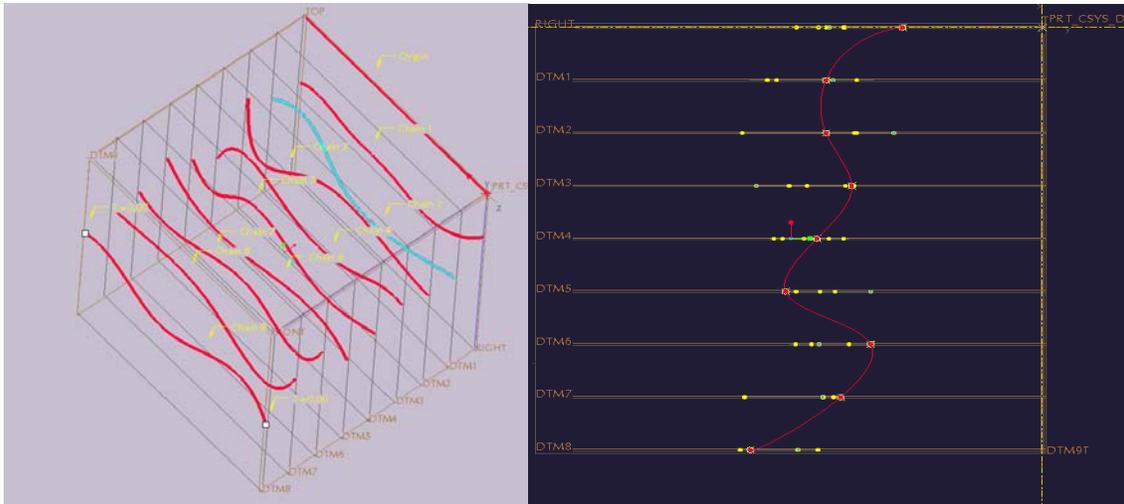


Figure 4

After the free form surface is defined, a solid using the surface as a boundary can be created. The following pictures show the surface and the solid.

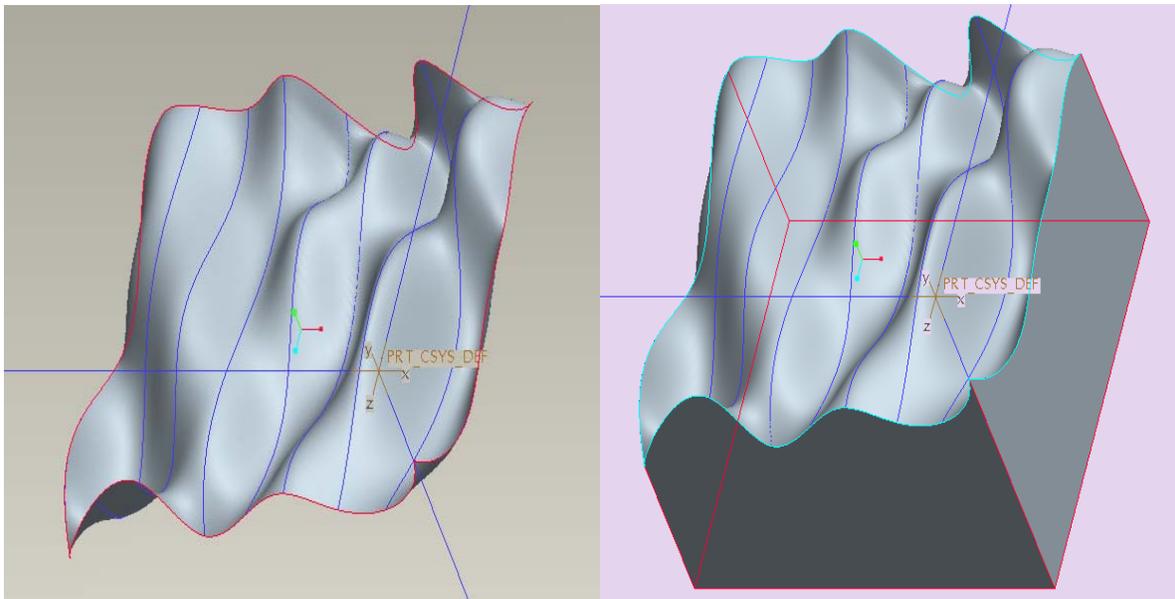


Figure 5

10.4 Automated Generation of CNC Tool Paths Using Pro/Manufacturing

Pro/ENGINEER Wildfire has advanced features in the manipulating curve surface and solid. The Pro/MANUFACTURE can be used to generate CNC tool paths for machining the surface and to perform NC-Check to verify the machined surface. Figures 6 and 7 illustrate the results from Tool Path and NC-Check.

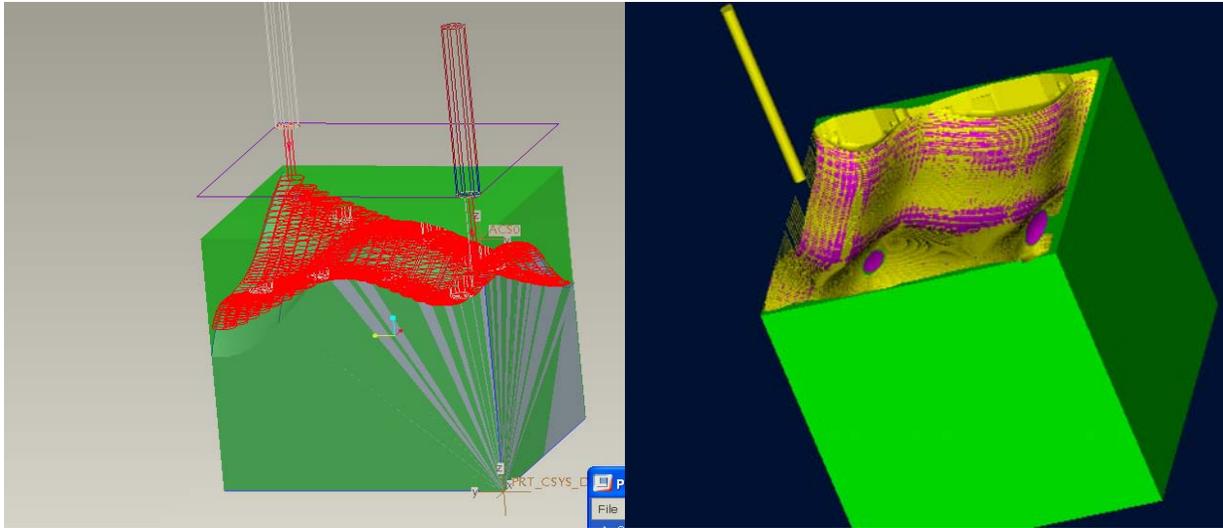
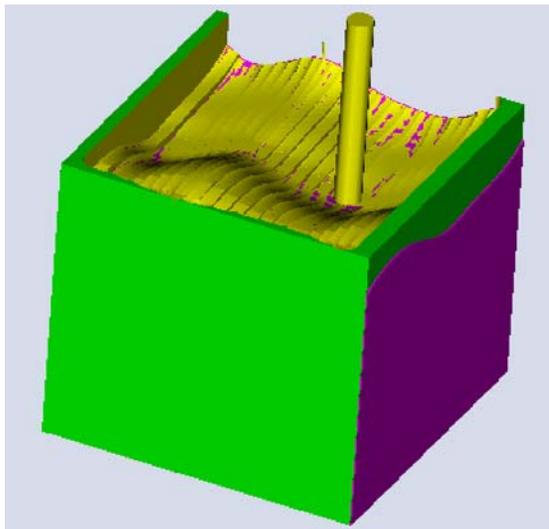
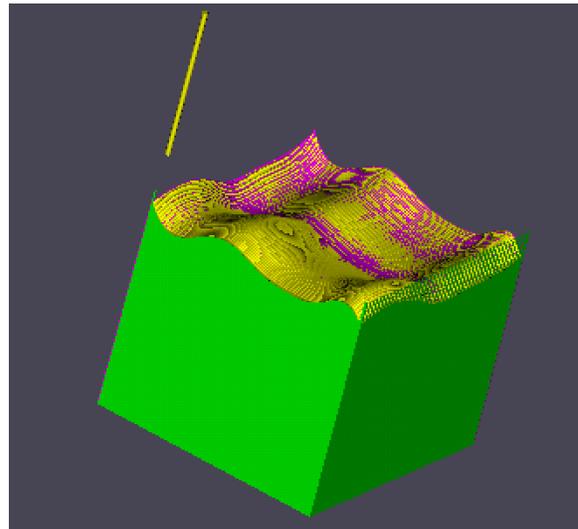


Figure 6



cutter dia = 0.5 step_over = .25



cutter dia = 0.125 step_over = .0625

Figure 7

10.5 Automated Generation of CNC Machining Program (G-Code)

Pro/E Wildfire supports the Post Process operation to automatically generate CNC machine program, G-Code, according to calculated CL file (cutter location). As different CNC machines is controlled by controllers, and different controllers operate on different versions of operating programs, machine programs, G-Code, are different from one machine to another. The CNC machine center in our lab, Victor, operates on a FANUC controller. There exist several different types of FANUC controllers. As a result, there is no ready Post Processor in Pro/E Wildfire that can support our Victor machine center operation. A C program that supports the Post Process for our particular machine is used to get the G-codes.

11 Programming in Pro/ENGINEER

11.1 Introduction

Pro/ENGINEER supports interactive graphical programming at two different levels. At the higher level, C++ programs are supported through Pro/ENGINEER API Toolkit. At lower level, a micro programming environment, Pro/E PROGRAM Tool, is supported. These programming environments serve different needs. In this document, we will discuss the background of the Pro/E API Toolkit, while focus on the Pro/E Program Tool due to its ease of use.

1) Pro/ENGINEER API Toolkit

Pro/ENGINEER API Toolkit allows customers to extend, automate, and customize a wide range of Pro/ENGINEER design-through-manufacturing functionality.

Pro/ENGINEER API Toolkit consists of a library of functions:

- an application-programming interface (API), written in the C programming language. These functions are typically used by MIS organizations to create applications that run in parallel with Pro/ENGINEER and to integrate product information with the customer's corporate MRP/ERP systems.
- applications used extensively by companies participating in PTC's Cooperative Software Partner (CSP) program to interface their commercial information management products with Pro/INTRALINK. Normally, participation of a three day tutorial on the API Toolkit is needed to get the API Toolkit function module.

The extensive Pro/ENGINEER API Toolkit provides programmatic access for creating, interrogating, and manipulating almost every aspect of the engineering model and its data management.

Typical toolkit applications include:

- automating the creation of complex features
- automating the production of Pro/ENGINEER deliverables, such as BOMs, drawings, and manufacturing operations
- improving product quality by performing design rule verification based on inputs from an external, knowledge-based system.

More specifically, this Pro/ENGINEER API Toolkit functionality allows:

- Customization of the Pro/ENGINEER-Foundation menu system
- Datum, solid, and manufacturing feature creation
- Assemblies
- Drawing automation
- Access to model geometry

The Pro/ENGINEER API Toolkit provides complete access to the information within the Pro/INTRALINK environment, allowing customers to further leverage the product information contained within Pro/INTRALINK.

Specifically, this functionality allows:

- Integration with MRP/ERP Systems
- Custom client applications, such as Web integrated clients
- Triggered verification, notification and enforcement of business process actions

Product Capabilities:

- Create automated, single-use or derived designs by geometric and parametric constraints
- Extend the Pro/ENGINEER user interface with custom processes seamlessly embedded in the interface
- Customize the Pro/ENGINEER menu system
- Collaborate between Pro/ENGINEER applications
- Access peer-to-peer communications for better application diagnoses

Customer Benefits:

- Integrate expert systems and knowledge-based applications into the Pro/ENGINEER environment.

Improve product quality with design rule verification based on inputs from an external, knowledge-based systems.

2) Pro/E Program Tool

The Pro/E Program environment, on the other hand, support quick and relatively straightforward interactive graphical programming in Pro/E for every users.

The programming environment is simply Pro/E and *Microsoft Notepad* or *Word*.

One can enter the Pro/E PROGRAM environment, by clicking **Tools > Program...** from the pull-down menu in the Pro/E **PART** or **ASSEMBLY** mode.

To show or edit the program, one can click **Show Design** or **Edit Design** from the **PROGRAM** menu.

A typical Pro/E PROGRAM routine may contain any of the following:

- Input variables
- Relations
- IF-ELSE clauses
- Lists of all the features, and parts
- INTERACT statements
- MASSPROP statement

After the Pro/E PROGRAM routine is edited, the user will be asked whether the changes are to be incorporated (in the message window at the bottom). To proceed, enter Y. If N entered, the program will not be executed and changes will be lost.

11.2 Programming Details

The ingredients of a typical program include:

1) Input Variables

INPUT variables may be specified at the beginning of the listing; and their values can be provided by the user at the program beginning.

The format of INPUT is as follows:

```
INPUT
Variable_Name Variable_Type
END INPUT
```

The INPUT statement must define the name and type of the variable. Variable names must always begin with a character. The following variable types are supported:

Number

String: This enables the user to enter parameters or model names.

Logical (YES_NO): Enter either Y or N.

An example:

```
INPUT
THICKNESS NUMBER
"Enter wall thickness for the cylinder"
END INPUT
```

2) Relations

All valid relations in a Pro/ENGINEER model can be entered in a Pro/PROGRAM.

An example:

```
d0 = d6 * 2
```

Here, d0 and d6 are dimension ID name.

3) IF-ELSE Clauses

Conditional statements, i.e. IF _ ELSE, can be used to create a program branch.

For example:

```
ADD PROTRUSION.....
IF d1 > d2
ADD HOLE ...
END ADD
ENDIF
ADD CUT.....
END ADD
```

So, when d1 is smaller than d2, a CUT is added, instead of a HOLE.

4) Lists of Features and Parts

The program that Pro/E PROGRAM brings up simply includes all feature building commands used in creating the model and the properties of these features. All features and parts are listed in the program. For instance, the ADD feature by EXTRUSION operation is recorded as:

```
ADD FEATURE (initial number 8)
INTERNAL FEATURE ID 106
PARENTS = 100(#7)
PROTRUSION: Extrude
```

NO.	ELEMENT NAME	INFO
---	-----	-----
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Reference Sketch	F7(SKETCH_2)
5	Feature Form	Solid
6	Direction	Side 2
7	Depth	Defined
7.1	Side One	Defined
7.1.1	Side One Depth	None
7.2	Side Two	Defined
7.2.1	Side Two Depth	Variable
7.2.2	Value	70.00

```
SECTION NAME = Sketch 2
```

```
FEATURE'S DIMENSIONS:
```

```
d11 = 70.00
```

```
END ADD
```

Additional operations can be added, and this ADD operation can be changed.

5) INTERACT

INTERACT statements provide a placeholder for creating interactive part. They can be inserted anywhere within the FEATURE ADD - END ADD.

Here is an example,

```
ADD PROTRUSION.....  
IF d1 > d2  
ADD HOLE.....  
ELSE  
INTERACT  
END IF  
ADD CUT.....
```

In this example, an alternate set of features will be created if d1 is not greater than d2. The ADD CUT command has to be input by the user.

6) MASSPROP

The MASSPROP statement is used to update mass properties each time geometry changes. Format is as follows:

```
MASSPROP  
END MASSPROP
```

7) Other Operations for Feature Editing

a. Changing feature dimension

The dimensions of features in the program can be updated by a DIMENSION statement with:

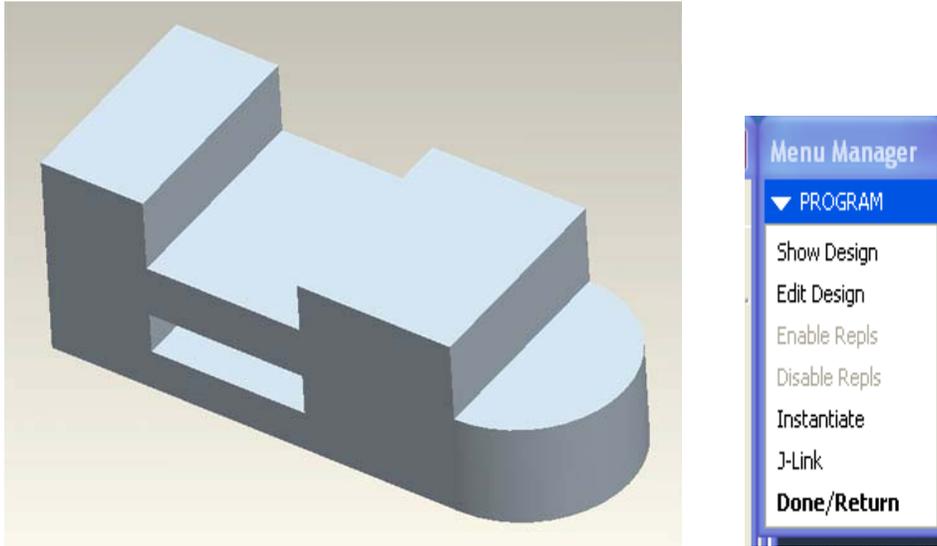
```
MODIFY d# = value
```

b. Editing Errors

Common editing errors include:

- Having an IF statement without an END IF statement or vice versa
- Typing a variable name incorrectly in a relation or a condition
- Reordering a child before the parent
- Deleting a parent feature

11.3 An Example Part and the PROGRAM Window



The Pro/Program for this Part Model

- Start Pro/E
- Open the Part Model file: **part5.prt**
- Use Pull Down Menu **T**ool > **P**rogram...
- In the **PROGRAM** Window
 - **Show Design** and **Edit Design** options will display the Pro/Program that is used to create the displayed part model.
 - **Edit Design** option allows you make changes to the model through “Programming Logic” rather than through “drawing and modeling”. Automated tasks can be achieved. If you exit from the Edit window and answer “Yes” in the message window at the bottom of the screen to the prompt: “Do you want to incorporate your changes into the model?” The programmed change will be added to the existing model. You can start from a simple template model to write various programs.
 - The **J-Link** function allows you to load in Java codes.

The List of the Pro/E PROGRAM for this Part Model

VERSION 2.0
 REVNUM 365
 LISTING FOR PART LESSON5

INPUT
 END INPUT

RELATIONS
 END RELATIONS

ADD FEATURE (initial number 1)
 INTERNAL FEATURE ID 1

DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	X Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = RIGHT

END ADD

ADD FEATURE (initial number 2)
 INTERNAL FEATURE ID 3

DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Y Axis
3	Flip Datum Dir	Defined
4	Fit	Defined
4.1	Fit Type	Default

NAME = TOP

END ADD

ADD FEATURE (initial number 3)
 INTERNAL FEATURE ID 5

DATUM PLANE

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Constraints	Defined
2.1	Constraint #1	Defined
2.1.1	Constr Type	Z Axis
3	Flip Datum Dir	Defined

4	Fit	Defined
4.1	Fit Type	Default

NAME = FRONT

END ADD

ADD FEATURE (initial number 4)
 INTERNAL FEATURE ID 7
 PARENTS = 1(#1) 3(#2) 5(#3)

PROTRUSION: Extrude

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	FRONT:F3(DATUM PLANE)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Top
4.1.4	Reference	TOP:F2(DATUM PLANE)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Direction	Side 2
7	Depth	Defined
7.1	Side One	Defined
7.1.1	Side One Depth	None
7.2	Side Two	Defined
7.2.1	Side Two Depth	Variable
7.2.2	Value	10.00

NAME = BLOCK
 SECTION NAME = S2D0001

FEATURE'S DIMENSIONS:
 d2 = 20.00
 d3 = 10.00
 d4 = 10.00
 END ADD

ADD FEATURE (initial number 5)
 INTERNAL FEATURE ID 28
 PARENTS = 7(#4)

PROTRUSION: Extrude

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Add
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	Surf:F4(PROTRUSION)
4.1.2	View Direction	Defined
4.1.3	Orientation	Right
4.1.4	Reference	Surf:F4(PROTRUSION)

4.2 Sketch Defined
 5 Feature Form Solid
 6 Material Side Side Two
 7 Direction Side 2
 8 Depth Defined
 8.1 Side One Defined
 8.1.1 Side One Depth None
 8.2 Side Two Defined
 8.2.1 Side Two Depth Variable
 8.2.2 Value 5.00

NAME = ROUND_END
 SECTION NAME = S2D0002
 OPEN SECTION

FEATURE'S DIMENSIONS:
 d9 = 5.00
 END ADD

ADD FEATURE (initial number 6)
 INTERNAL FEATURE ID 52
 PARENTS = 7(#4) 28(#5)

CUT: Extrude

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	Surf:F4(PROTRUSION)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Top
4.1.4	Reference	Surf:F5(PROTRUSION)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Direction	Side 1
8	Depth	Defined
8.1	Side One	Defined
8.1.1	Side One Depth	None
8.2	Side Two	Defined
8.2.1	Side Two Depth	Variable
8.2.2	Value	10.00

NAME = TOP_CUT

SECTION NAME = S2D0003
 OPEN SECTION

FEATURE'S DIMENSIONS:
 d12 = 5.00
 d13 = 7.50
 d14 = 3.50
 d15 = 2.40
 d16 = 10.00
 END ADD

ADD FEATURE (initial number 7)
 INTERNAL FEATURE ID 149
 PARENTS = 3(#2) 52(#6) 7(#4)

CUT: Extrude

NO.	ELEMENT NAME	INFO
1	Feature Name	Defined
2	Extrude Feat type	Solid
3	Material	Remove
4	Section	Defined
4.1	Setup Plane	Defined
4.1.1	Sketching Plane	Surf:F4(PROTRUSION)
4.1.2	View Direction	Side 1
4.1.3	Orientation	Top
4.1.4	Reference	Surf:F6(CUT)
4.2	Sketch	Defined
5	Feature Form	Solid
6	Material Side	Side Two
7	Direction	Side 1
8	Depth	Defined
8.1	Side One	Defined
8.1.1	Side One Depth	None
8.2	Side Two	Defined
8.2.1	Side Two Depth	Thru All

NAME = INSIDE_CUT
 SECTION NAME = S2D0001

FEATURE'S DIMENSIONS:
 d24 = 2.00
 d25 = 2.00
 END ADD

MASSPROP
 END MASSPRO

References

1. Roger Toogood, *Pro/Engineer Wildfire Tutorial and Multimedia CD*, Schroff Development Corporation, 2003.
2. Kurowski, P. M., "When Good Engineers Deliver Bad FEA," *Machine Design*, November 9, 1995, pp. 61-66.
3. Kurowski, P. M., "Avoiding Pitfalls in FEA," *Machine Design*, November 7, 1994, pp. 78-86.
4. Toogood, R., *Pro/MECHANICA Structure Tutorial*, SDC Publications, 2004.
5. Tutorials from Parametric Technology Ltd., <http://ptc-mss.com/Tutorial/tutorial.htm>

Appendix: Format of Reports

A1. Format of the Laboratory Report

Title of the Assignment
Names and Student Numbers

1. Objective
2. Description of the Assignment
3. Your Experience and Suggestions
4. Illustrations (Images and Drawings from Pro/E)
5. New Procedures Developed (if there is any)

Email the following documents to: mech410@me.uvic.ca

- Lab report in *MS Word* named as: `LastName1_LastName2 (.doc)`
- The Pro/E Model File with the same name as above (different extension name).

A2. Format of the Project Report

Title of the Project
Names and Student Numbers

Abstract (50 – 100 words)

Table of Contents

1. Introduction (Description of the Project, Problem Definition, Theory or Algorithm)
2. Implementations
3. Technical Challenges
4. Special Features and Highlights
5. Summary (Experience and Suggestions)

References

Appendix

- A. Important figures, drawings, calculations, etc.
- B. Electronic copy of all related and necessary Pro/E files and other source codes.

Email the following documents to: mech410@me.uvic.ca

- A Microsoft PowerPoint Presentation (4-6 slides)
- Project report in *MS Word* named as: `LastName1_LastName2 (.doc)`
- The Pro/E model files with the same name as above (different extension name).