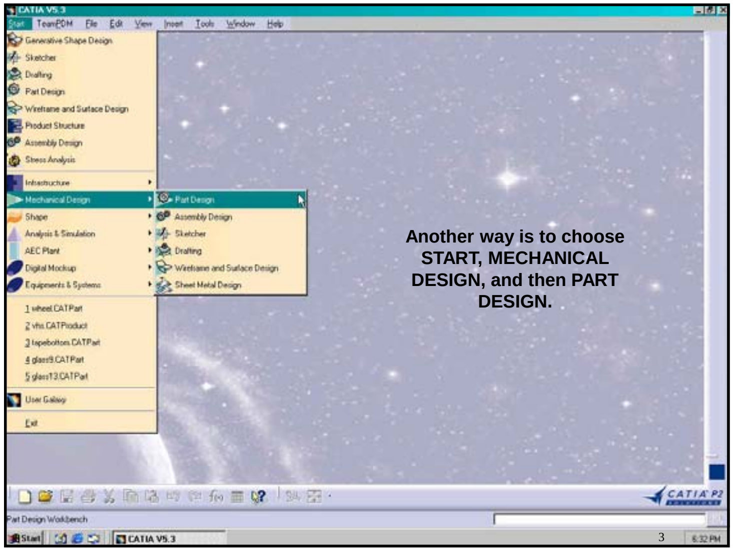
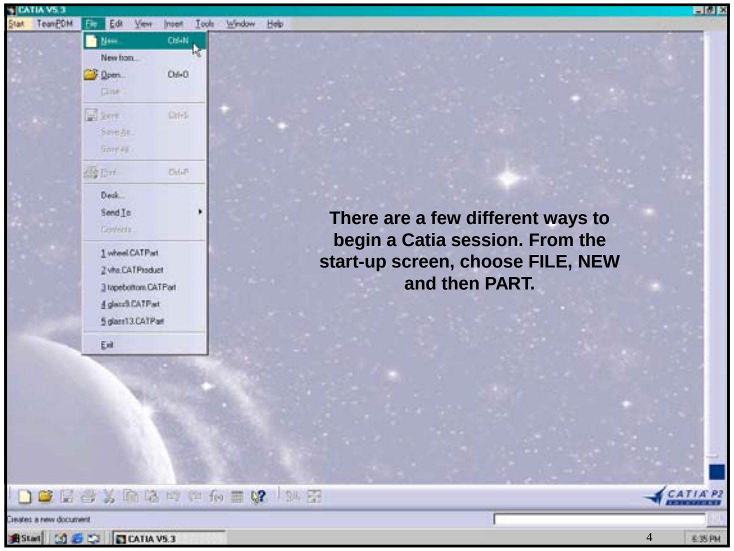
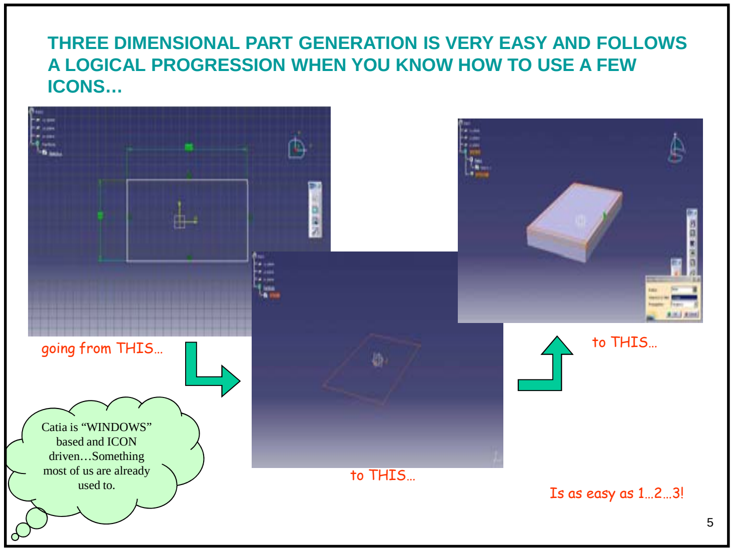
Lab-01
Chapter-01

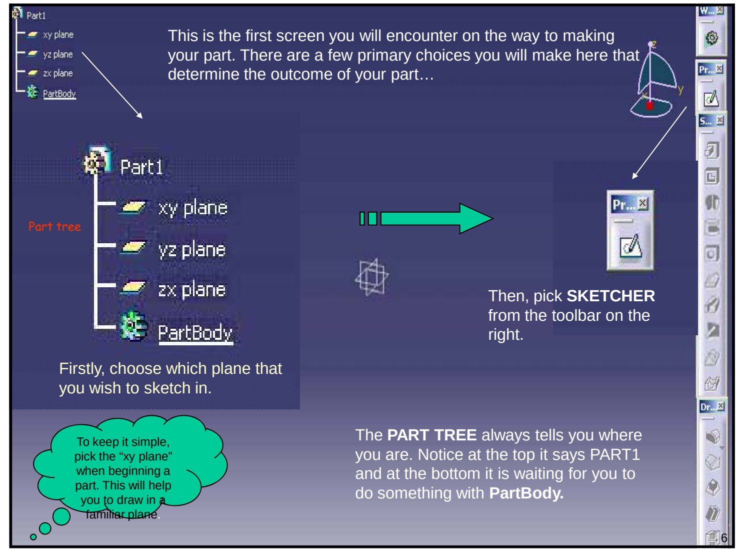
YOUR FIRST PART – START TO FINISH

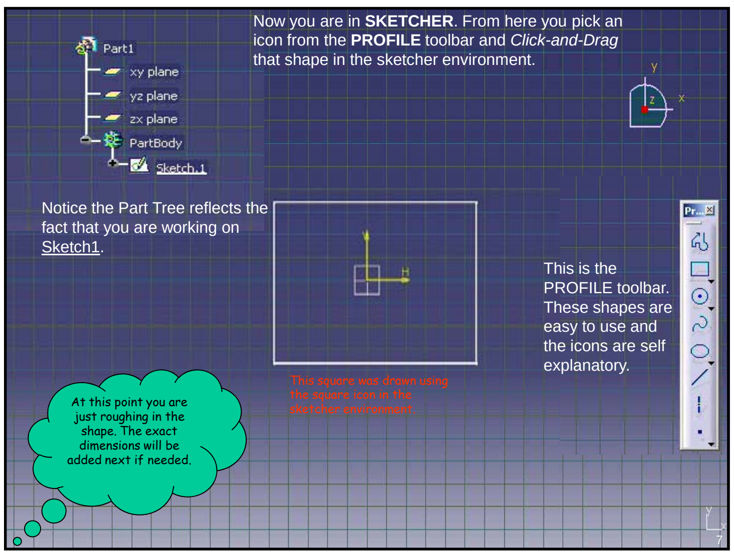
2

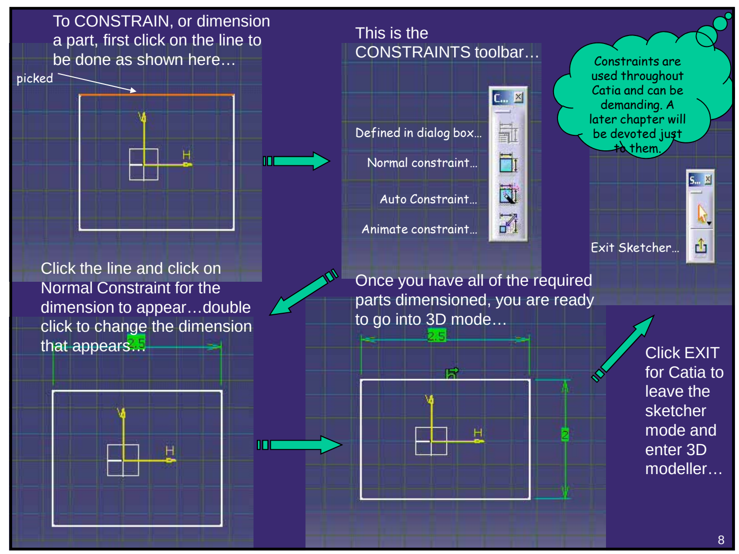




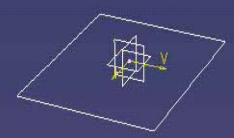

THREE DIMENSIONAL PART GENERATION IS VERY EASY AND FOLLOWS A LOGICAL PROGRESSION WHEN YOU KNOW HOW TO USE A FEW ICONS…
going from THIS…
to THIS…
to THIS…
Catia is "WINDOWS" based and ICON driven…Something most of us are already used to.
Is as easy as 1…2…3!
5


This is the first screen you will encounter on the way to making your part. There are a few primary choices you will make here that determine the outcome of your part…
Part1
xy plane
yz plane
zx plane
PartBody
Part tree
Firstly, choose which plane that you wish to sketch in.
Then, pick SKETCHER from the toolbar on the right.
To keep it simple, pick the "xy plane" when beginning a part. This will help you to draw in a familiar plane
The PART TREE always tells you where you are. Notice at the top it says PART1 and at the bottom it is waiting for you to do something with PartBody.
6


Now you are in SKETCHER. From here you pick an icon from the PROFILE toolbar and Click-and-Drag that shape in the sketcher environment.
Part1
xy plane
yz plane
zx plane
PartBody
Sketch.1
Notice the Part Tree reflects the fact that you are working on Sketch1.
This is the PROFILE toolbar. These shapes are easy to use and the icons are self explanatory.
This square was drawn using the square icon in the sketcher environment
At this point you are just roughing in the shape. The exact dimensions will be added next if needed.
7


To CONSTRAIN, or dimension a part, first click on the line to be done as shown here…
picked
This is the CONSTRAINTS toolbar…
Constraints are used throughout Catia and can be demanding. A later chapter will be devoted just to them.
Defined in dialog box…
Normal constraint…
Auto Constraint…
Animate constraint…
Exit Sketcher…
Click the line and click on Normal Constraint for the dimension to appear…double click to change the dimension that appears…
Once you have all of the required parts dimensioned, you are ready to go into 3D mode…
Click EXIT for Catia to leave the sketcher mode and enter 3D modeller…
8

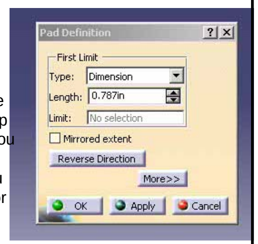
Once you enter the 3D environment, the part profile you were working on takes on an isometric orientation as seen here.



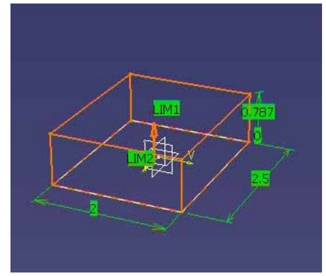
This is the SKETCHER toolbar from which you can choose a process of building your 3D model…here we have chosen PAD

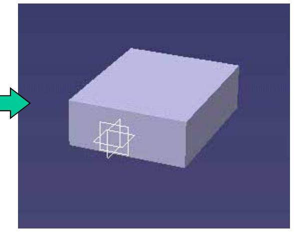
Notice that not all of the icons shown on the shortened SKETCHER toolbar are active. This is because some other variable must be satisfied for Catia to allow it's use. We will cover all of the icons and their uses later.

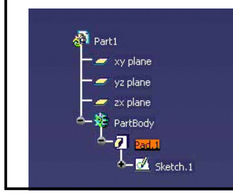
Once PAD is chosen, the PAD DEFINITION pop-up will appear. From here you define the TYPE and LENGTH of the pad. You can also choose to mirror the pad or reverse it's direction from here.



The pad definition box will cause the limits that have been selected to be applied in a wire frame representation first. Now if you click on APPLY and then OK, your wire frame will finally become a solid 3D model.







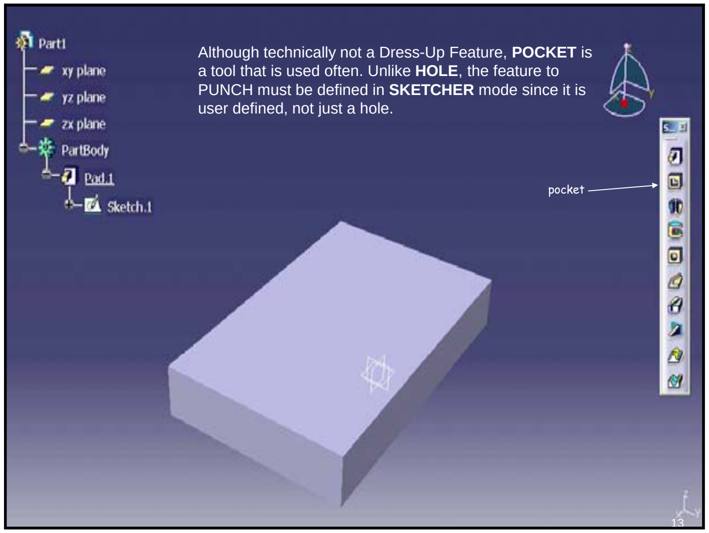
Notice that on your PARTS TREE, pad1 has been added BEFORE sketch1 that was already there. This is all part of Catia's hierarchy system.

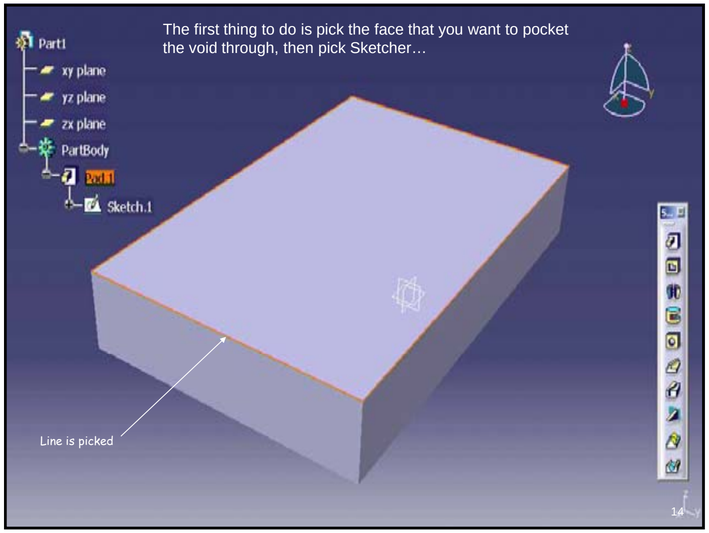
Congratulations! Your FIRST 3D part!

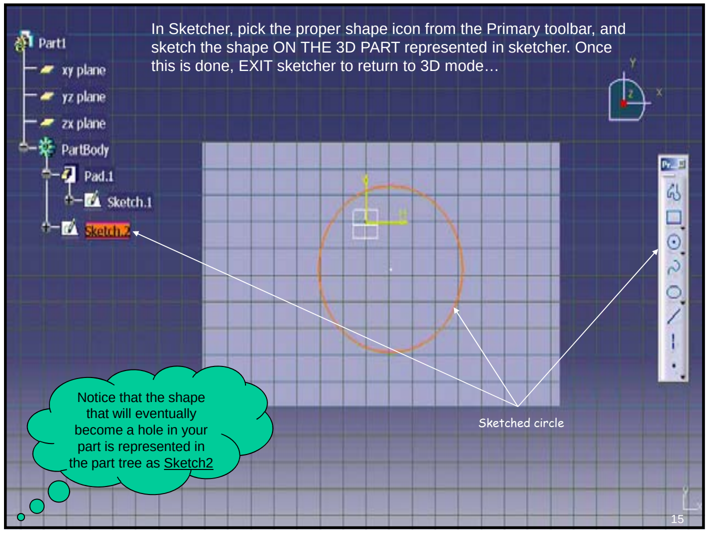
Lab-02
Chapter -02

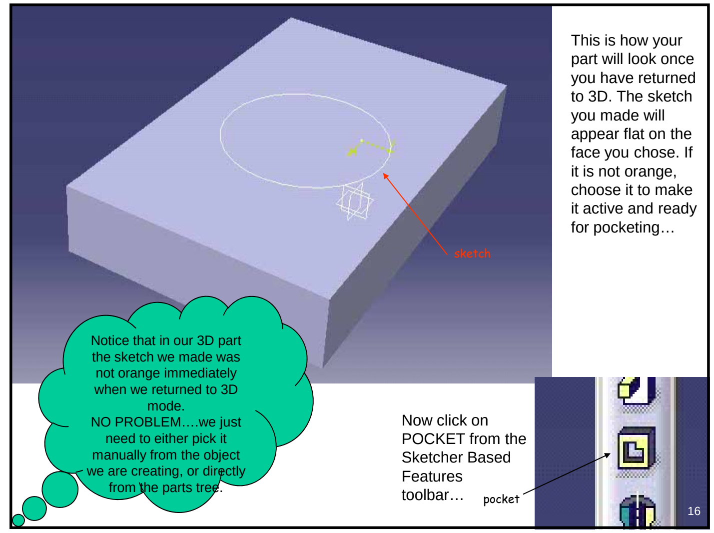
BASIC DRESS-UP FEATURES

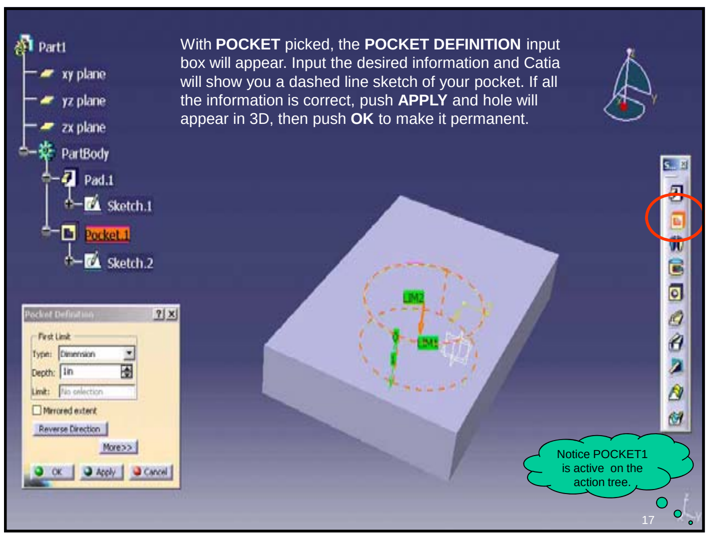
11

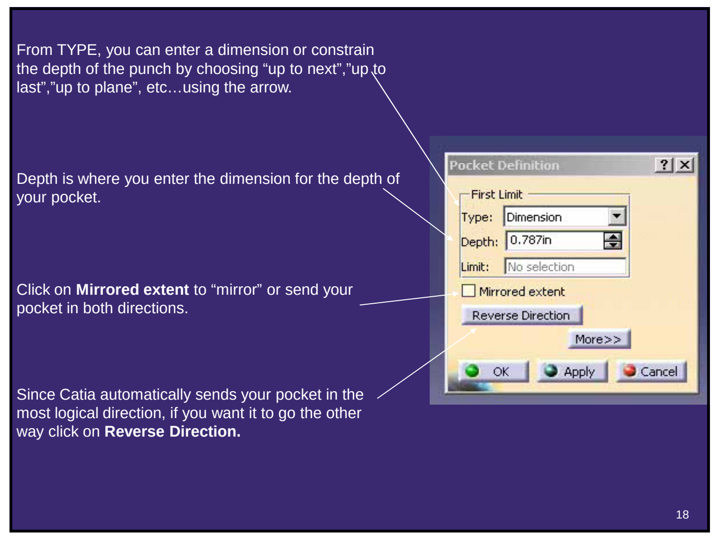





In Sketcher, pick the proper shape icon from the Primary toolbar, and sketch the shape ON THE 3D PART represented in sketcher. Once this is done, EXIT sketcher to return to 3D mode…
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Sketch.2
Notice that the shape that will eventually become a hole in your part is represented in the part tree as Sketch2
Sketched circle
15

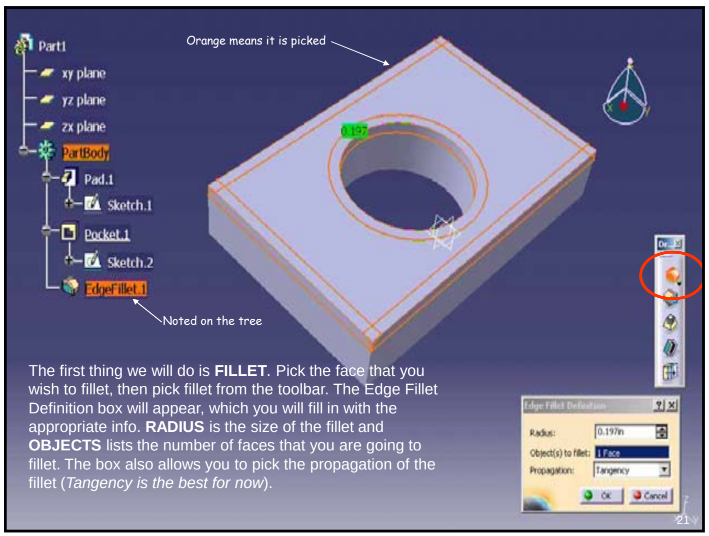

This is how your part will look once you have returned to 3D. The sketch you made will appear flat on the face you chose. If it is not orange, choose it to make it active and ready for pocketing…
sketch
Notice that in our 3D part the sketch we made was not orange immediately when we returned to 3D mode.
NO PROBLEM….we just need to either pick it manually from the object we are creating, or directly from the parts tree.
Now click on POCKET from the Sketcher Based Features toolbar…
pocket
16

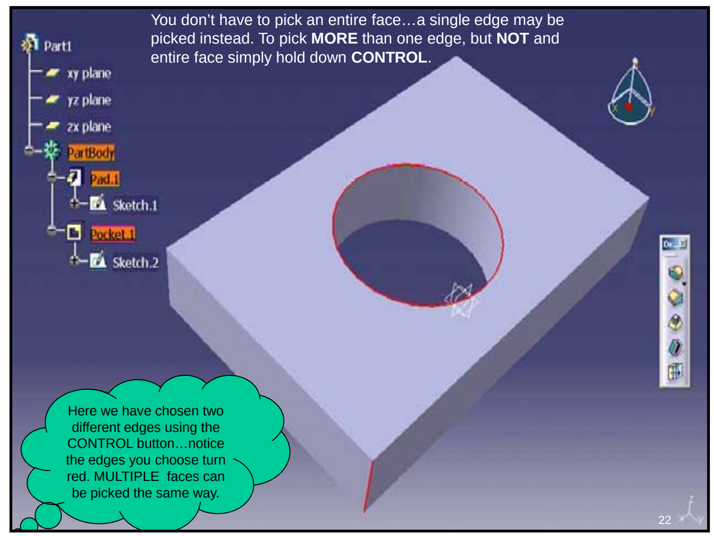

With POCKET picked, the POCKET DEFINITION input box will appear. Input the desired information and Catia will show you a dashed line sketch of your pocket. If all the information is correct, push APPLY and hole will appear in 3D, then push OK to make it permanent.
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pocket.1
Sketch.2
Notice POCKET1 is active on the action tree.
17


From TYPE, you can enter a dimension or constrain the depth of the punch by choosing “up to next”,”up to last”,”up to plane”, etc…using the arrow.
Depth is where you enter the dimension for the depth of your pocket.
Click on Mirrored extent to “mirror” or send your pocket in both directions.
Since Catia automatically sends your pocket in the most logical direction, if you want it to go the other way click on Reverse Direction.
Pocket Definition
First Limit
Type: Dimension
Depth: 0.787in
Limit: No selection
Mirrored extent
Reverse Direction
More>>
OK
Apply
Cancel
18


Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pocket.1
Sketch.2
Here is your part with your pocket in it.
Now lets begin to **DRESS IT UP…**
Dress Up Features toolbar
19


A close up look at your **DRESS UP TOOLBAR** shows that it is ready to do five basic functions for you…
Fillet
Chamfer
Draft
Shell
Mirror
DRAFT is what you use to angle the sides of your part in preparation for possible casting processes.
20

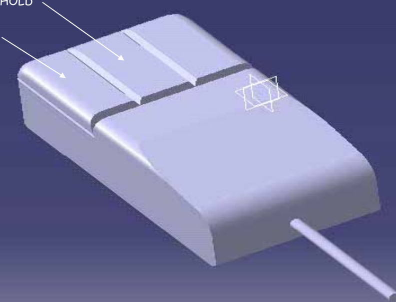

Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pocket.1
Sketch.2
EdgeFillet.1
Orange means it is picked
Noted on the tree
The first thing we will do is **FILLET**. Pick the face that you wish to fillet, then pick fillet from the toolbar. The Edge Fillet Definition box will appear, which you will fill in with the appropriate info. **RADIUS** is the size of the fillet and **OBJECTS** lists the number of faces that you are going to fillet. The box also allows you to pick the propagation of the fillet (*Tangency is the best for now*).
Edge Fillet Definition
Radius:
Object(s) to fillet:
Propagation:
Tangency
OK
Cancel
21


You don't have to pick an entire face…a single edge may be picked instead. To pick **MORE** than one edge, but **NOT** and entire face simply hold down **CONTROL**.
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pocket.1
Sketch.2
Here we have chosen two different edges using the CONTROL button…notice the edges you choose turn red. MULTIPLE faces can be picked the same way.
22

Lab-03 Chapter-03

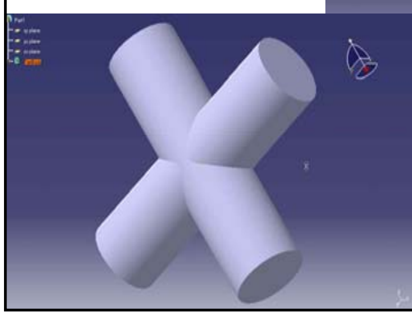
MOUSE MANIPULATION

The 3 button mouse is your tool for manipulation of the parts and assemblies that you have created. With it you can ZOOM, ROTATE and PAN your parts or manipulate the specification tree. First, place your cursor ANYWHERE on the screen…



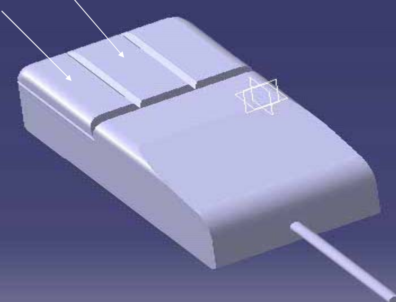
To ZOOM, click and hold the MIDDLE mouse button, click and release the LEFT mouse button, then PUSH the mouse away from you to make your part smaller and PULL it towards you to enlarge your part.

ZOOM in from a small part…

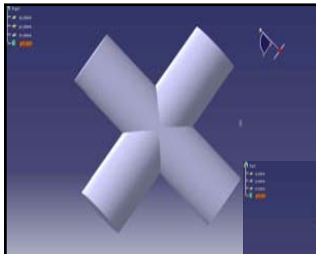


To a LARGE part with this simple technique.

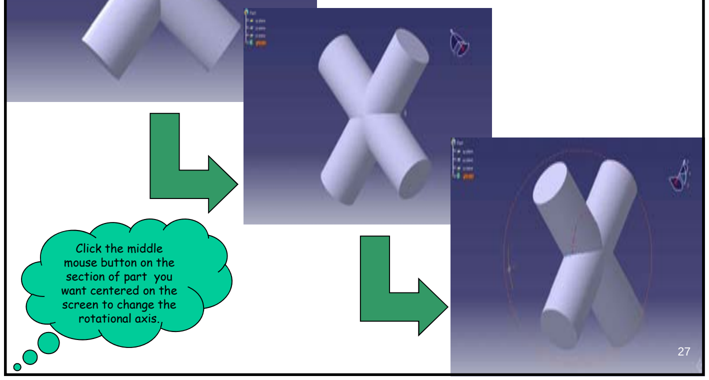
The next thing you can do is ROTATE your part…



This is accomplished by HOLDING the MIDDLE mouse button and then the LEFT mouse button while keeping the middle one depressed.

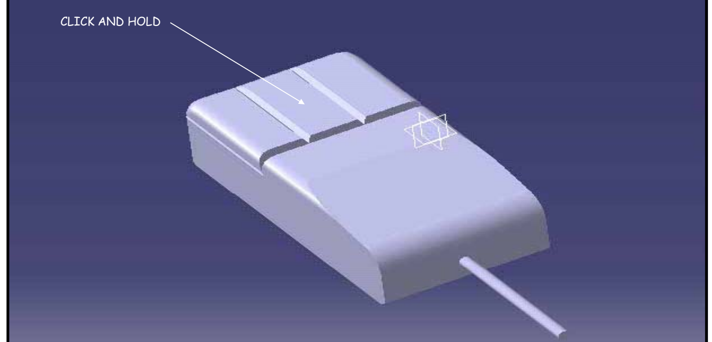


You can **ROTATE** your part in any orientation, 360 degrees in 3D.



27

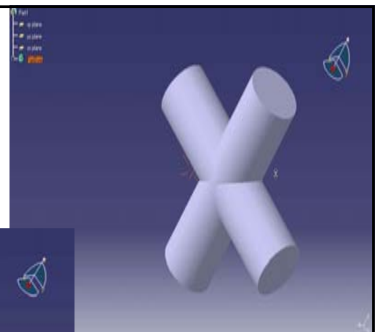
Another useful mouse aided motion is PANNING…

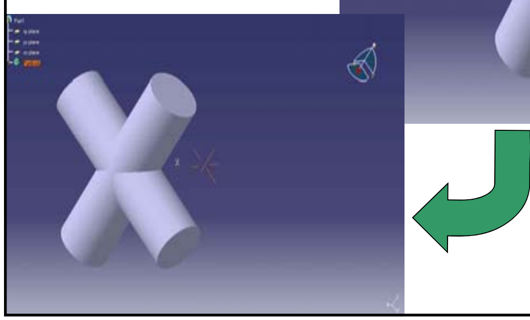


To PAN a part across the screen, click and hold the MIDDLE mouse button only. This allows you to move your part around the screen in a single orientation.

28

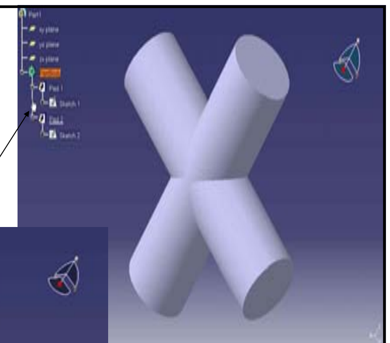
Your part will remain the same size and in the same orientation, but you can move it around in relation to the screen itself.

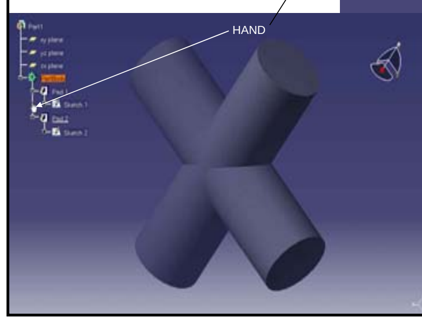




29

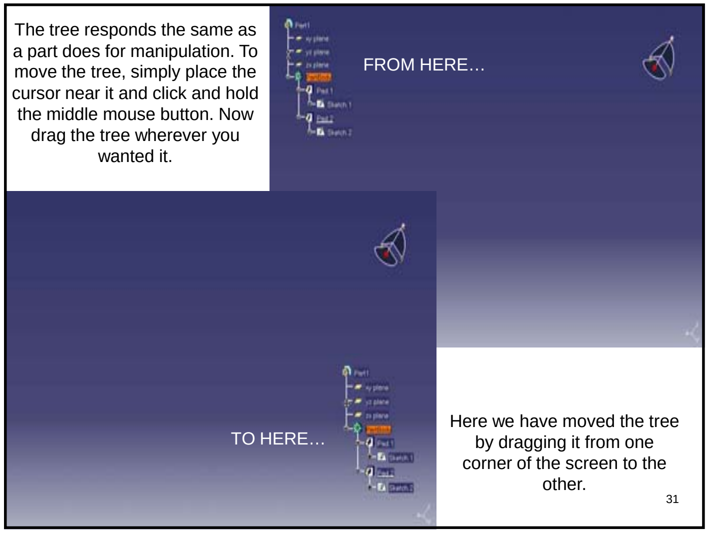
The SPECIFICATION TREE can also be manipulated. You can EXPAND and SHRINK, MOVE and ZOOM it.

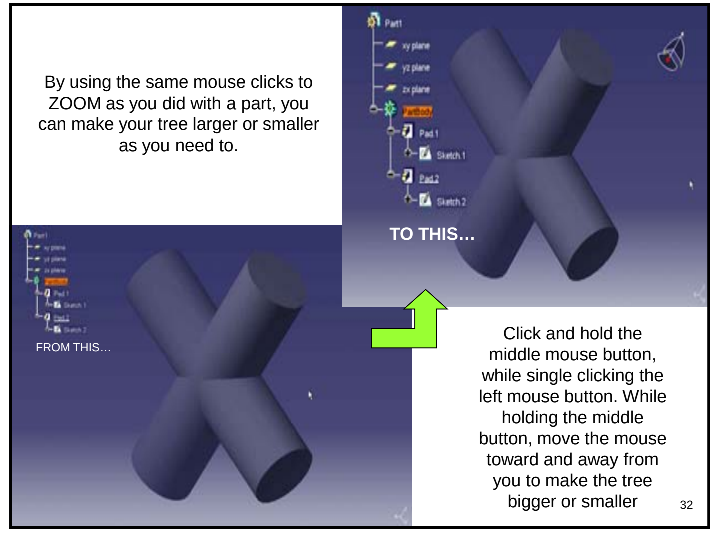


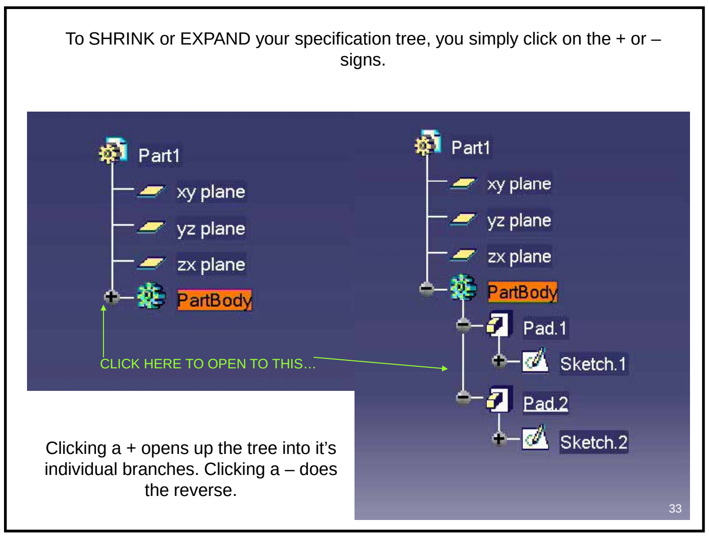


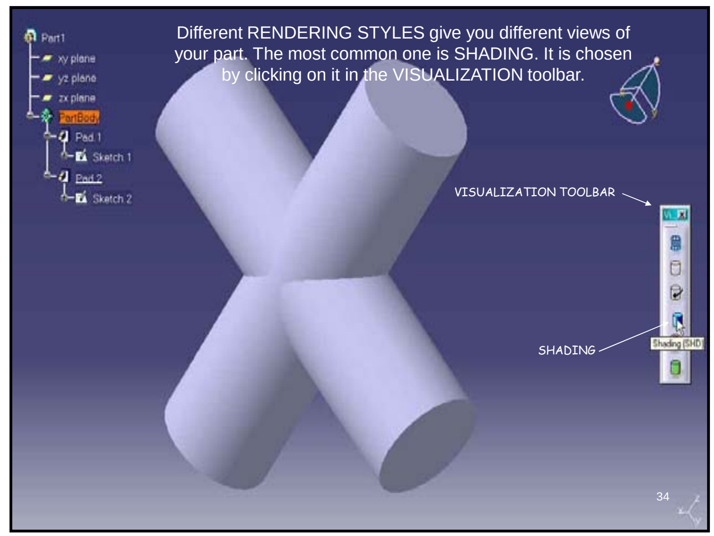
Note the HAND cursor on the tree. To manipulate the tree, you must first click on it. This will turn your part DARK, signaling you are ready to perform various tree functions.

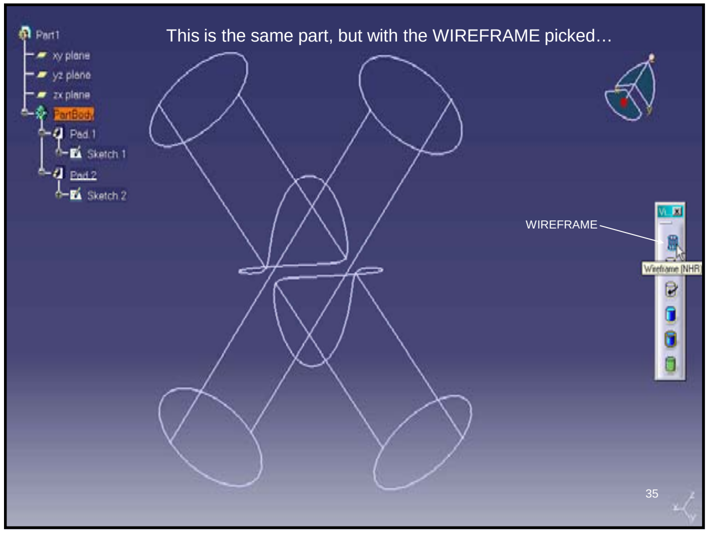
30

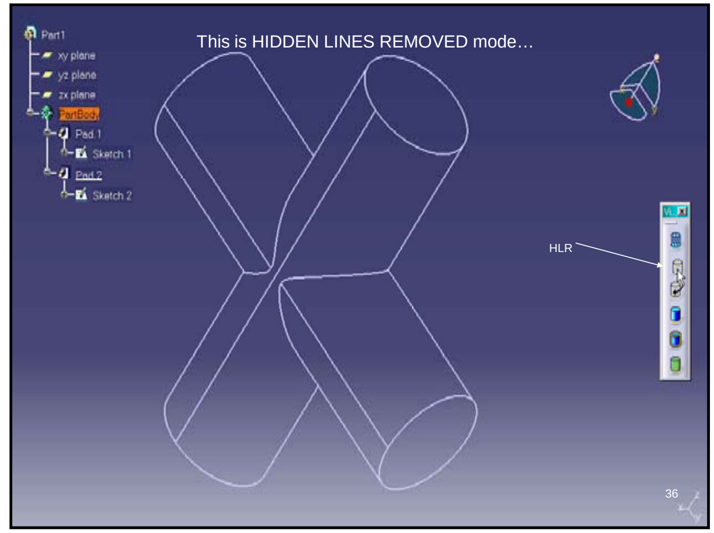

The tree responds the same as a part does for manipulation. To move the tree, simply place the cursor near it and click and hold the middle mouse button. Now drag the tree wherever you wanted it.
FROM HERE…
TO HERE…
Here we have moved the tree by dragging it from one corner of the screen to the other.
31

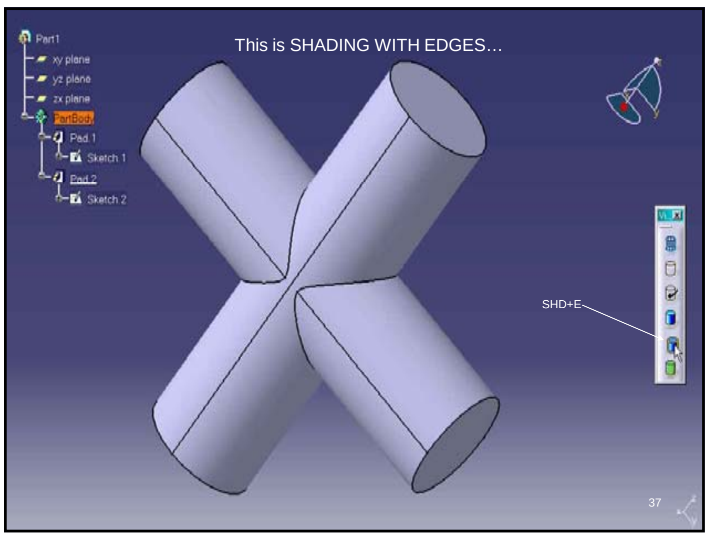

By using the same mouse clicks to ZOOM as you did with a part, you can make your tree larger or smaller as you need to.
TO THIS…
FROM THIS…
Click and hold the middle mouse button, while single clicking the left mouse button. While holding the middle button, move the mouse toward and away from you to make the tree bigger or smaller
32

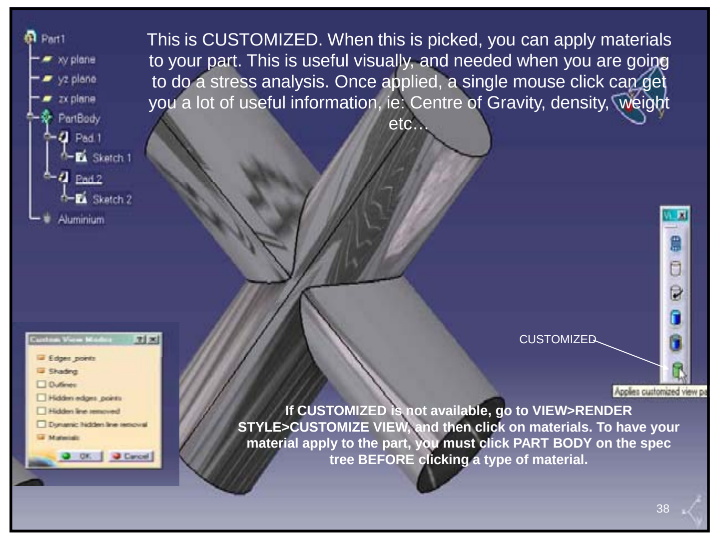

To SHRINK or EXPAND your specification tree, you simply click on the + or – signs.
Part1
xy plane
yz plane
zx plane
PartBody
CLICK HERE TO OPEN TO THIS…
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pad.2
Sketch.2
Clicking a + opens up the tree into it's individual branches. Clicking a – does the reverse.
33


Different RENDERING STYLES give you different views of your part. The most common one is SHADING. It is chosen by clicking on it in the VISUALIZATION toolbar.
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pad.2
Sketch.2
VISUALIZATION TOOLBAR
SHADING
34


This is the same part, but with the WIREFRAME picked…
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch 1
Pad.2
Sketch 2
WIREFRAME
35


This is HIDDEN LINES REMOVED mode…
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch 1
Pad.2
Sketch 2
HLR
36

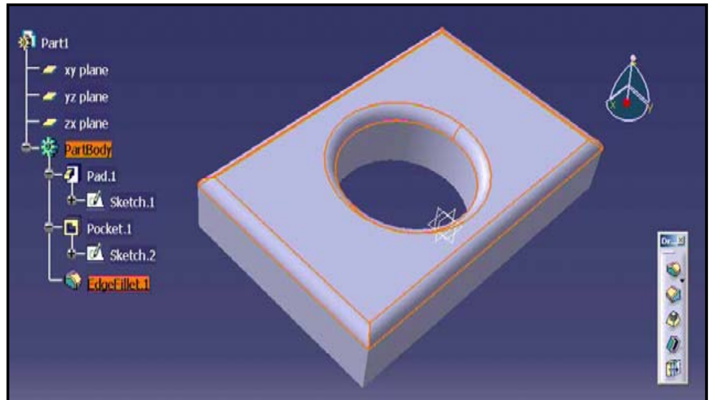

This is SHADING WITH EDGES…
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch 1
Pad.2
Sketch 2
SHD+E
37


This is CUSTOMIZED. When this is picked, you can apply materials to your part. This is useful visually, and needed when you are going to do a stress analysis. Once applied, a single mouse click can get you a lot of useful information, ie: Centre of Gravity, density, weight etc…
Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch 1
Pad.2
Sketch 2
Aluminium
CUSTOMIZED
If CUSTOMIZED is not available, go to VIEW>RENDER STYLE>CUSTOMIZE VIEW, and then click on materials. To have your material apply to the part, you must click PART BODY on the spec tree BEFORE clicking a type of material.
38

Lab-03
Chapter-04

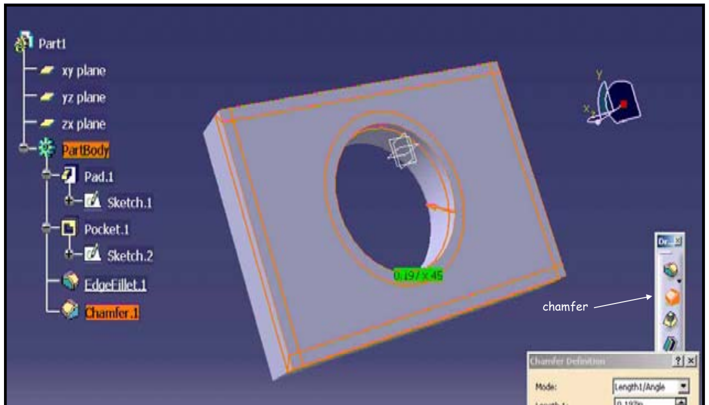
ADVANCED DRESS-UP FEATURES

39



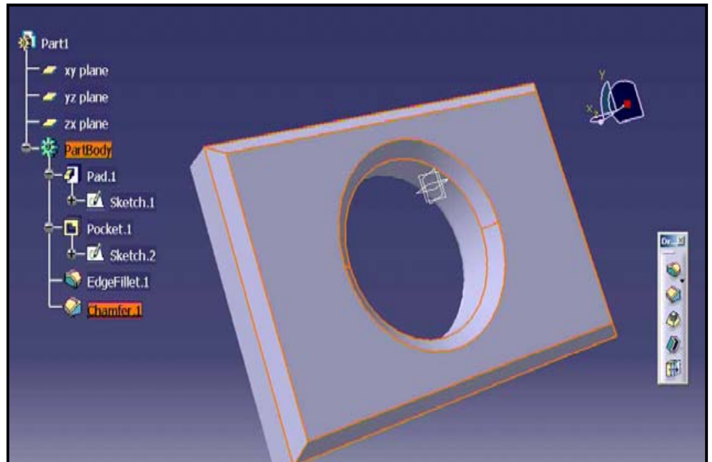
Once OK has been selected, your part will appear with the filleted area highlighted by orange lines at the boundaries. Click anywhere OFF the part to get rid of the lines and see your part

40



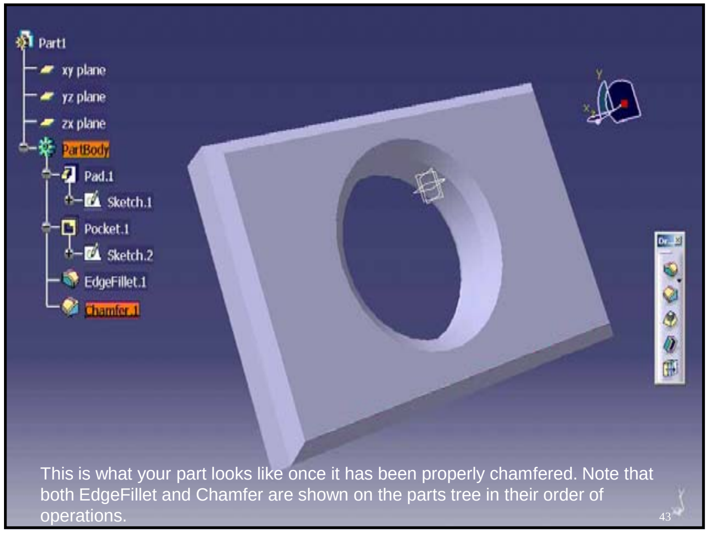
A **CHAMFER** is used the same way as the fillet command. Important changes that you can make in the Chamfer definition box include changing the **length** and **angle** of the chamfer.

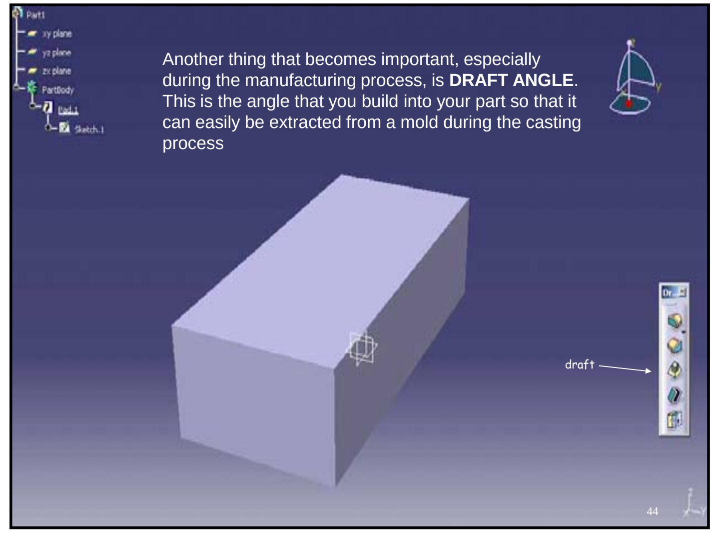
41

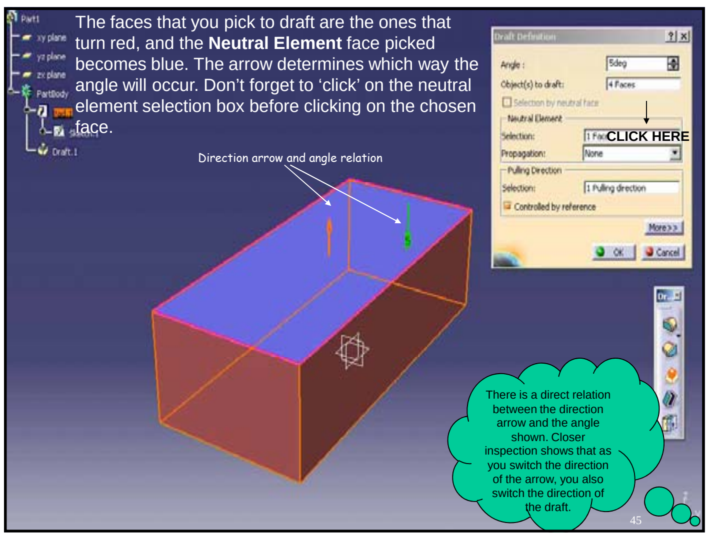


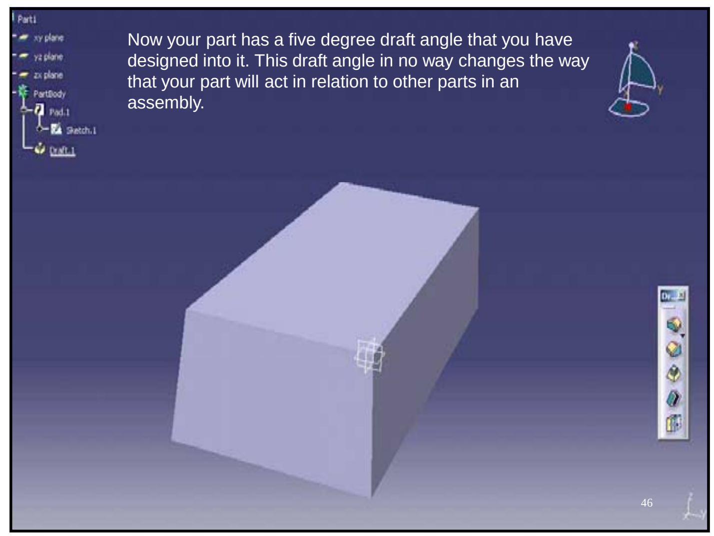
Just like FILLET, the CHAMFER is shown with an orange border until you click off of the part in the blue area.

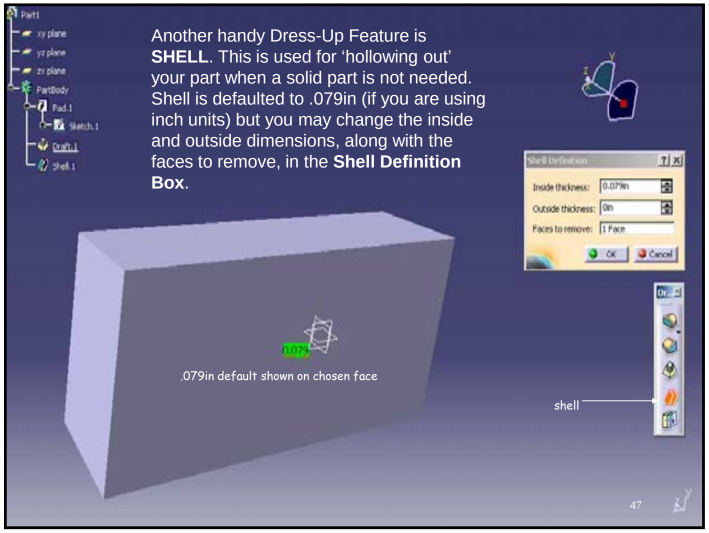
42

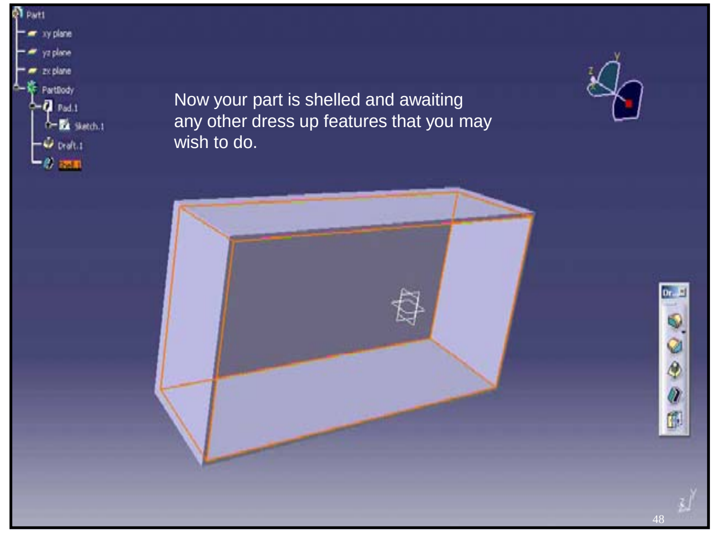

Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Pocket.1
Sketch.2
EdgeFillet.1
Chamfer.1
This is what your part looks like once it has been properly chamfered. Note that both EdgeFillet and Chamfer are shown on the parts tree in their order of operations.
43

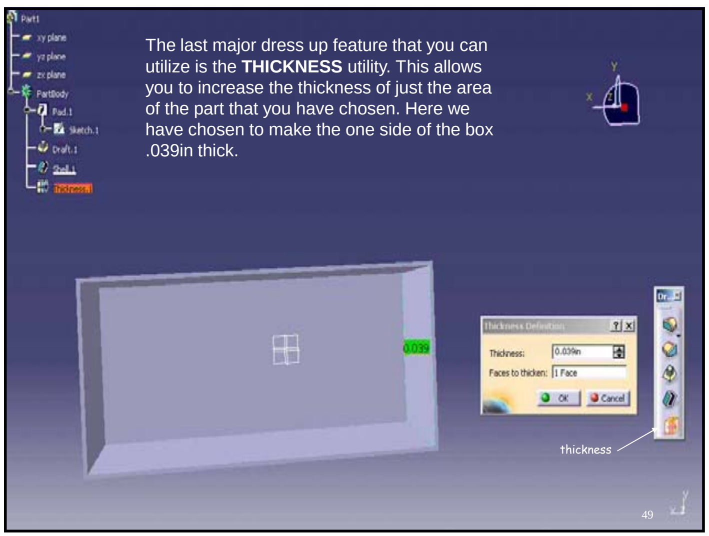

Another thing that becomes important, especially during the manufacturing process, is **DRAFT ANGLE**. This is the angle that you build into your part so that it can easily be extracted from a mold during the casting process
draft
44

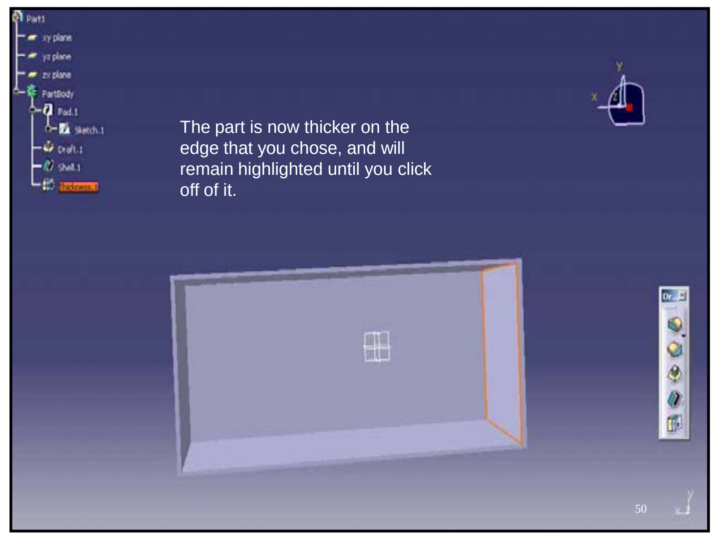

The faces that you pick to draft are the ones that turn red, and the **Neutral Element** face picked becomes blue. The arrow determines which way the angle will occur. Don't forget to 'click' on the neutral element selection box before clicking on the chosen face.
Direction arrow and angle relation
CLICK HERE
There is a direct relation between the direction arrow and the angle shown. Closer inspection shows that as you switch the direction of the arrow, you also switch the direction of the draft.
45

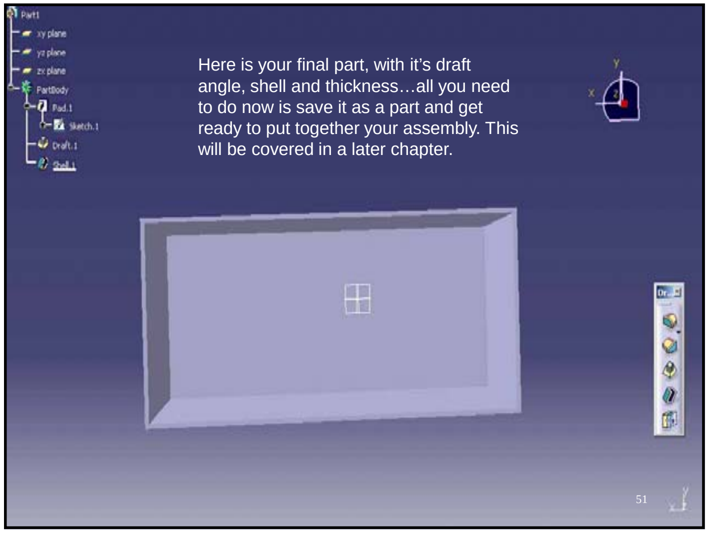

Now your part has a five degree draft angle that you have designed into it. This draft angle in no way changes the way that your part will act in relation to other parts in an assembly.
46


Another handy Dress-Up Feature is **SHELL**. This is used for 'hollowing out' your part when a solid part is not needed. Shell is defaulted to .079in (if you are using inch units) but you may change the inside and outside dimensions, along with the faces to remove, in the **Shell Definition Box**.
Shell Definition
Inside thickness: 0.079in
Outside thickness: 0in
Faces to remove: 1 Face
OK
Cancel
0.079
.079in default shown on chosen face
shell
47


Now your part is shelled and awaiting any other dress up features that you may wish to do.
48


The last major dress up feature that you can utilize is the **THICKNESS** utility. This allows you to increase the thickness of just the area of the part that you have chosen. Here we have chosen to make the one side of the box .039in thick.
Thickness Definition
Thickness: 0.039in
Faces to thicken: 1 Face
OK
Cancel
0.039
thickness
49

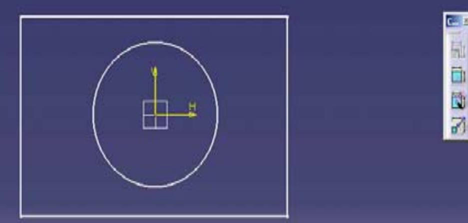

The part is now thicker on the edge that you chose, and will remain highlighted until you click off of it.
50


Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Draft.1
Shell.1
Here is your final part, with it's draft angle, shell and thickness…all you need to do now is save it as a part and get ready to put together your assembly. This will be covered in a later chapter.
51

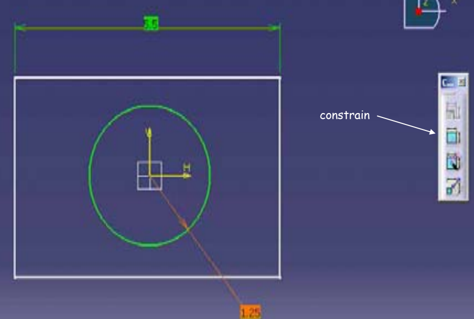
Lab-04
Chapter Five

CONSTRAINTS

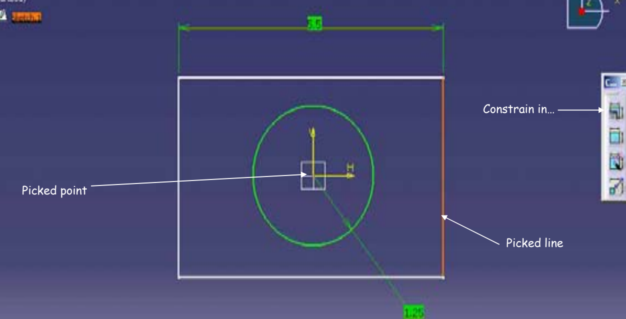
You do your **CONSTRAINING** in Sketcher mode to create your part to exacting dimensions. This is the opposite of free-form creating we have done up to this point.

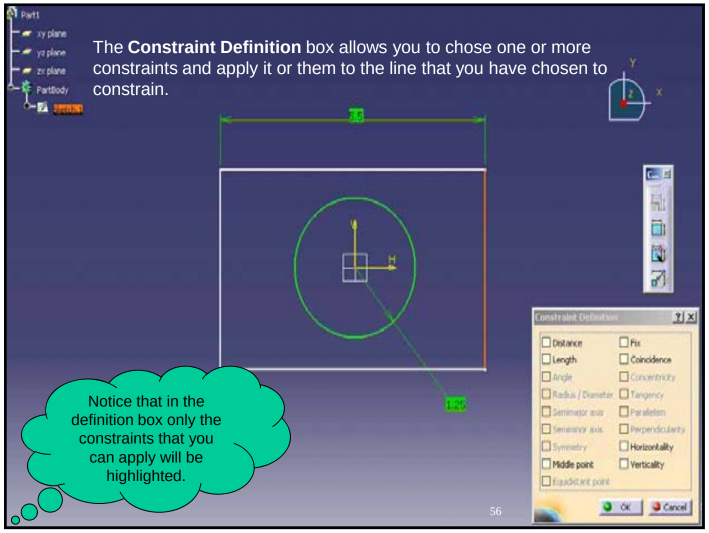


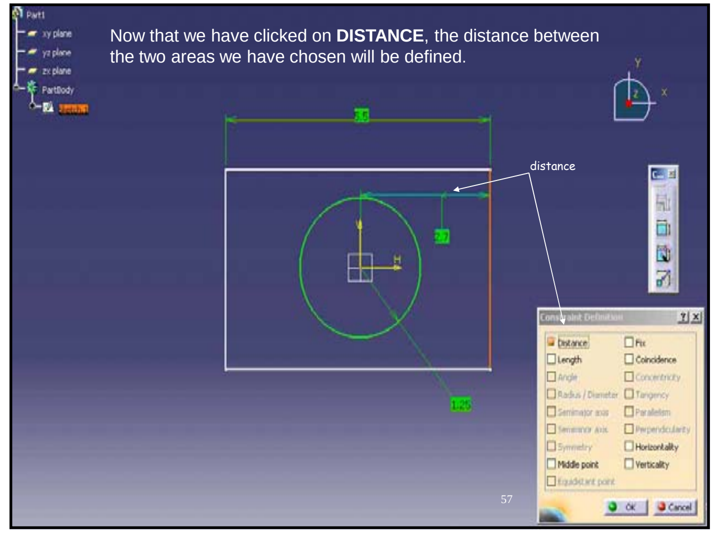
Pick the edge that you want to constrain, or give a definite dimension to, and then click **CONSTRAIN** from the toolbar.

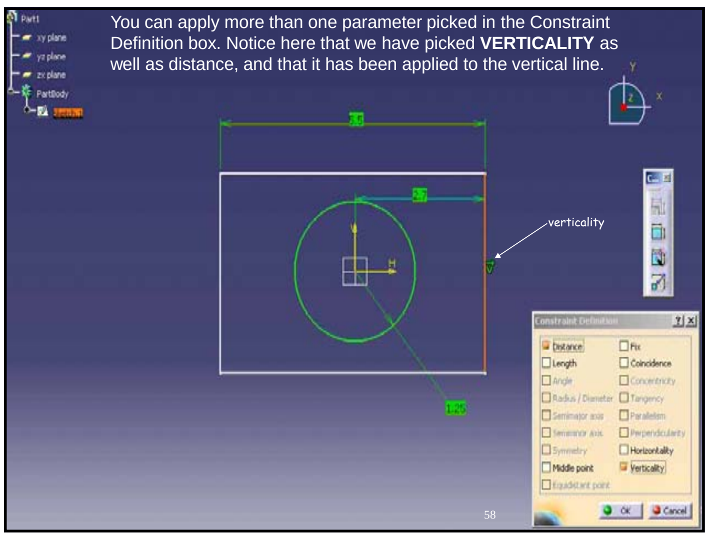


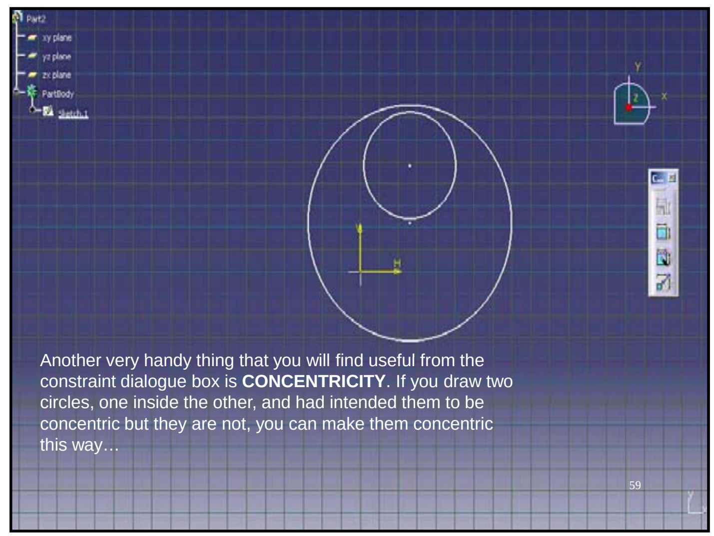
Another way to constrain a line on your part is to pick it and then click on **CONSTRAIN IN A DIALOGUE BOX**. You will have many different ways of constraining the area that will appear in the dialogue box. The ones that you can use will allow you to pick them.

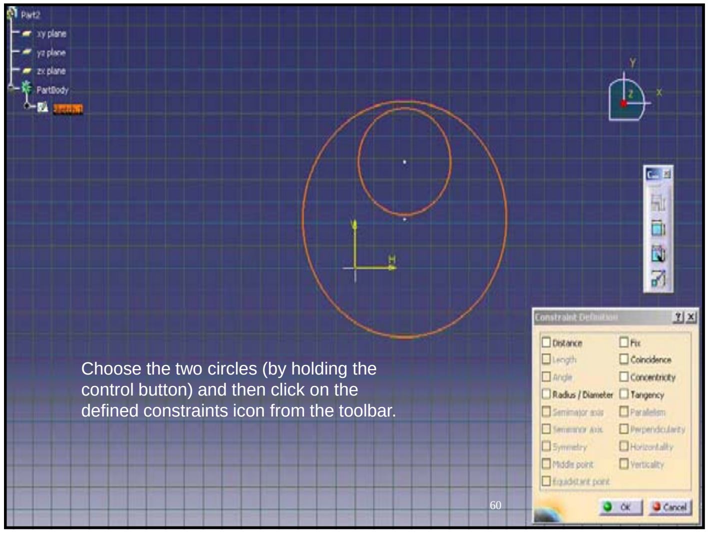


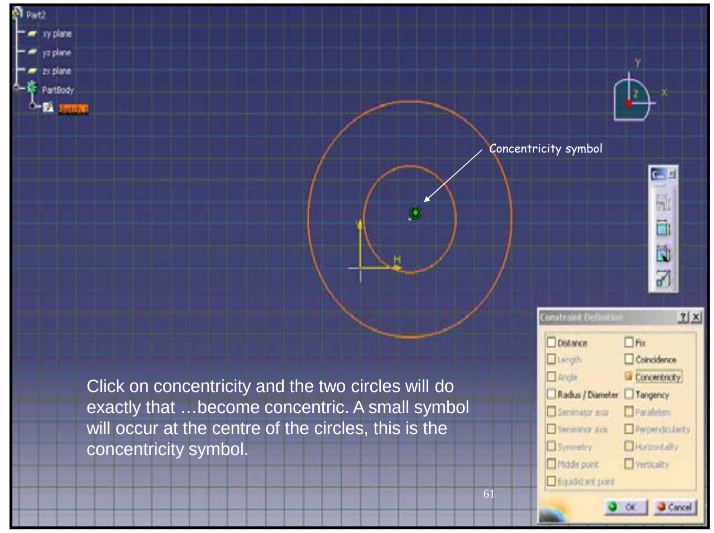

The **Constraint Definition** box allows you to chose one or more constraints and apply it or them to the line that you have chosen to constrain.
Notice that in the definition box only the constraints that you can apply will be highlighted.
56

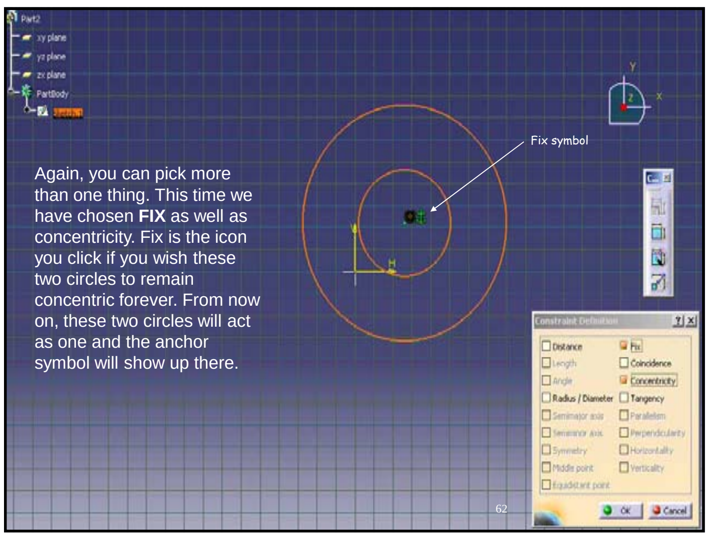

Now that we have clicked on **DISTANCE**, the distance between the two areas we have chosen will be defined.
distance
57

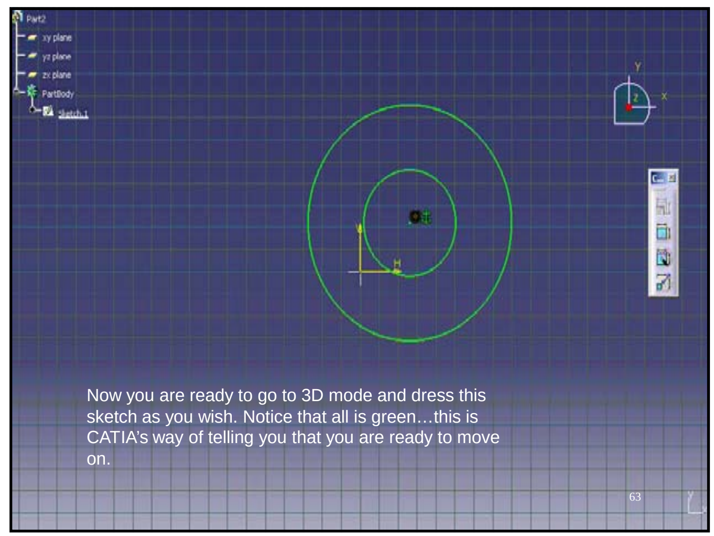

You can apply more than one parameter picked in the Constraint Definition box. Notice here that we have picked **VERTICALITY** as well as distance, and that it has been applied to the vertical line.
verticality
58


Another very handy thing that you will find useful from the constraint dialogue box is **CONCENTRICITY**. If you draw two circles, one inside the other, and had intended them to be concentric but they are not, you can make them concentric this way…
59


Choose the two circles (by holding the control button) and then click on the defined constraints icon from the toolbar.
60


Concentricity symbol
Click on concentricity and the two circles will do exactly that …become concentric. A small symbol will occur at the centre of the circles, this is the concentricity symbol.
61

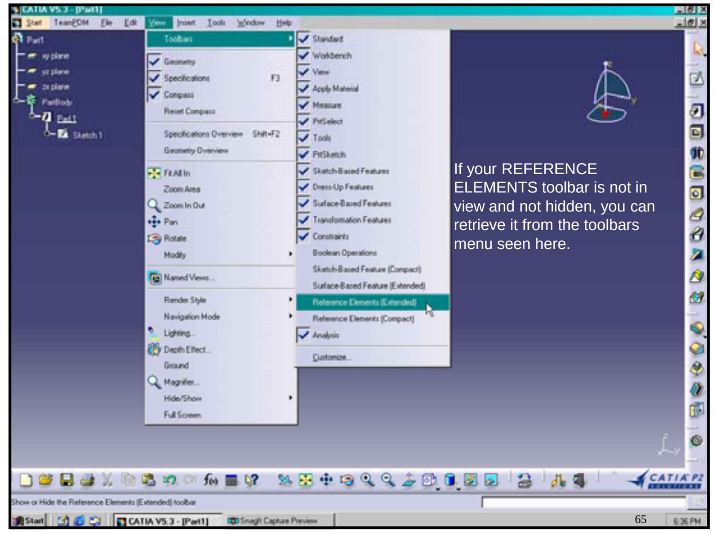

Fix symbol
Again, you can pick more than one thing. This time we have chosen **FIX** as well as concentricity. Fix is the icon you click if you wish these two circles to remain concentric forever. From now on, these two circles will act as one and the anchor symbol will show up there.
62


Now you are ready to go to 3D mode and dress this sketch as you wish. Notice that all is green…this is CATIA's way of telling you that you are ready to move on.
63

Lab-05
Chapter-06

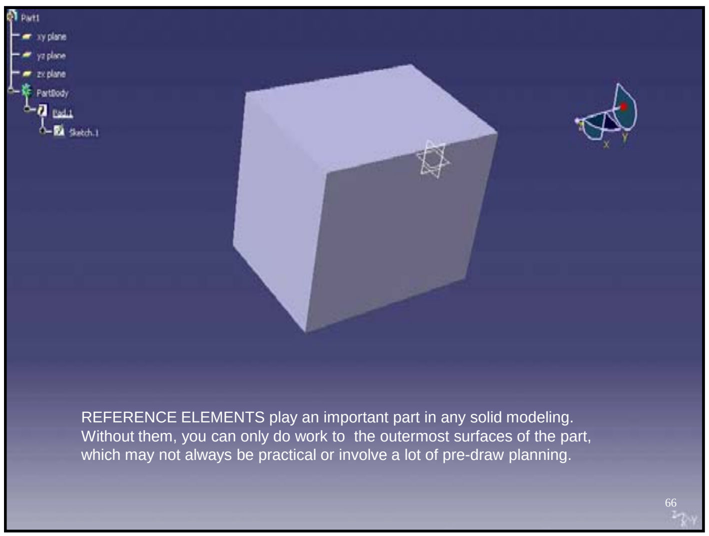
REFERENCE ELEMENTS

64



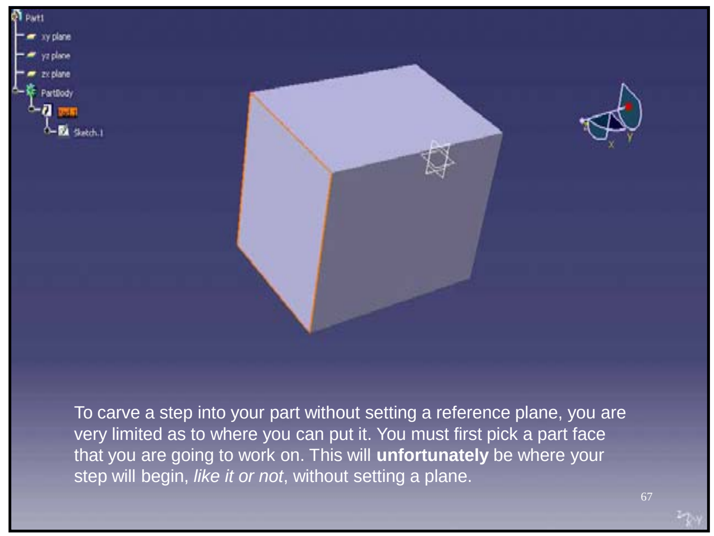
If your REFERENCE ELEMENTS toolbar is not in view and not hidden, you can retrieve it from the toolbars menu seen here.

65



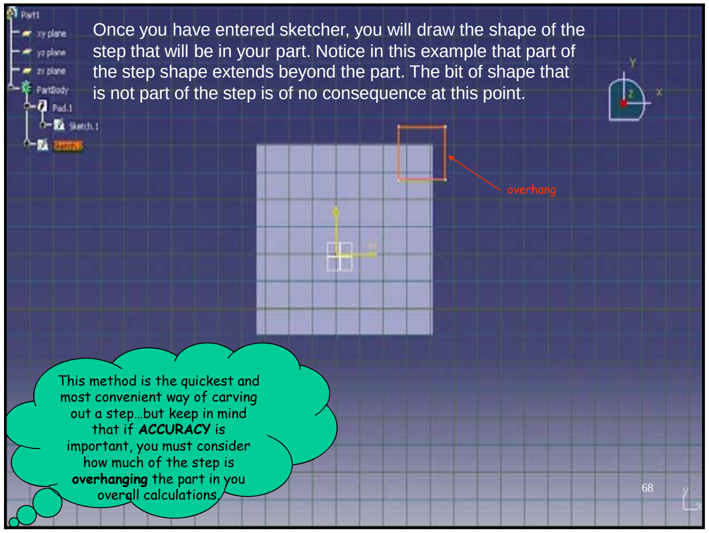
REFERENCE ELEMENTS play an important part in any solid modeling. Without them, you can only do work to the outermost surfaces of the part, which may not always be practical or involve a lot of pre-draw planning.

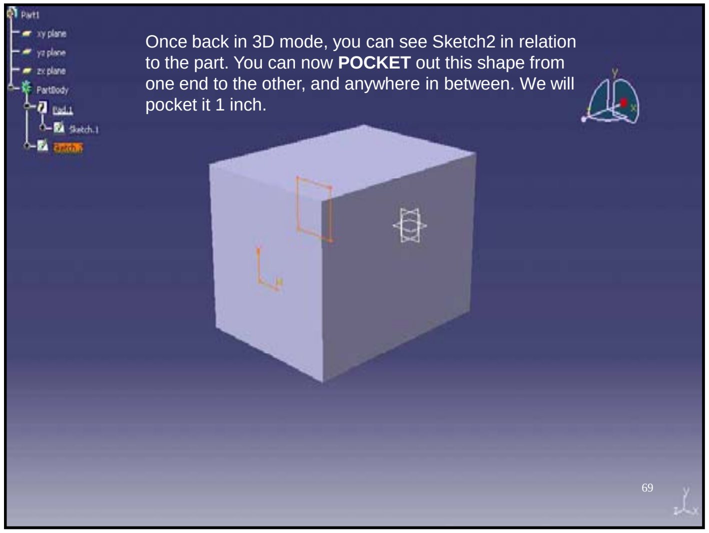
66

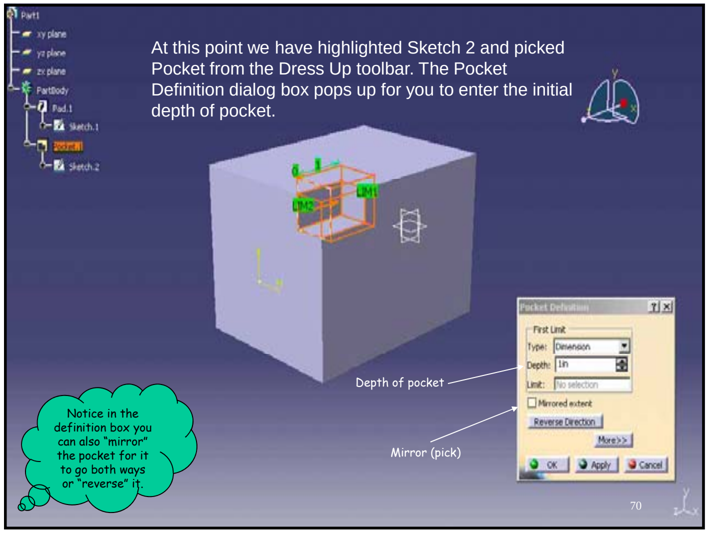


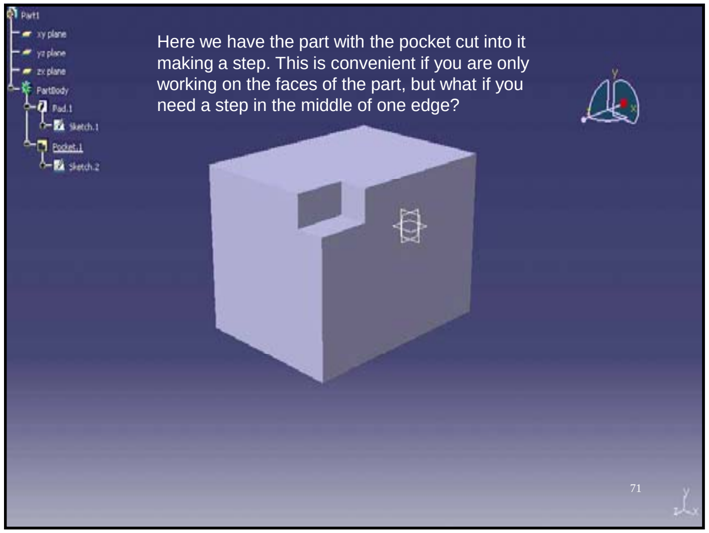
To carve a step into your part without setting a reference plane, you are very limited as to where you can put it. You must first pick a part face that you are going to work on. This will **unfortunately** be where your step will begin, *like it or not*, without setting a plane.

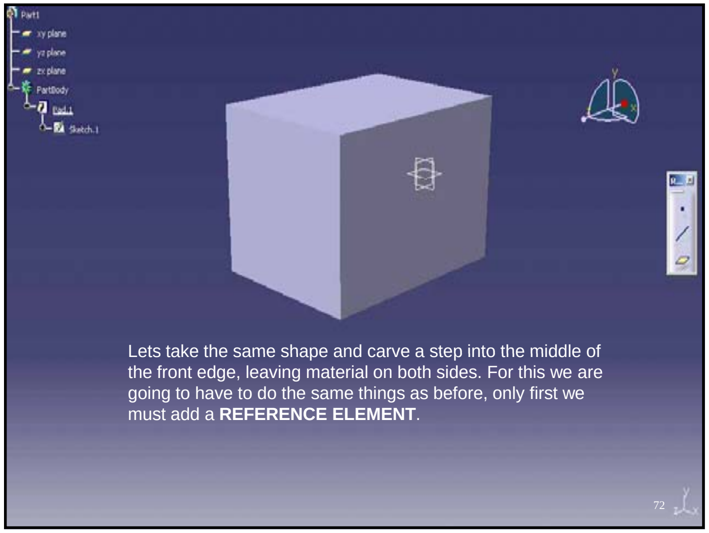
67

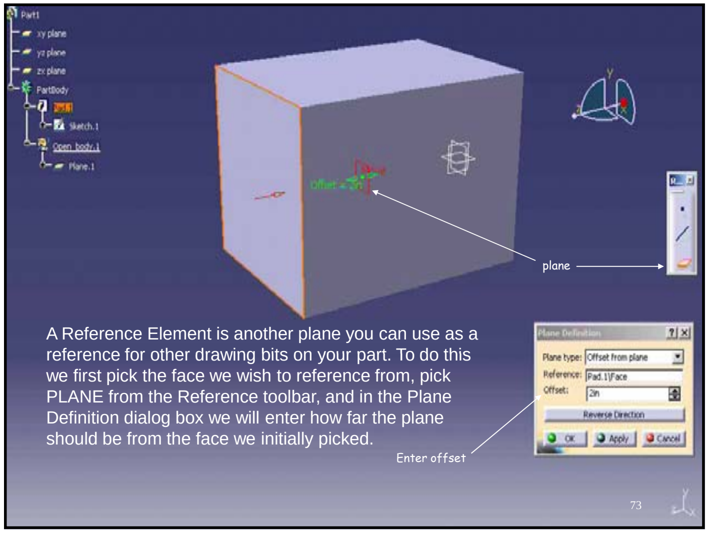

Once you have entered sketcher, you will draw the shape of the step that will be in your part. Notice in this example that part of the step shape extends beyond the part. The bit of shape that is not part of the step is of no consequence at this point.
overhang
This method is the quickest and most convenient way of carving out a step…but keep in mind that if ACCURACY is important, you must consider how much of the step is overhanging the part in you overall calculations.
68

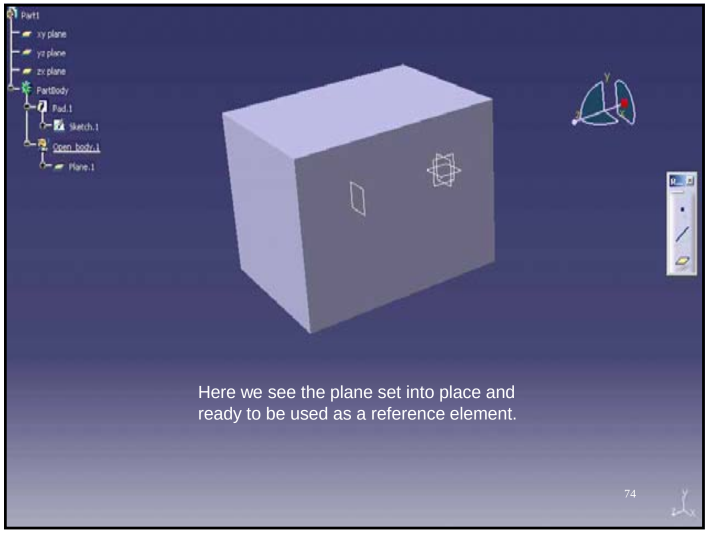

Once back in 3D mode, you can see Sketch2 in relation to the part. You can now POCKET out this shape from one end to the other, and anywhere in between. We will pocket it 1 inch.
69

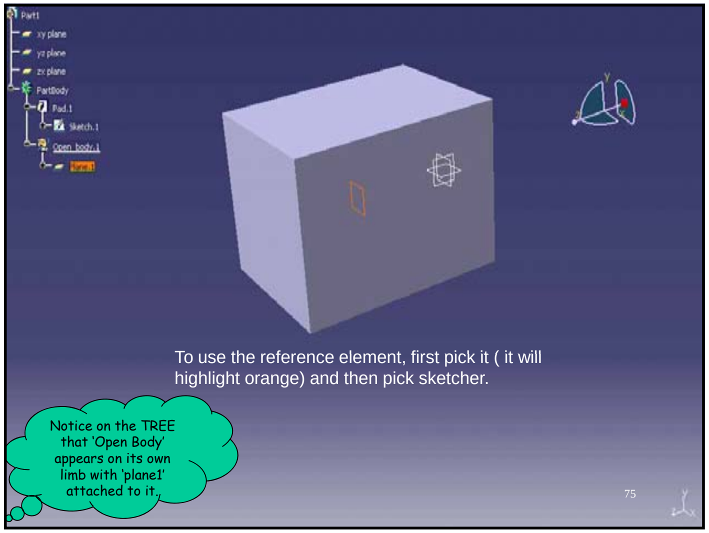

At this point we have highlighted Sketch 2 and picked Pocket from the Dress Up toolbar. The Pocket Definition dialog box pops up for you to enter the initial depth of pocket.
Depth of pocket
Mirror (pick)
Notice in the definition box you can also "mirror" the pocket for it to go both ways or "reverse" it.
70

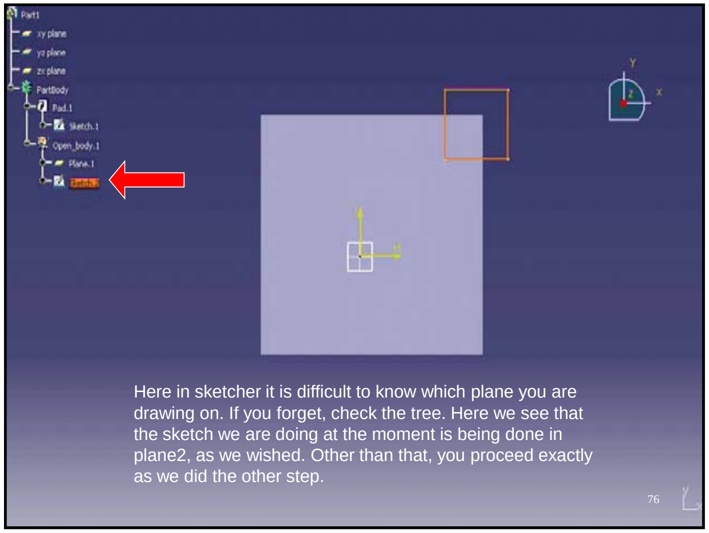

Here we have the part with the pocket cut into it making a step. This is convenient if you are only working on the faces of the part, but what if you need a step in the middle of one edge?
71

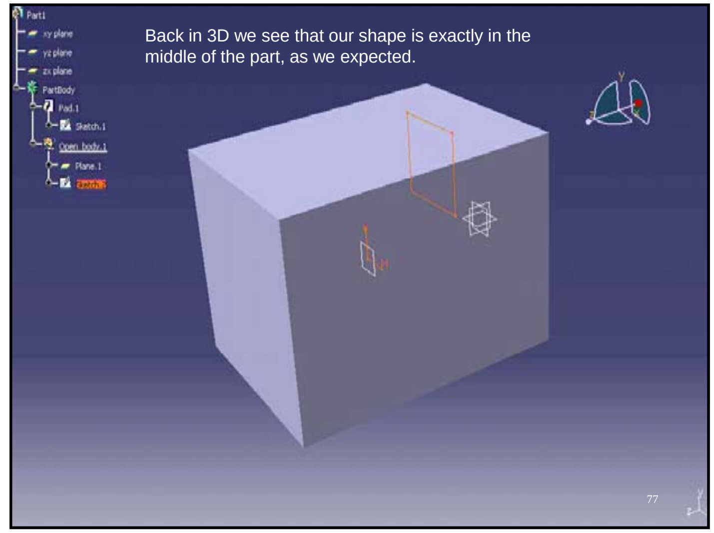

Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Lets take the same shape and carve a step into the middle of the front edge, leaving material on both sides. For this we are going to have to do the same things as before, only first we must add a **REFERENCE ELEMENT**.
72

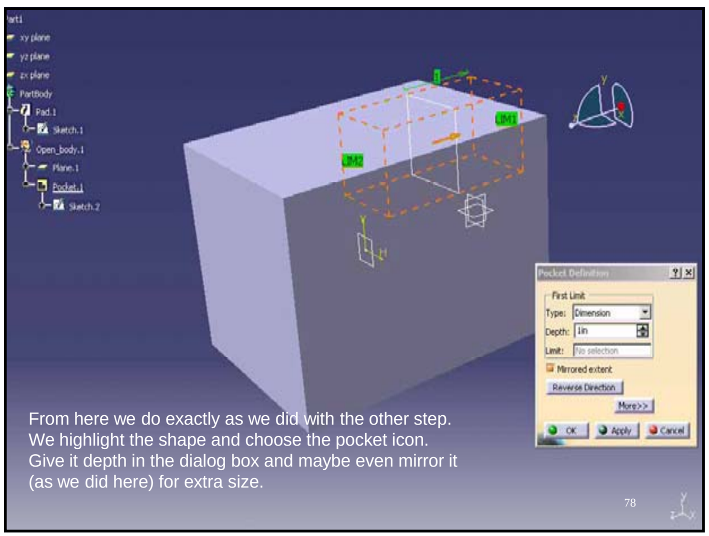

Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Open body.1
Plane.1
plane
A Reference Element is another plane you can use as a reference for other drawing bits on your part. To do this we first pick the face we wish to reference from, pick PLANE from the Reference toolbar, and in the Plane Definition dialog box we will enter how far the plane should be from the face we initially picked.
Plane Definition
Plane type: Offset from plane
Reference: Pad.1\Face
Offset: 2in
Reverse Direction
OK
Apply
Cancel
Enter offset
73


Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Open body.1
Plane.1
Here we see the plane set into place and ready to be used as a reference element.
74


Part1
xy plane
yz plane
zx plane
PartBody
Pad.1
Sketch.1
Open body.1
Plane.1
To use the reference element, first pick it (it will highlight orange) and then pick sketcher.
Notice on the TREE that 'Open Body' appears on its own limb with 'plane1' attached to it.
75


Here in sketcher it is difficult to know which plane you are drawing on. If you forget, check the tree. Here we see that the sketch we are doing at the moment is being done in plane2, as we wished. Other than that, you proceed exactly as we did the other step.
76

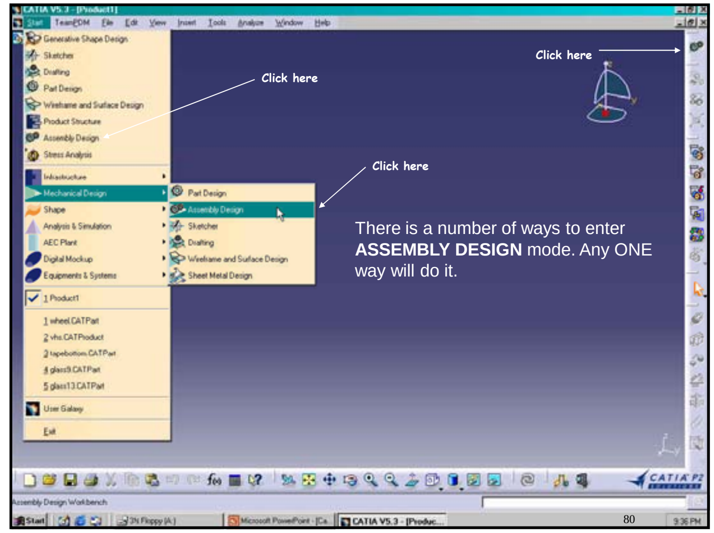

Back in 3D we see that our shape is exactly in the middle of the part, as we expected.
77

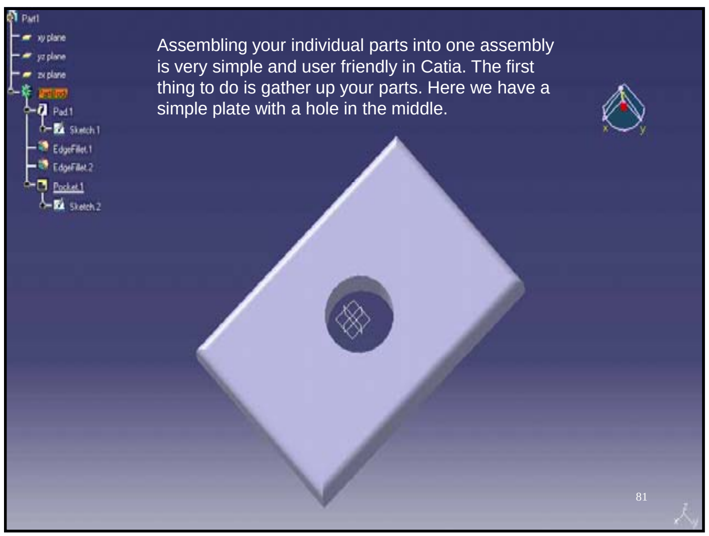

From here we do exactly as we did with the other step. We highlight the shape and choose the pocket icon. Give it depth in the dialog box and maybe even mirror it (as we did here) for extra size.
78

Lab-05
Chapter-07

BASIC ASSEMBLY DESIGN

79



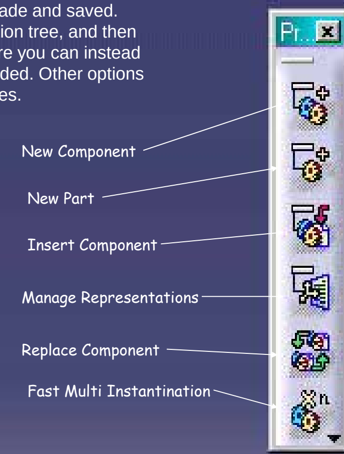


Product1

Now you are in Assembly mode. Notice on the beginning of the new specification tree it says product. Your product will be the assembly of the parts you made.

83

This is your **PRODUCT STRUCTURE** toolbar. It is how you will bring in your parts you have made and saved. Single click “Product” in your specification tree, and then click **INSERT COMPONENT**. From here you can instead click **NEW PART** and draw one as needed. Other options are also here self-evident by their names.



84

Product1
Part1 (Part1.1)

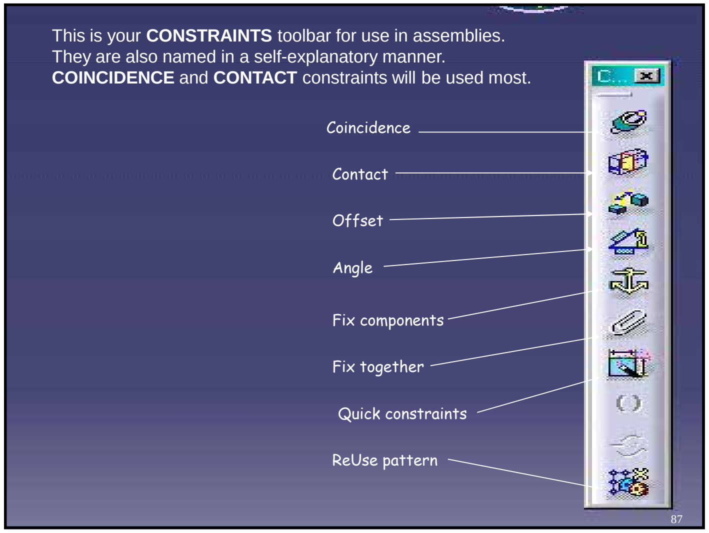
Here is the first part brought in using Insert Component. It's existence is reflected in the specification tree. Now we will bring in the next part the same way.

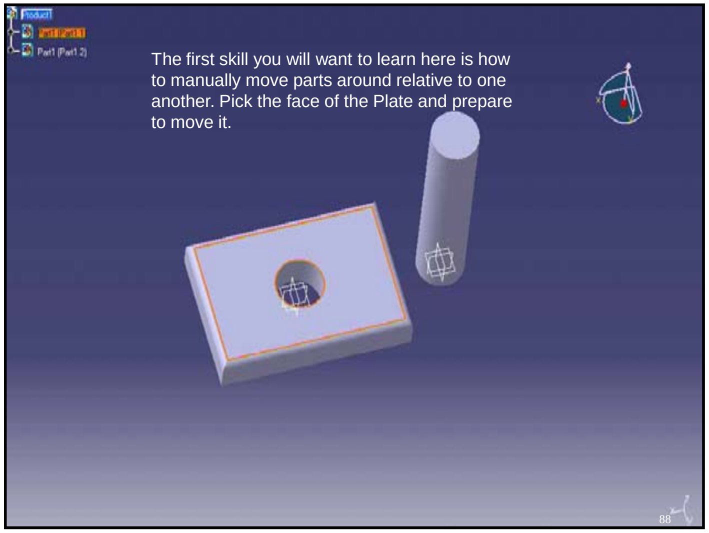
85

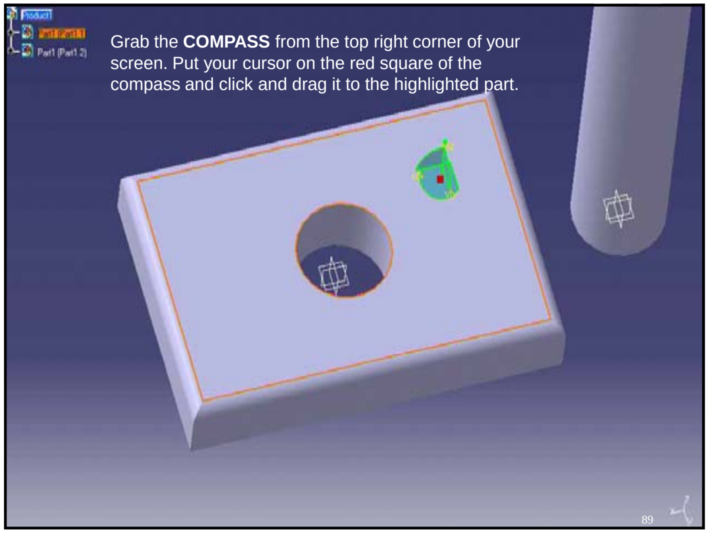
Product1
Part1 (Part1.1)
Part1 (Part1.2)

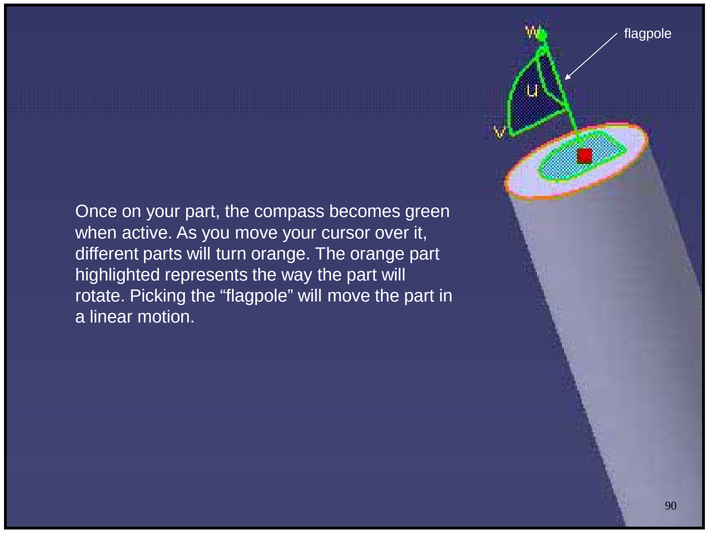
Both parts are now in the Assembly environment, and both are reflected in the tree. They enter the environment in approximately the same orientation that they were saved. You can now move and orient them the way you wish. Catia will help you here.

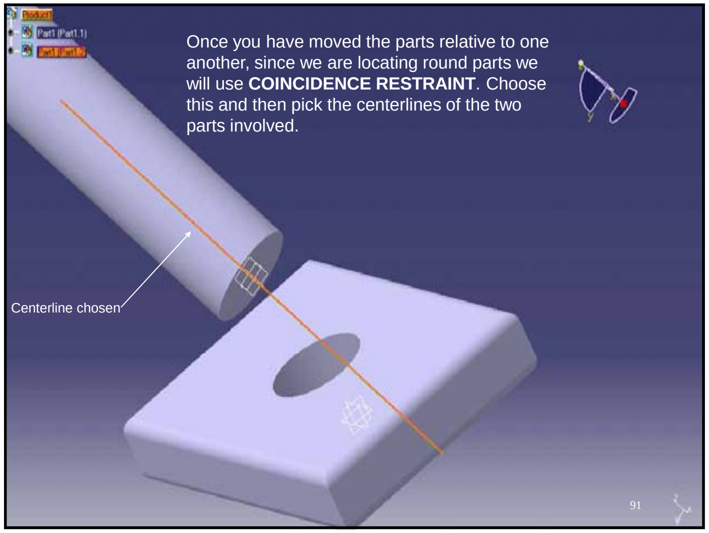
86

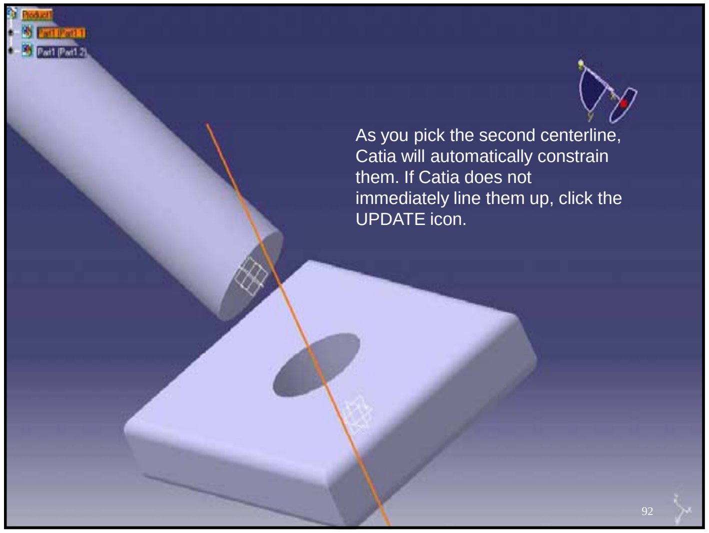

This is your **CONSTRAINTS** toolbar for use in assemblies. They are also named in a self-explanatory manner. **COINCIDENCE** and **CONTACT** constraints will be used most.
Coincidence
Contact
Offset
Angle
Fix components
Fix together
Quick constraints
ReUse pattern
87

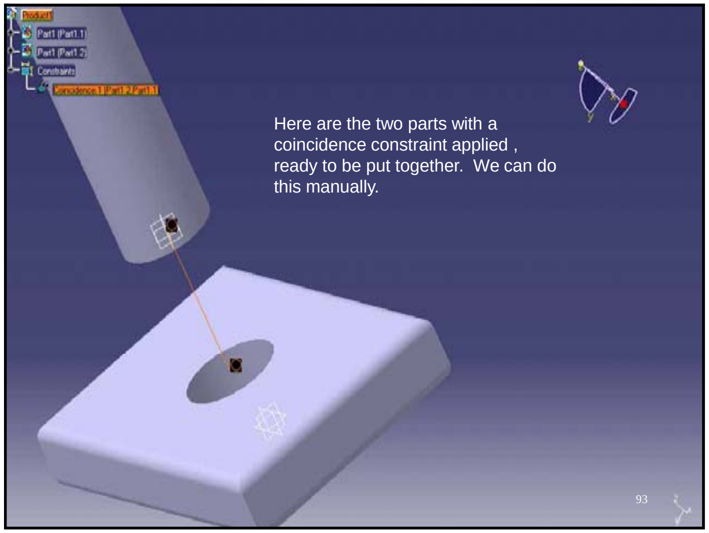

The first skill you will want to learn here is how to manually move parts around relative to one another. Pick the face of the Plate and prepare to move it.
88

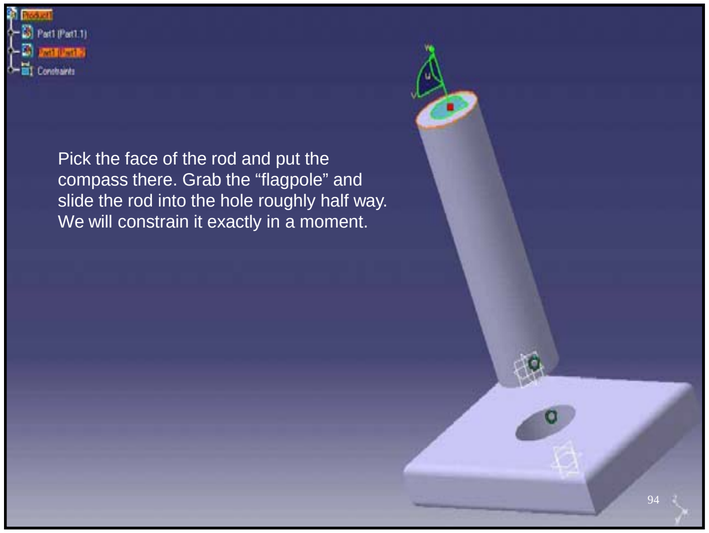

Grab the **COMPASS** from the top right corner of your screen. Put your cursor on the red square of the compass and click and drag it to the highlighted part.
89

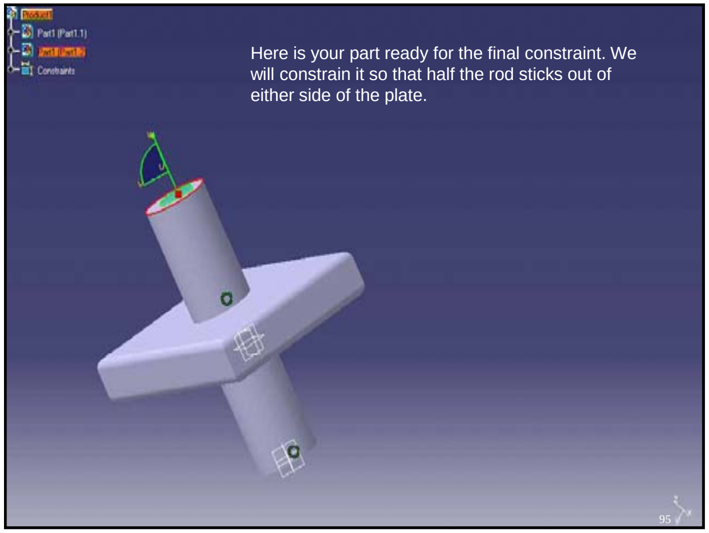

flagpole
W
U
V
Once on your part, the compass becomes green when active. As you move your cursor over it, different parts will turn orange. The orange part highlighted represents the way the part will rotate. Picking the “flagpole” will move the part in a linear motion.
90

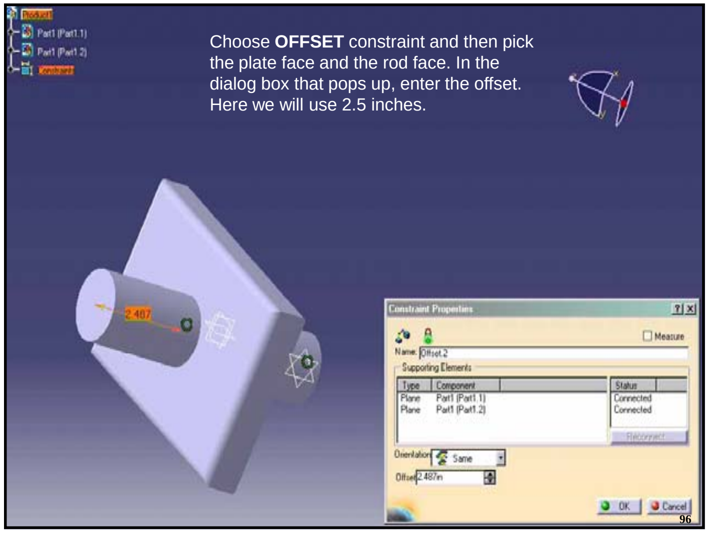

Once you have moved the parts relative to one another, since we are locating round parts we will use **COINCIDENCE RESTRAINT**. Choose this and then pick the centerlines of the two parts involved.
Centerline chosen
91

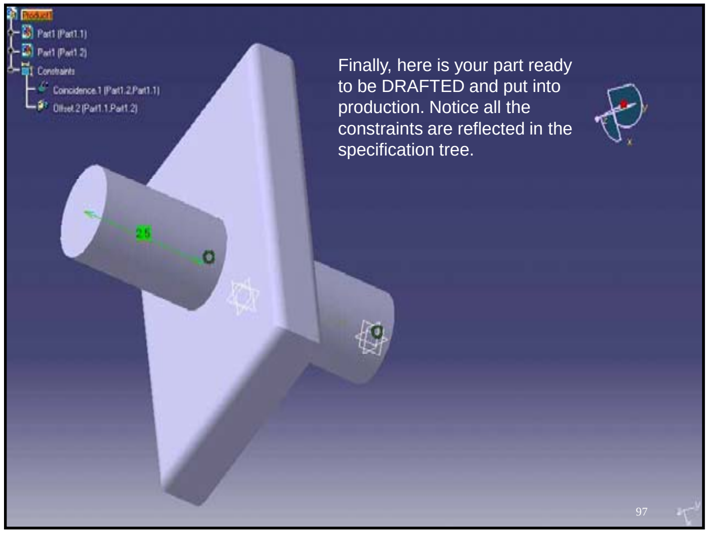

As you pick the second centerline, Catia will automatically constrain them. If Catia does not immediately line them up, click the UPDATE icon.
92


Here are the two parts with a coincidence constraint applied , ready to be put together. We can do this manually.
93


Pick the face of the rod and put the compass there. Grab the “flagpole” and slide the rod into the hole roughly half way. We will constrain it exactly in a moment.
94


Here is your part ready for the final constraint. We will constrain it so that half the rod sticks out of either side of the plate.
95

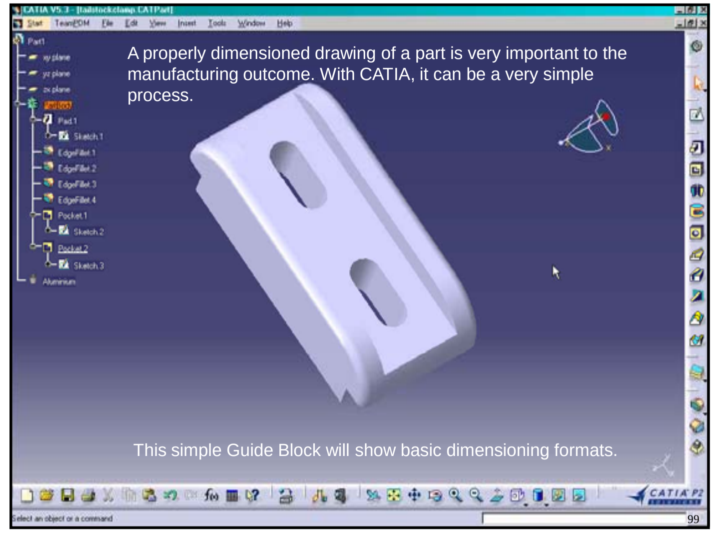

Choose **OFFSET** constraint and then pick the plate face and the rod face. In the dialog box that pops up, enter the offset. Here we will use 2.5 inches.
Constraint Properties
Measure
Name: Offset.2
Supporting Elements
Type
Component
Status
Plane
Part1 (Part1.1)
Connected
Plane
Part1 (Part1.2)
Connected
Orientation
Same
Offset
2.487in
OK
Cancel
96

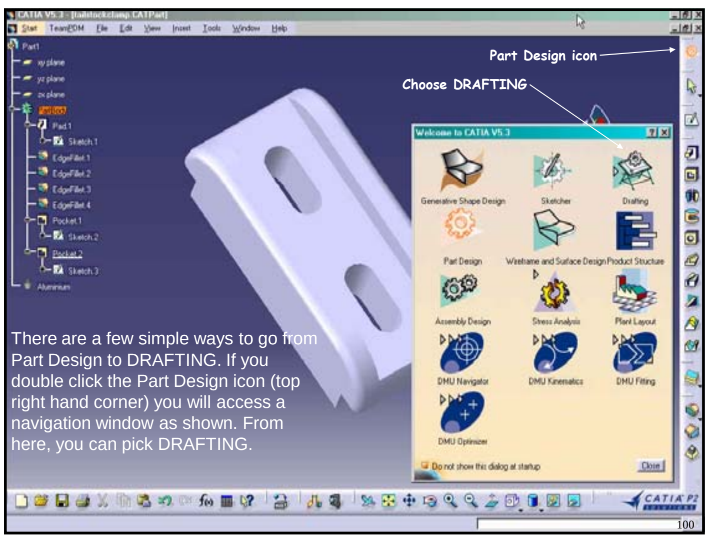

Finally, here is your part ready to be DRAFTED and put into production. Notice all the constraints are reflected in the specification tree.
97

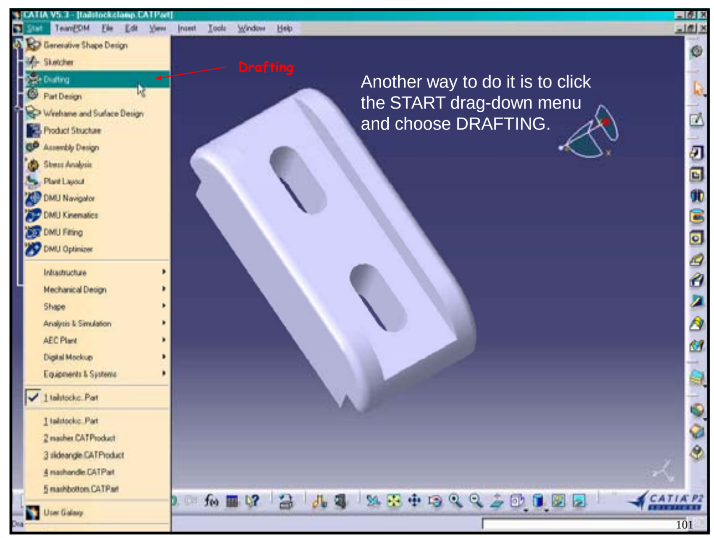
Lab-06
Chapter-08

DRAFTING and DIMENSIONING

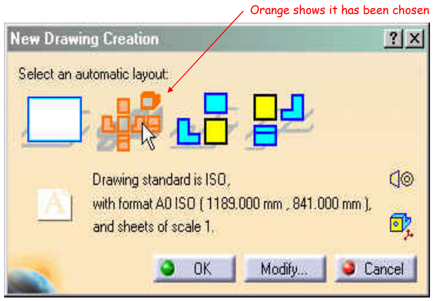
98



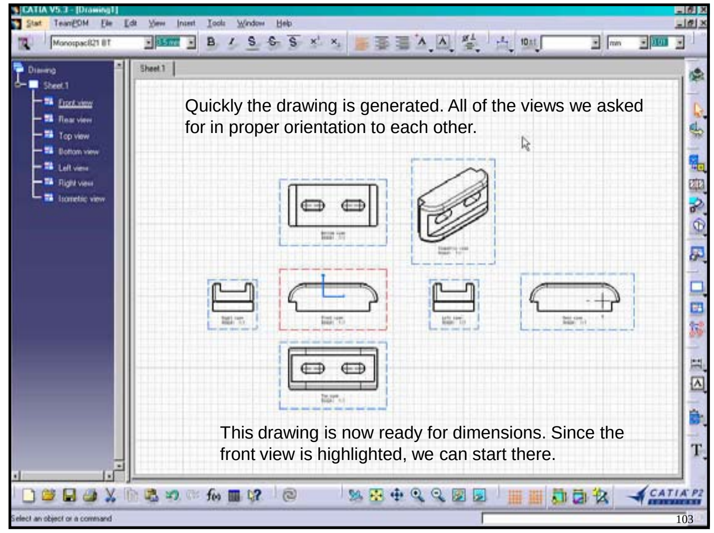




The NEW DRAWING CREATION dialog box will appear.
The first thing to do is pick your automatic layout.

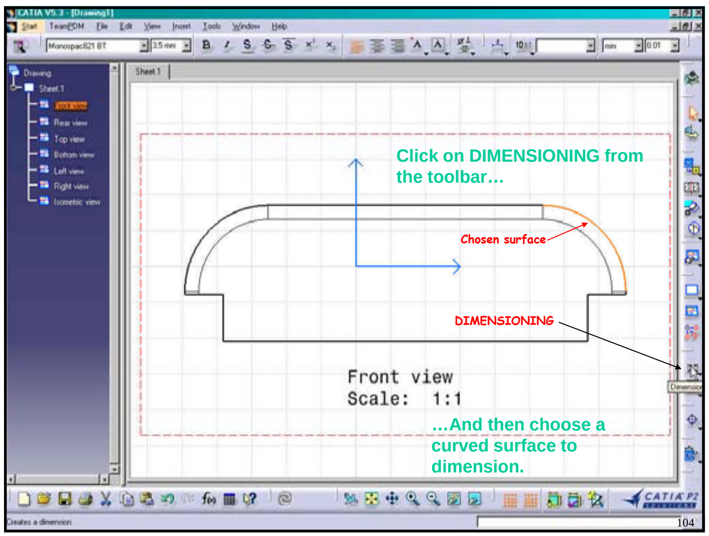


Next, you can choose MODIFY and change ISO and ANSI standards, number of sheets and orientation, scale and others.



Quickly the drawing is generated. All of the views we asked for in proper orientation to each other.

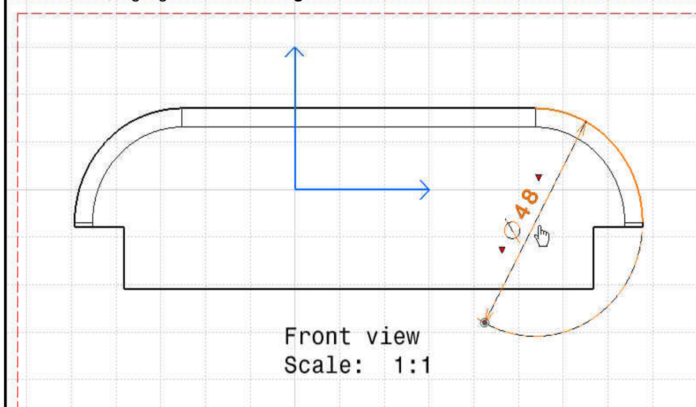
This drawing is now ready for dimensions. Since the front view is highlighted, we can start there.



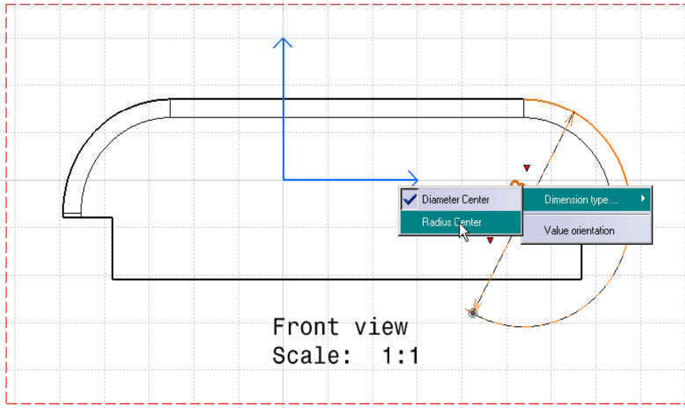
Click on DIMENSIONING from the toolbar…

…And then choose a curved surface to dimension.

If default is **DIAMETER**, this is what you will see. If you wish to dimension a **RADIUS**, highlight the line and ***right click*** on it.

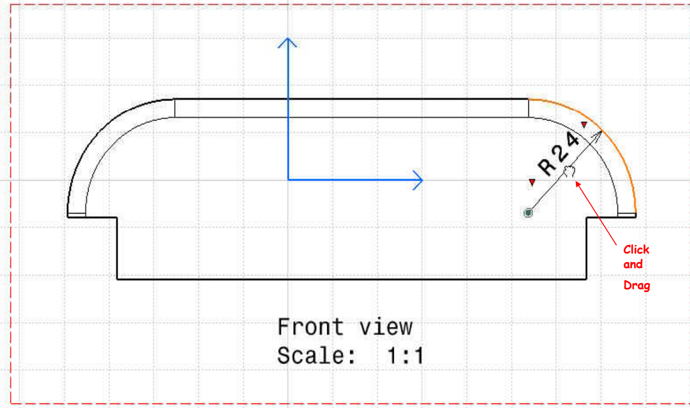


When **DIMENSION TYPE** appears, notice that it is default to Diameter. Highlight **RADIUS** Center and click on it.



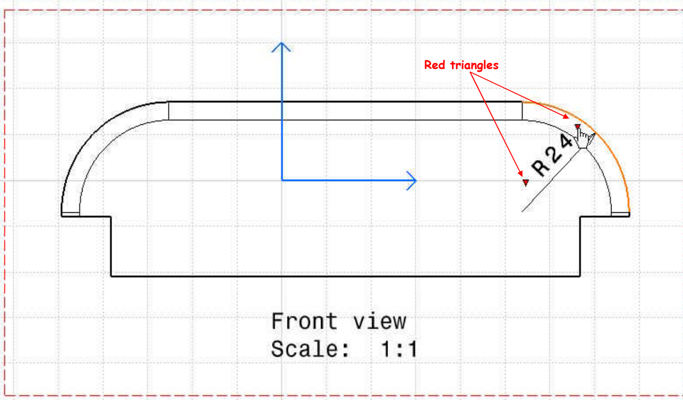
106

Your dimension is now a Radius. Lengthen and shorten the dimension line by dragging it.



107

You can add notes before and after the dimension by clicking on the red triangles

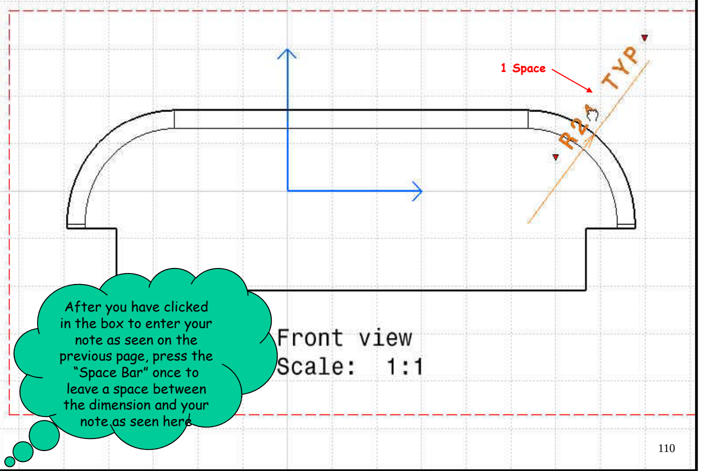


108

This text box will appear. Simply click in the empty field next to "Text After" and type the note. Click on OK when you're done and your note will appear in the drafting.

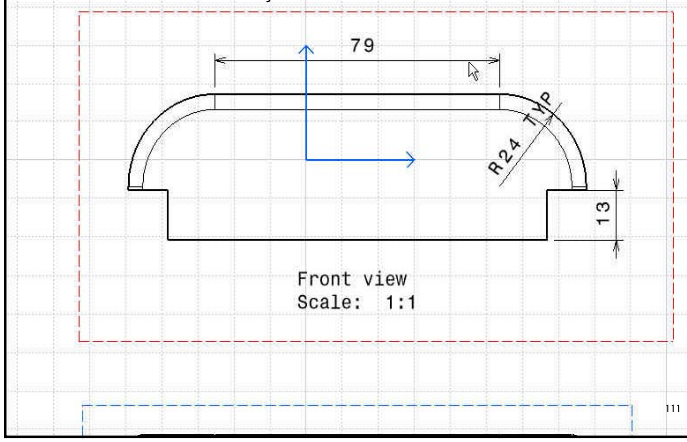
109

In this example, "TYP" was typed to indicate the dimension is TYPICAL for these radii



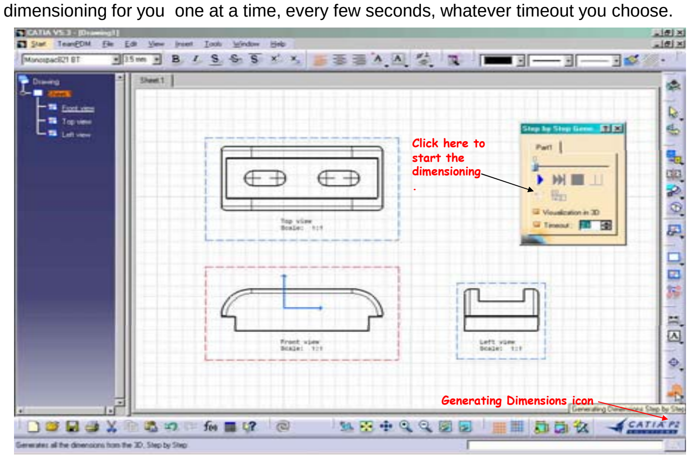
110

All lines can be done in the same way. Simply pick them and click on Dimension. If you are working on a large drawing, this single line picking would be rather tedious. There is another way…



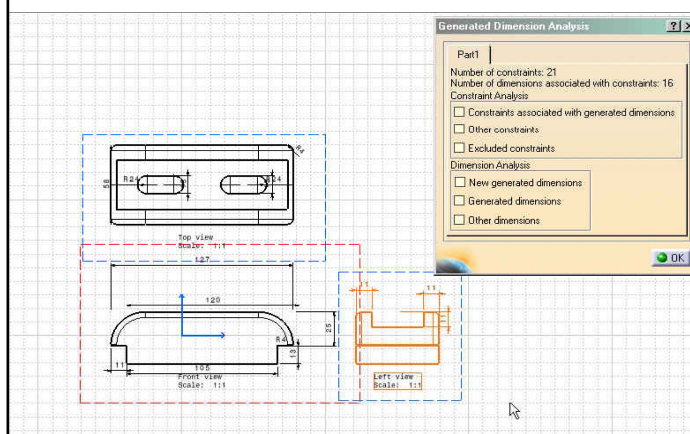
111

Click on **GENERATING DIMENSIONS Step By Step**. This will allow CATIA to do the dimensioning for you one at a time, every few seconds, whatever timeout you choose.

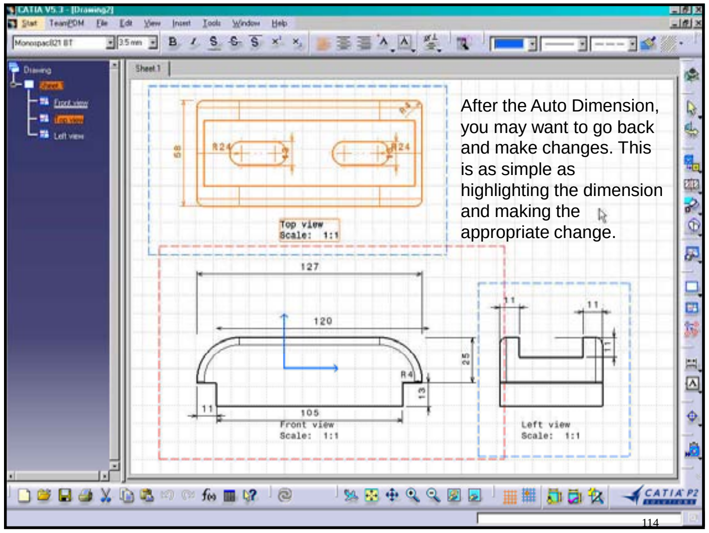


112

Once all of the automatic dimensioning is finished, you will get an ANALYSIS of what was done.

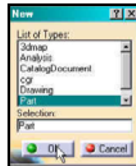


113


After the Auto Dimension, you may want to go back and make changes. This is as simple as highlighting the dimension and making the appropriate change.
Top view
Scale: 1:1
Front view
Scale: 1:1
Left view
Scale: 1:1
127
120
105

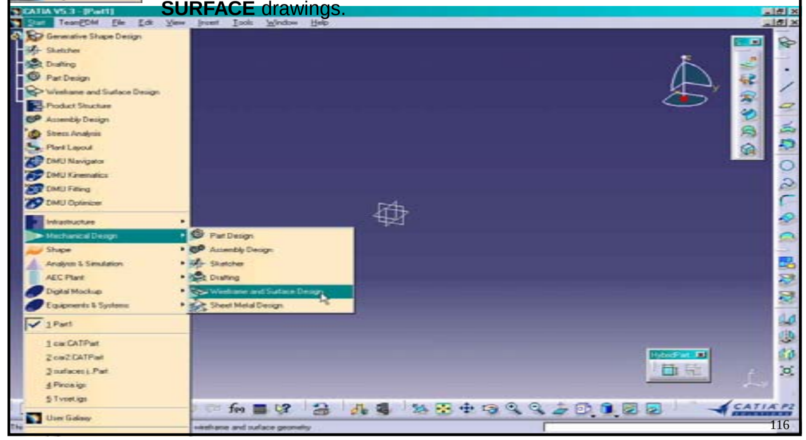
Lab-07
Chapter- 09

DRAWING USING SURFACES



To start your **SURFACES** drawing, go to new drawing, choose **PART**. Once the Part screen appears, click on **START**, choose **MECHANICAL DESIGN**, and then **WIREFRAME and SURFACE DESIGN**.

New icons will appear, and these are the ones you will use for your **SURFACE** drawings.

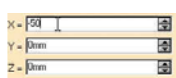


Now we will profile an auto body…

The first thing you will want to do is layout your guide lines. You can do this by choosing POINT from the toolbar, and putting them where you need them.

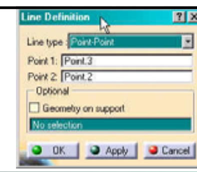
The first one you place is at the origin, or 0,0,0.

Hit OK.

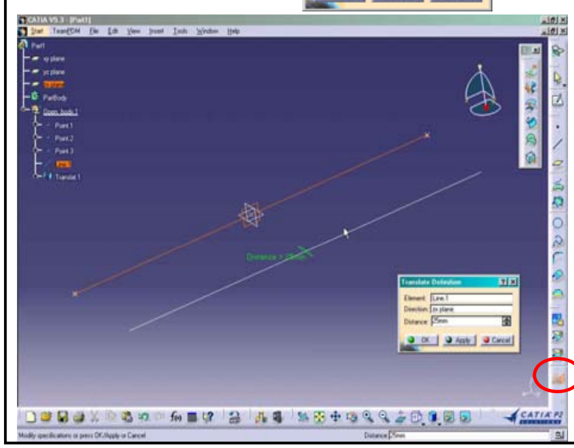


Then place one at x=50 and one at x= -50 so that you are working off of a reference point.

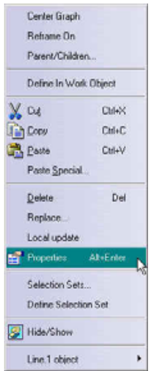
Choose LINE from the toolbar, and the Line Definition dialog box will appear.



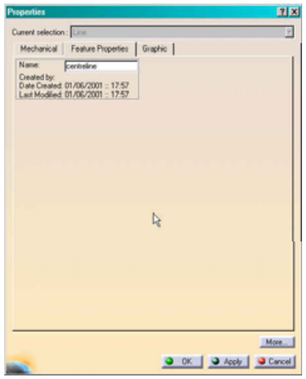
In LINE TYPE, pick Point-Point, then choose the x=50 point as point 1 and x=-50 as point 2. Hit OK.



Now we can copy the newly created line over 25mm. To do this click TRANSLATE from the toolbar, choose the line as ELEMENT, ZX plane as the direction and 25mm as the distance.



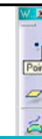
We can now **RENAME** the first line we made as Centreline. To do this, put your pointer on the line to highlight it and right click on it. Then choose **PROPERTIES**.



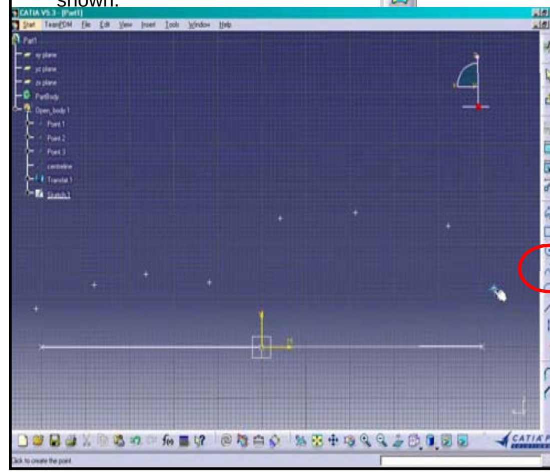
At the **Properties screen**, choose the Feature Properties tab (click on “**more**” if you get the ***short*** dialog box) highlight name be clicking on it and type **centreline**.

Hit OK to return to the work view. Now we can start to make the auto body shape.

Choose the ZX plane and then click on SKETCHER. It is here that we are going to put in the points to define the basic car shape as shown.

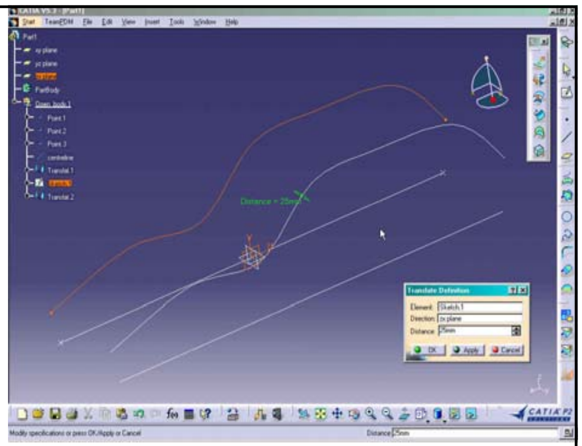


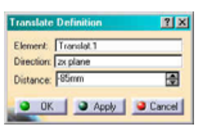
Pick **point** and lay out the points roughly as shown. (*If you **double click** point, it will stay active to do more than one point at a time*).



Now choose **SPLINE** and join all of the points to form one smoothly curved line. This is now your basic car shape.

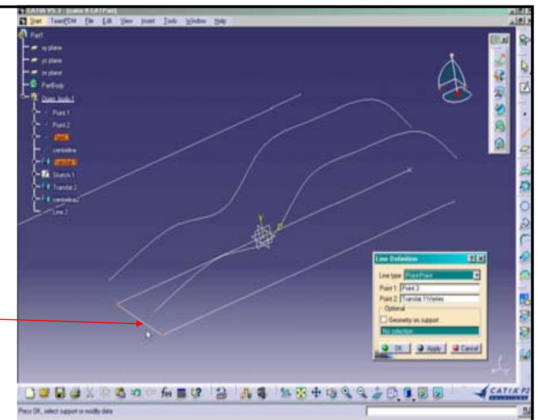
Now we can do a few TRANSLATIONS at the same time. First, click the **TRANSLATE** icon, then Sketch1 is the element, ZX plane is the Direction and distance is 25mm.

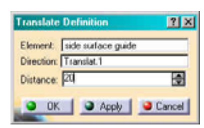




Choose Translate again. This time Translat.1 from the specification tree is the element, ZX plane is again the direction, and the distance is –85mm. Rename this line centeline2.

We now need to add some lines to the front and back of the car to guide the surfaces there. Join the centreline and translat.1 with a line. Rename it **SIDE SURFACE GUIDE**.

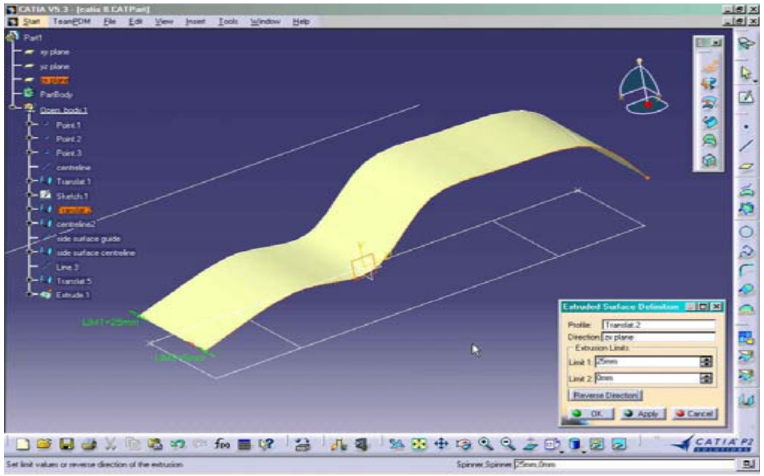




Now TRANSLATE that line 20mm in using Side Surface Guide as the element, Translat.1 As the direction. Rename it Side Surface Centre.

Join the other end of the car with a line, but do not translate it, as we will use Side Surface Centre as the centre of the curved surface.

Now make the first surface using **EXTRUDE**. Click on the icon, in the dialog box pick Translat.2 as Profile. ZX Plane as Direction and extrusion Limit 1 as 25mm. Reverse direction if needed, and click OK. Rename it TOP.



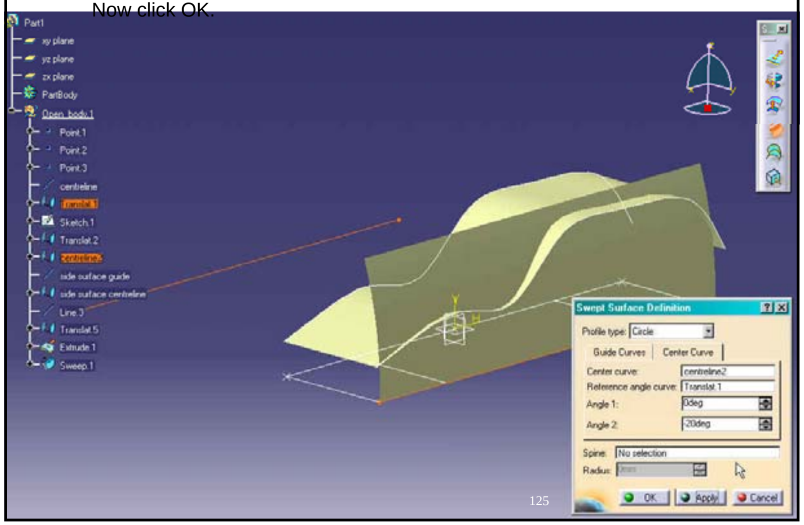
123

Now click the SWEEP icon. Choose CIRCLE as the Profile type.

With the Guide Curve tab picked, choose Translat.1 as the Guide Curve 1.

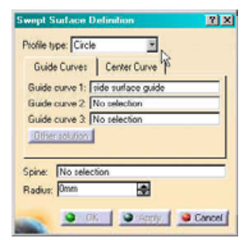
124

Now pick the Center Curve tab. Centreline2 is the Center curve, Translat1 is the Reference angle curve. Enter –20 in the angle 2 area. Now click OK.



125

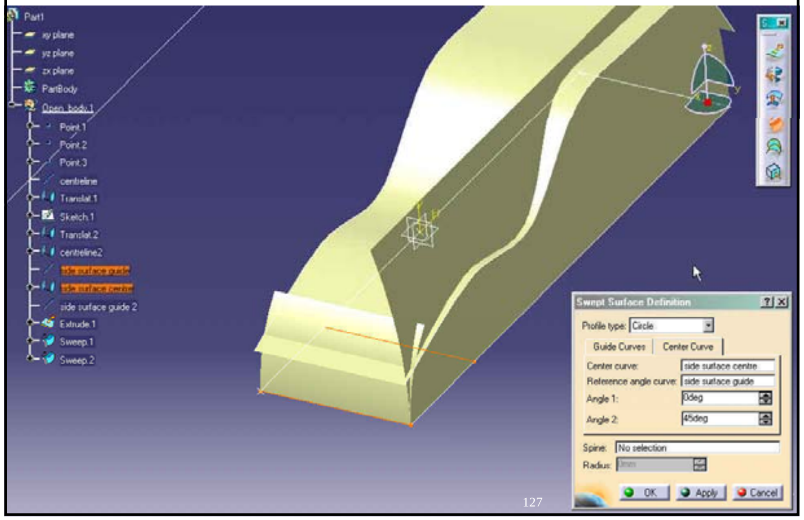
Now we can do the front and back surfaces…



Choose **SWEEP** again and enter the information as done here using the previously renamed lines…

126

Your car should begin to take shape now. We will do the same thing for the back…



127

For the back side, side surface guide 2 will be **Guide Curve** . Notice we are going to rotate this surface around the same center as the front one. Enter the **Center curve** information as shown.

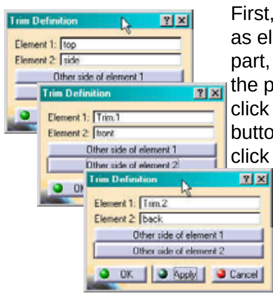
Rename Sweep1 as "**Side**", Sweep2 as "**Front**" and Sweep3 as "**Back**".

Now we can begin to **TRIM** the overlap.

128

This is the TRIM icon. We will use this to trim off the overlapping surfaces.

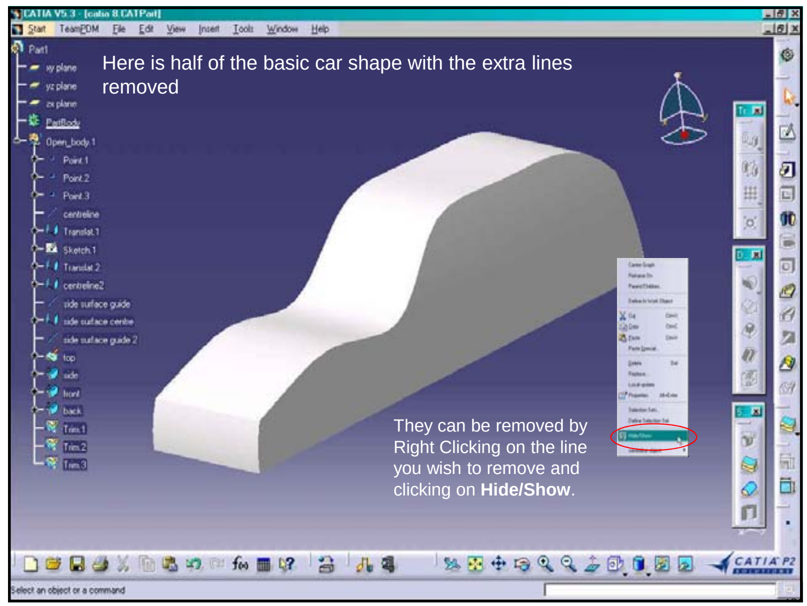
Once you click on the TRIM icon, the TRIM DEFINITION dialog box will appear.



First, choose TOP as Element 1 and SIDE as element 2. You will notice that on your part, an overlap will turn opaque. I this is the part you want to remove, hit OK, if not, click on the "Other side of element 1or 2" button. Once the correct part is opaque, click Apply and OK.

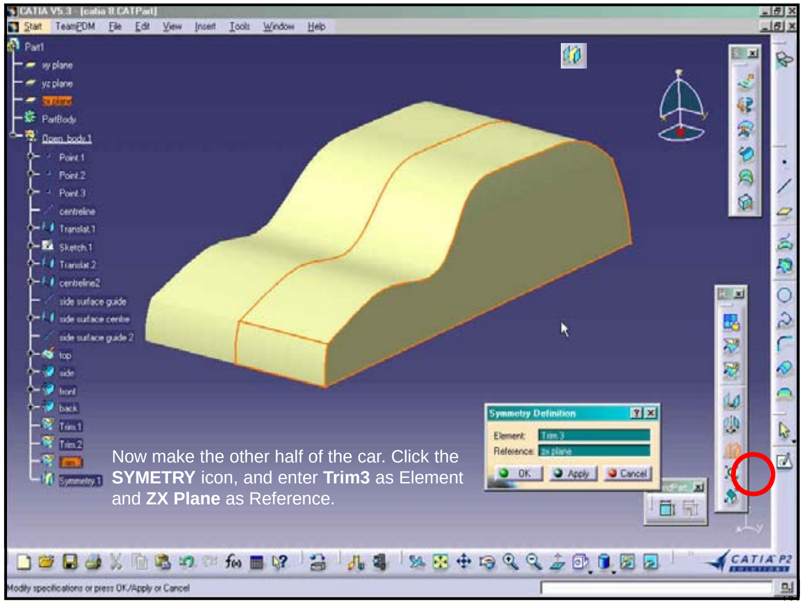
For the other 2 trims, follow what is shown. Trim 1 and Trim 2 can be chosen from the tree, but they are simply the TOP surface, renamed by Catia.

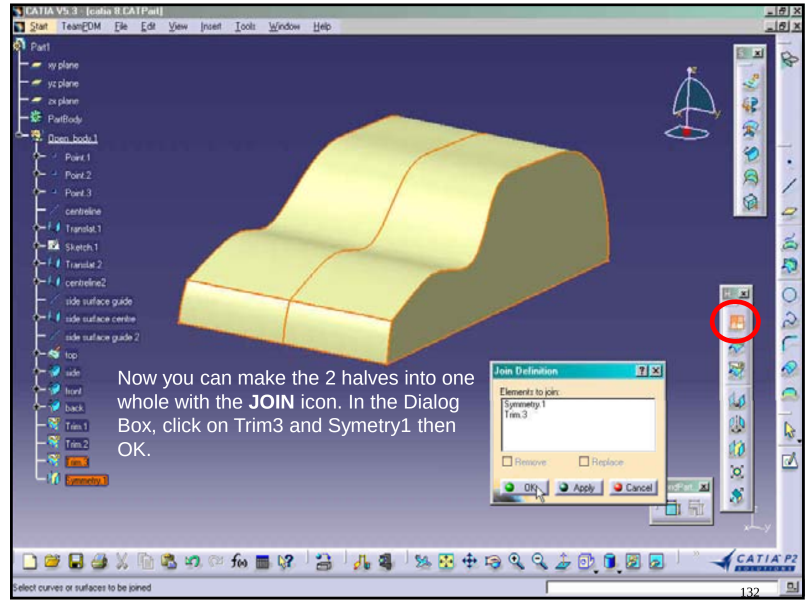
129

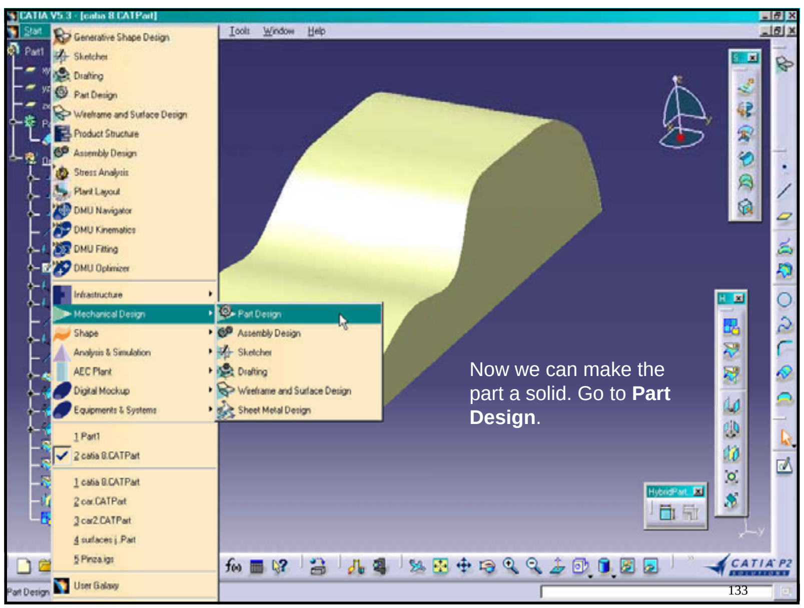


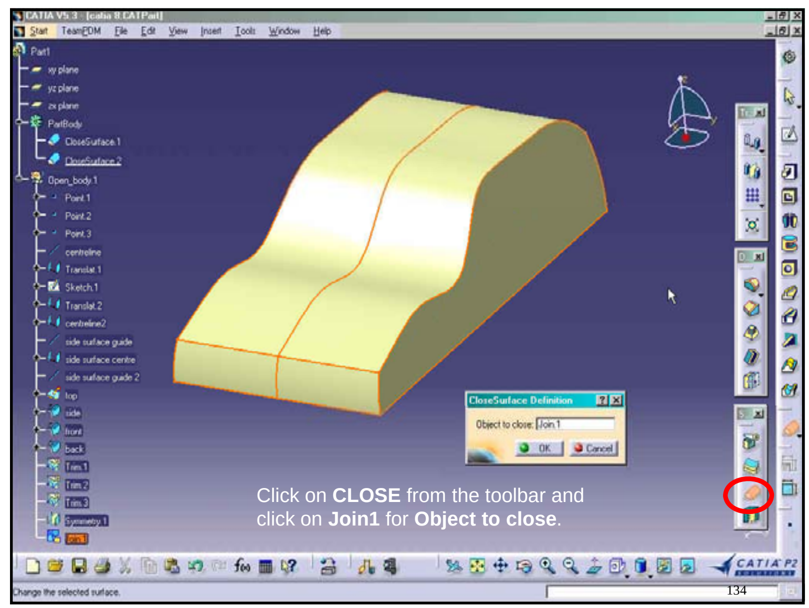
Here is half of the basic car shape with the extra lines removed

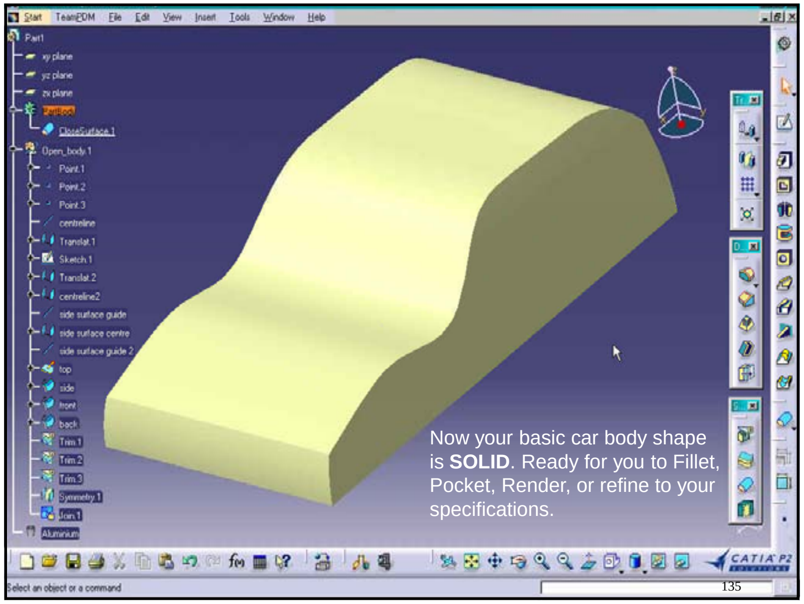
They can be removed by Right Clicking on the line you wish to remove and clicking on **Hide/Show**.


Now make the other half of the car. Click the **SYMETRY** icon, and enter **Trim3** as Element and **ZX Plane** as Reference.


Now you can make the 2 halves into one whole with the **JOIN** icon. In the Dialog Box, click on Trim3 and Symetry1 then OK.
132


Now we can make the part a solid. Go to **Part Design**.
133


Click on **CLOSE** from the toolbar and click on **Join1** for **Object to close**.
134


Now your basic car body shape is **SOLID**. Ready for you to Fillet, Pocket, Render, or refine to your specifications.

Lab: CAD / CAM

Title: NC Manufacturing With Catia V5

Introduction

NC Manufacturing Infrastructure in a Nutshell

NC Manufacturing Infrastructure offers the following main functions:

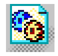
- Common platform for 2.5 to 5-axis axis machining capabilities, which include mill, drill and turn operations
- Management of tools and tool catalogs
- Flexible management of the manufacturing program with intuitive and easy-to-learn user interface based on graphic dialog boxes
- Tight interaction between tool path definition, verification and generation
- Knowledge ware customization facilities through f(x) formula and Edit search facilities
- Seamless NC data generation thanks to an integrated Post Processor Access solution
- Automatic shop floor documentation in HTML format

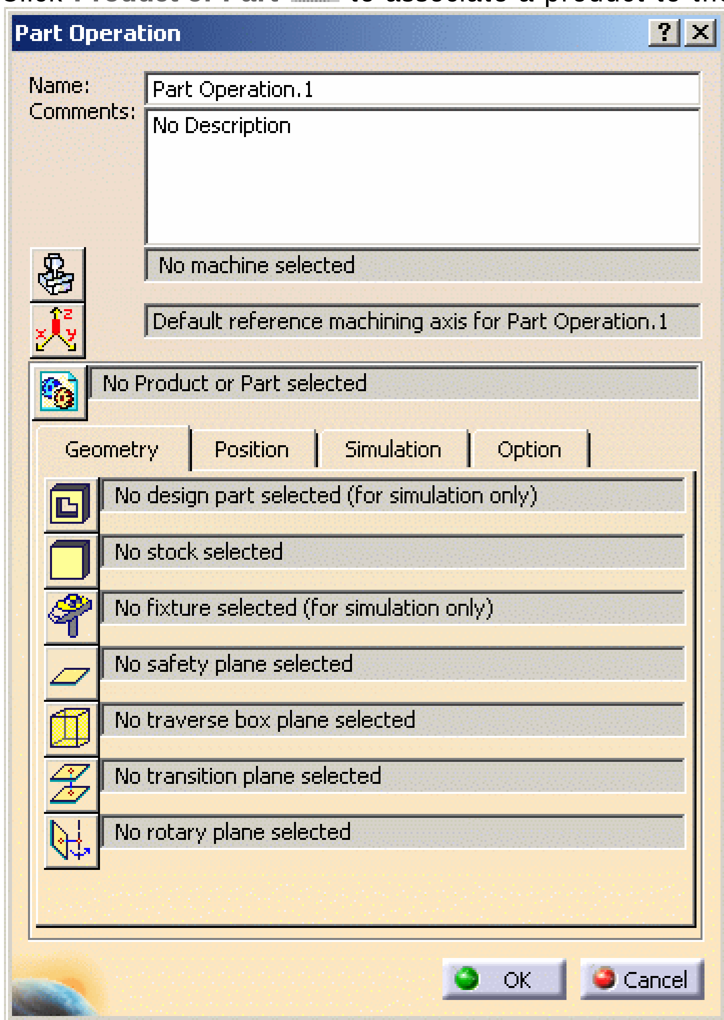
Set Up and Part Positioning

You must create a CAT Product entity for each part set up you want to represent.

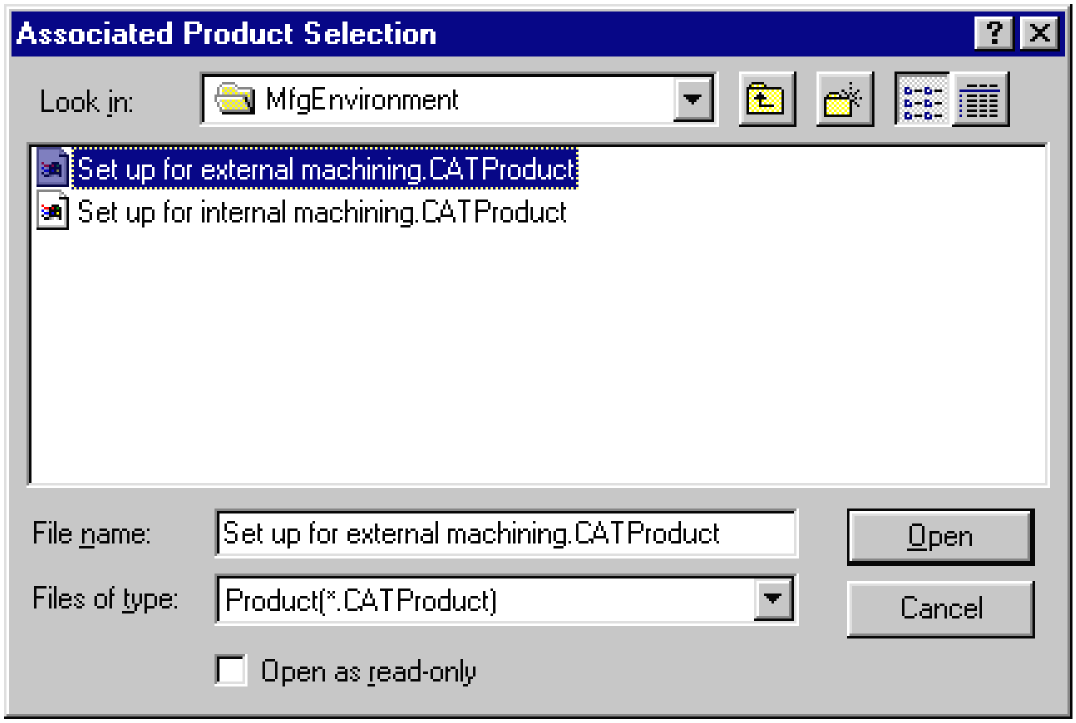
1. Enter a Machining workbench and double-click the Part Operation.1 entity in the tree.

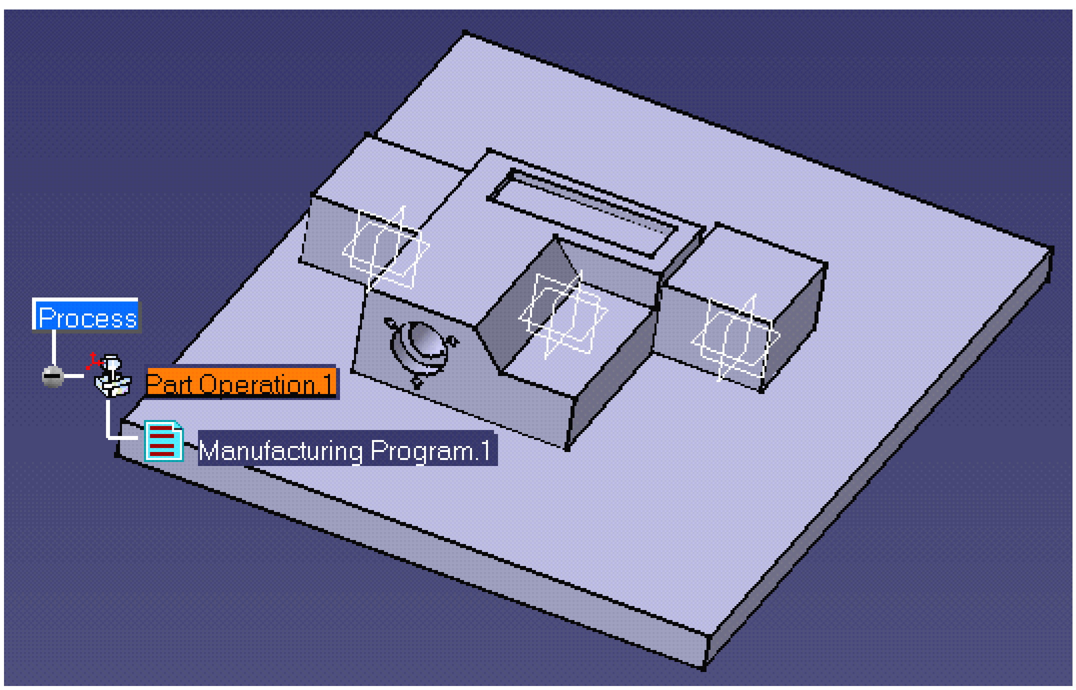
The Part Operation dialog box appears.

2. Click **Product or Part**  to associate a product to the part operation.

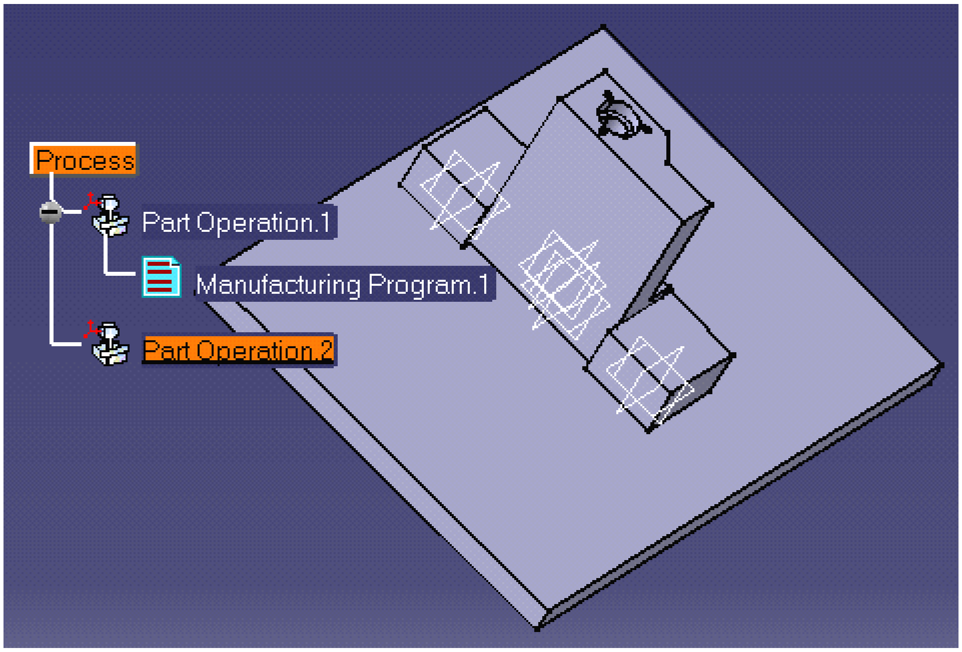


3. Select a CATProduct from the Associated Product list, then click **Open** to display the corresponding part set up.





4. Click OK in the Part Operation dialog box.
5. Click **Part Operation**  to create the Part Operation.2 entity in the tree.
6. Associate another product to Part Operation.2 in the same way as described above.
7. Click OK in the Part Operation dialog box.

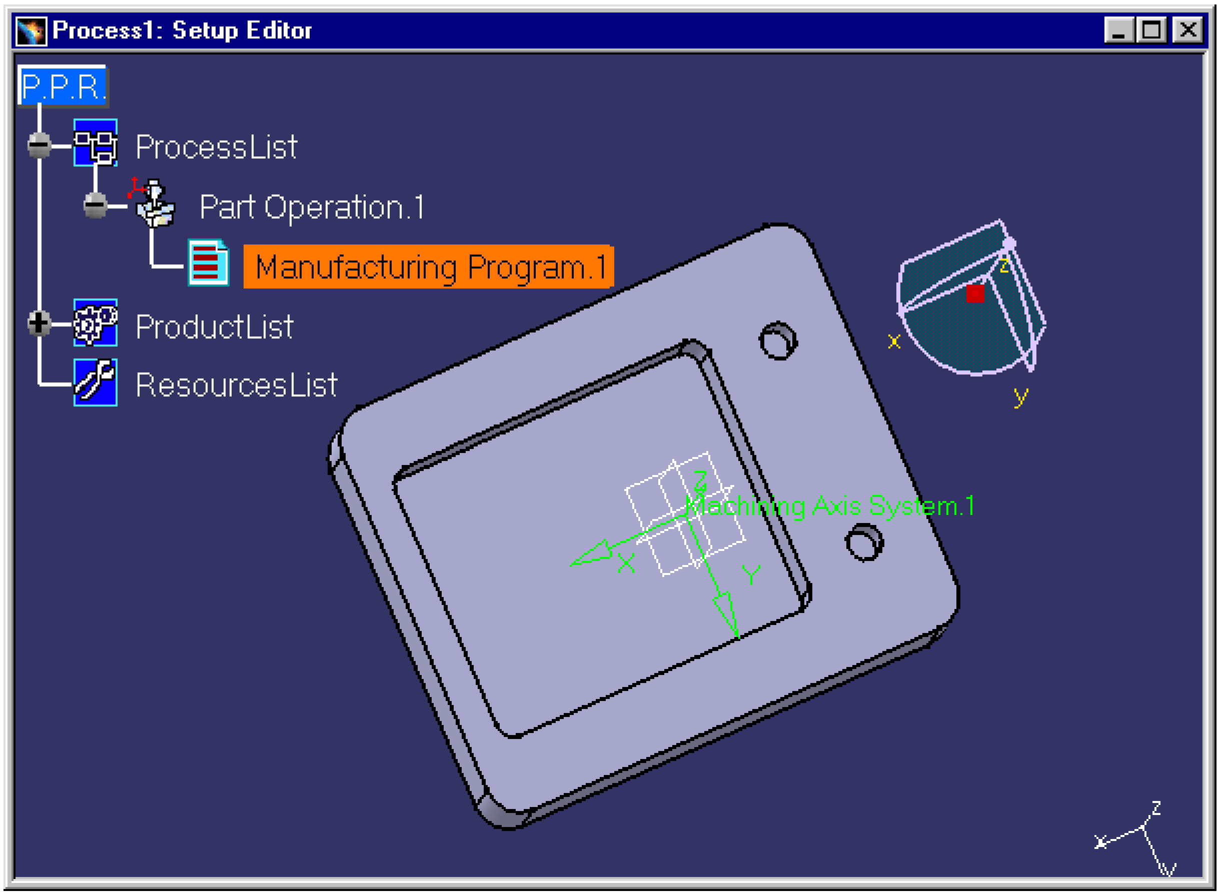


To display the desired part set up, just select the corresponding Part Operation in the tree

Manage Workbenches

1 Select File > Open then select the desired CATPart document.

2 Select Machining > Prismatic Machining from the Start menu. The Prismatic Machining workbench appears. The part is displayed in the Setup Editor window along with the PPR specification tree.



The CATPart is automatically associated to the Part Operation and an instance of the part is created in the Product List.

3 Select Manufacturing Program.1 in the tree to make it the current entity.

To insert program entities such as machining operations, tools and auxiliary commands you can either:

- make the program current before clicking the *insert program entity* command
- click the *insert program entity* command then make the program current.

4 Double-click the Part entity in the tree to switch to a Mechanical Design workbench (such as Part Design or Wireframe and Surface Design depending on your configuration).

5 Double-click a Machining entity in the tree to switch back to the Machining workbench

Prismatic Machining in a Nutshell

Prismatic Machining enables you to define and manage NC programs dedicated to machining parts designed in 3D wireframe or solids geometry using 2.5 axis machining techniques.

It offers an easy-to-use and easy-to-learn graphic interface that makes it suitable for shop floor-oriented use. Moreover, its leading edge technologies together with a tight integration with Version 5 design methodologies and DELMIA's digital manufacturing environment, fully satisfy the requirements of office programming. Prismatic Machining is a unique solution that reconciliates office and shop floor activities.

It is integrated to a Post Processor Access execution engine, allowing the product to cover the whole manufacturing process from tool trajectory (APT source) to NC code.

This product is particularly adapted for tooling and simple machined parts, and is also an ideal complement to other manufacturing applications.

Prismatic Machining offers the following main functions:

- 2.5 axis milling and drilling capabilities
- Management of tools and tool catalogs
- Flexible management of the manufacturing program with intuitive and easy-to-learn user interface based on graphic dialog boxes
- Tight interaction between tool path definition, verification and generation
- Seamless NC data generation thanks to an integrated Post Processor Access solution
- Automatic shop floor documentation in HTML format

Step 01: Enter the Workbench

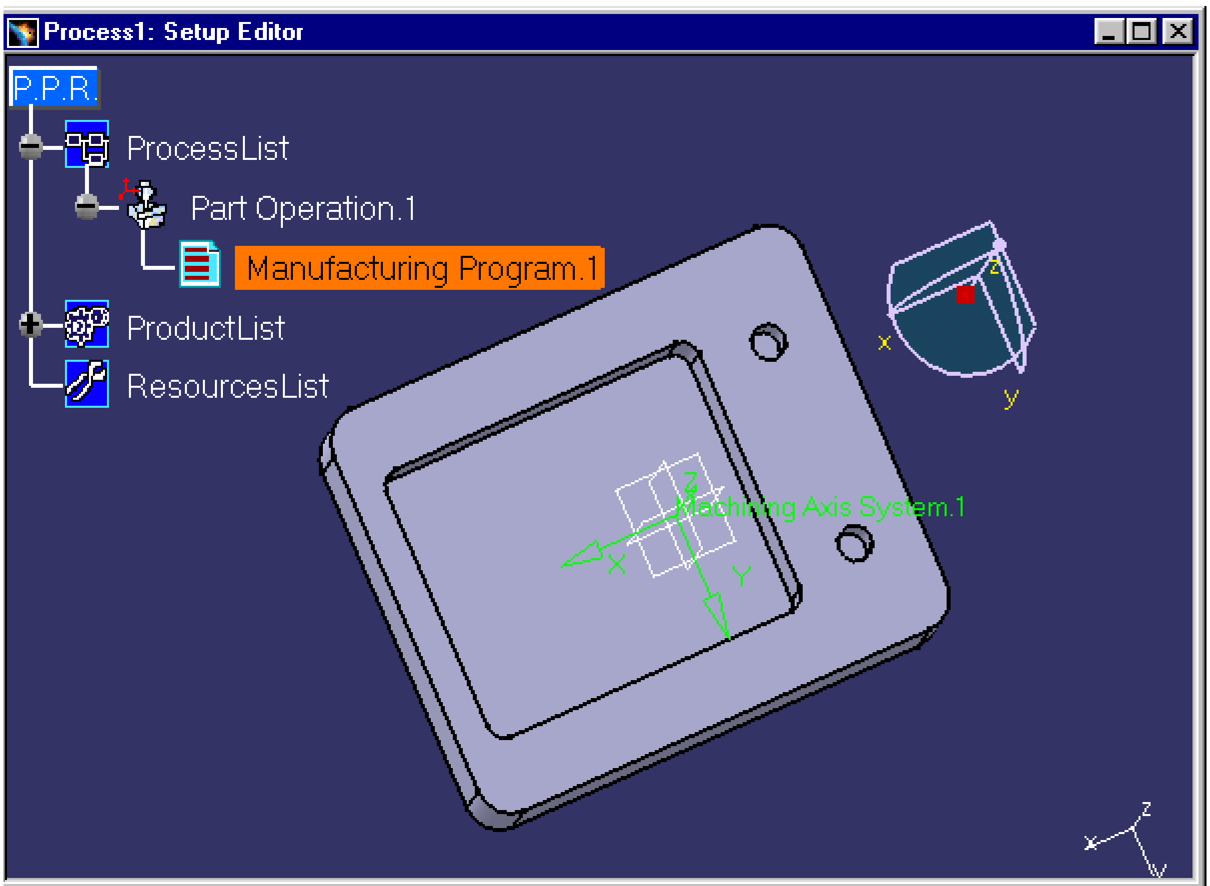
This first task shows you how to open a part and enter the Prismatic Machining workbench.

1. Select **File > Open** then select the GettingStartedPrismaticMachining.CATPart document.

2. Select **Machining > Prismatic Machining** from the Start menu.

   The Prismatic Machining workbench appears.

   The part is displayed in the Setup Editor window along with the manufacturing specification tree.



3. Select Manufacturing Program.1 in the tree to make it the current entity.
   To insert program entities such as machining operations, tools and auxiliary commands you can either:

   - Make the program current before clicking the *insert program entity* command
   - Click the *insert program entity* command then make the program current.

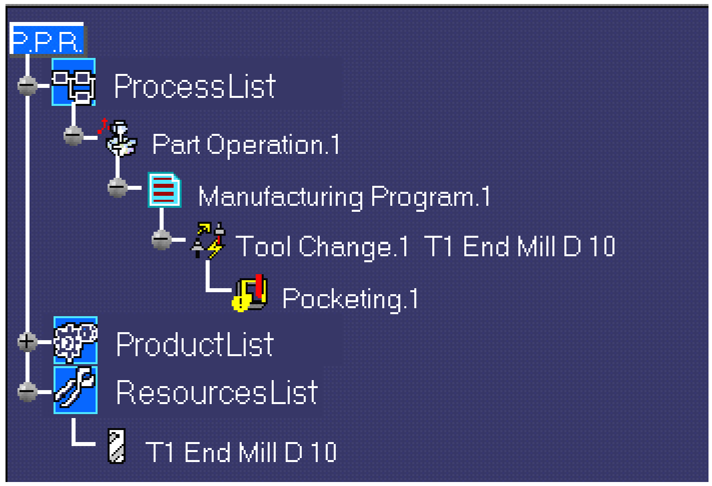
Step: 02 Create a Pocketing Operation

This task shows you how to insert a pocketing operation in the program.

As this operation will use the default tool and options proposed by the program, you just need to specify the geometry to be machined.

1. Select **Pocketing**  in the Machining Operations toolbar.

A Pocketing.1 entity along with a default tool is added to the program.

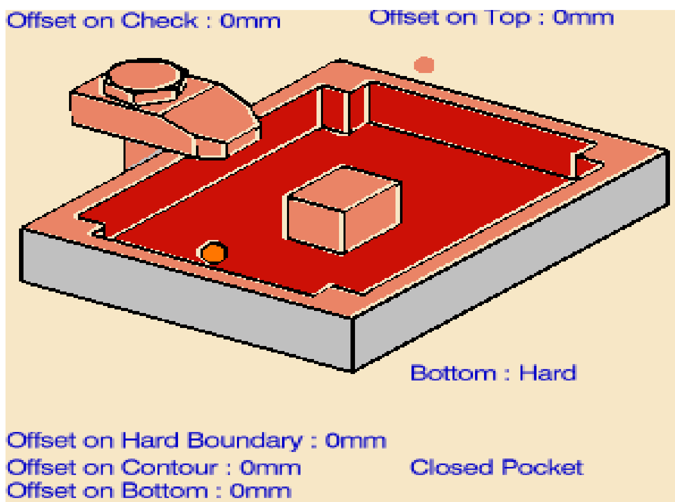


The Pocketing dialog box appears directly at the**Geometry** tab page .

The red status light on the tab indicates that you must select the pocket geometry in order to create the operation.

The Geometry page includes an icon representing a simple pocket.

There are several sensitive areas and texts in the icon to help you specify the pocket geometry. Sensitive areas that are colored red indicate required geometry.



Make sure that the Pocketing style is set to **Closed Pocket**. Click on the **Open Pocket** text if this is not the case.

2. Right-click the red Bottom in the icon and select**Contour Detection**.

   Click the red Bottom area.

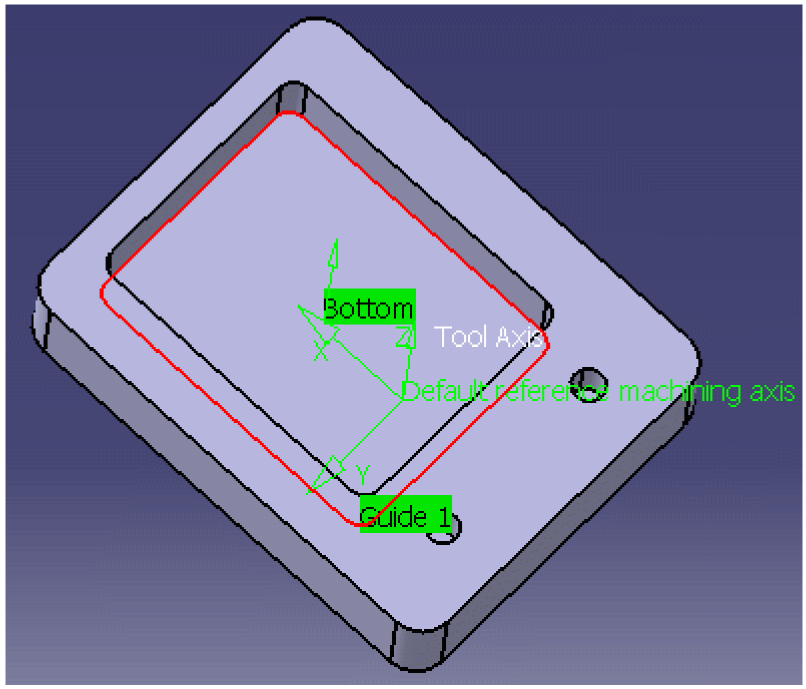
   The dialog box is reduced allowing you to select the corresponding part geometry.

3. Select the bottom of the pocket.

   The boundary of the selected pocket bottom is automatically proposed as guiding element for the operation thanks to the Contour Detection setting

   The dialog box reappears.

   The bottom and sides of the pocket in the icon are now colored green, indicating that the corresponding geometry is defined for the operation. The tab status is now green .



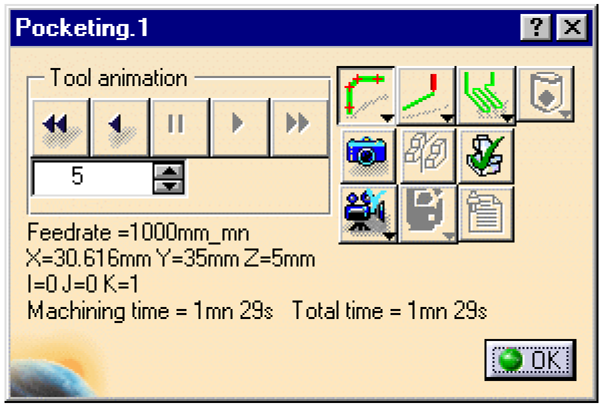
4. Click **OK** to create the operation.

Step: 03 Replay the Tool Path

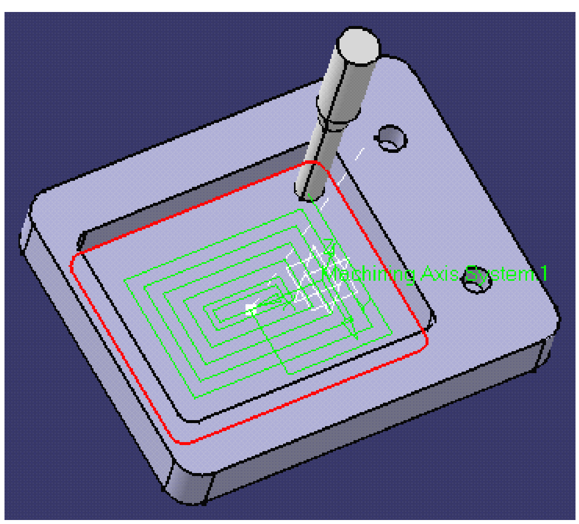
This task shows you how to replay the tool path of the pocketing operation.

1. Select the pocketing operation in the tree then select **Replay Tool Path** .

   The tool path is computed and the Replay dialog box appears.
2. Choose a **Point by Point** replay of the tool path.



   You can set the number of computed points to be replayed at each step of the verification by means of the spinner.
3. Click rewind button to position the tool at the start point of the operation.
4. Click the play button to start the replay and continue to click that button to move the tool along the computed trajectory.
5. Click **OK** to quit the replay mode.



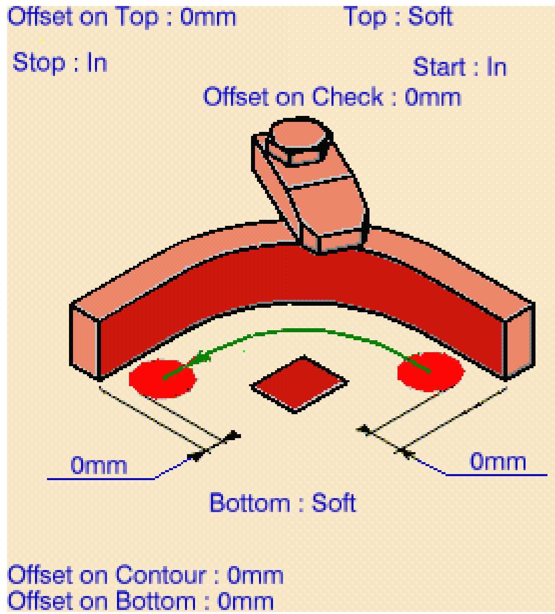
Step: 04 Create a Profile Contouring Operation

This task shows you how to insert a profile contouring operation in the program.

Make sure that the pocketing operation is the current entity in the program.

1. Select **Profile Contouring**.
   The Profile Contouring dialog box appears directly at the **Geometry**page.

   Make sure the Contouring mode is set to **Between Two Planes**.



2. Click the **Bottom: Hard** text in the sensitive icon to switch the type of bottom to Soft.
3. Click the Bottom plane then select the corresponding part geometry (that is, the underside of the part).

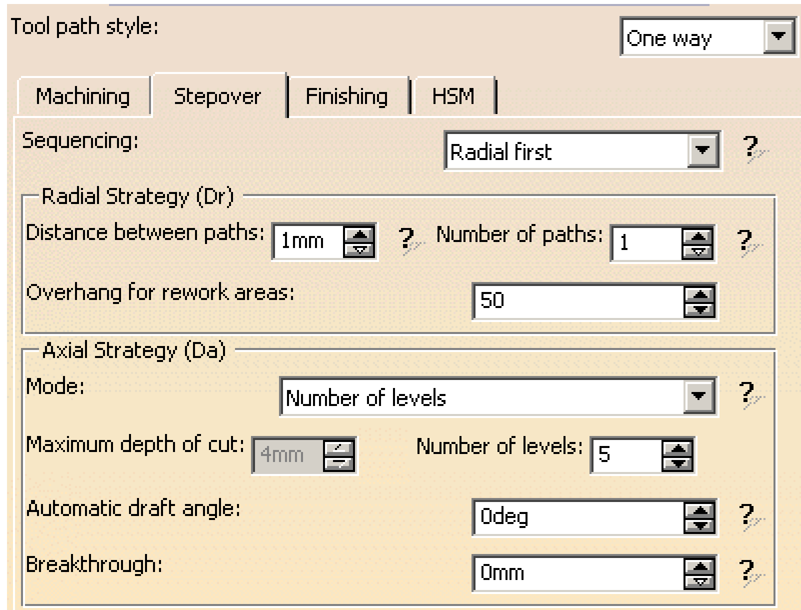
   The closed external contour of the bottom is proposed as Guide element for the operation.

Make sure that the arrow on the Guide element is pointing away from the part.

4. Click the Top plane in the icon, then select the corresponding part geometry.
5. Double click **Offset on Contour** in the icon.

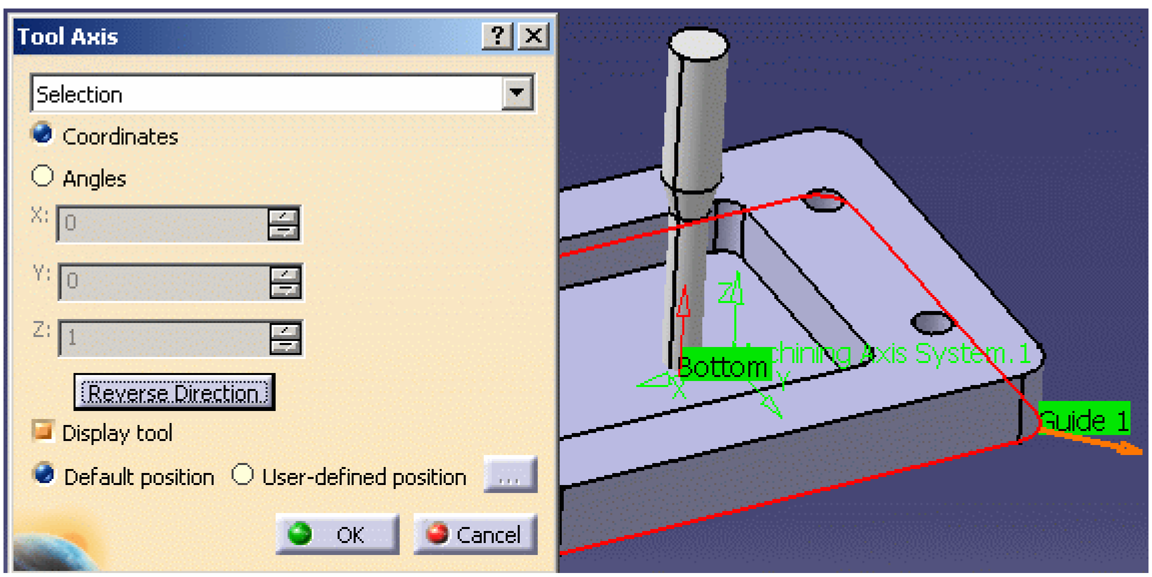
   Set this value to 1mm in the Edit Parameter dialog box and click **OK**.

6. Select the **Strategy** tab page  and set the parameters as shown.

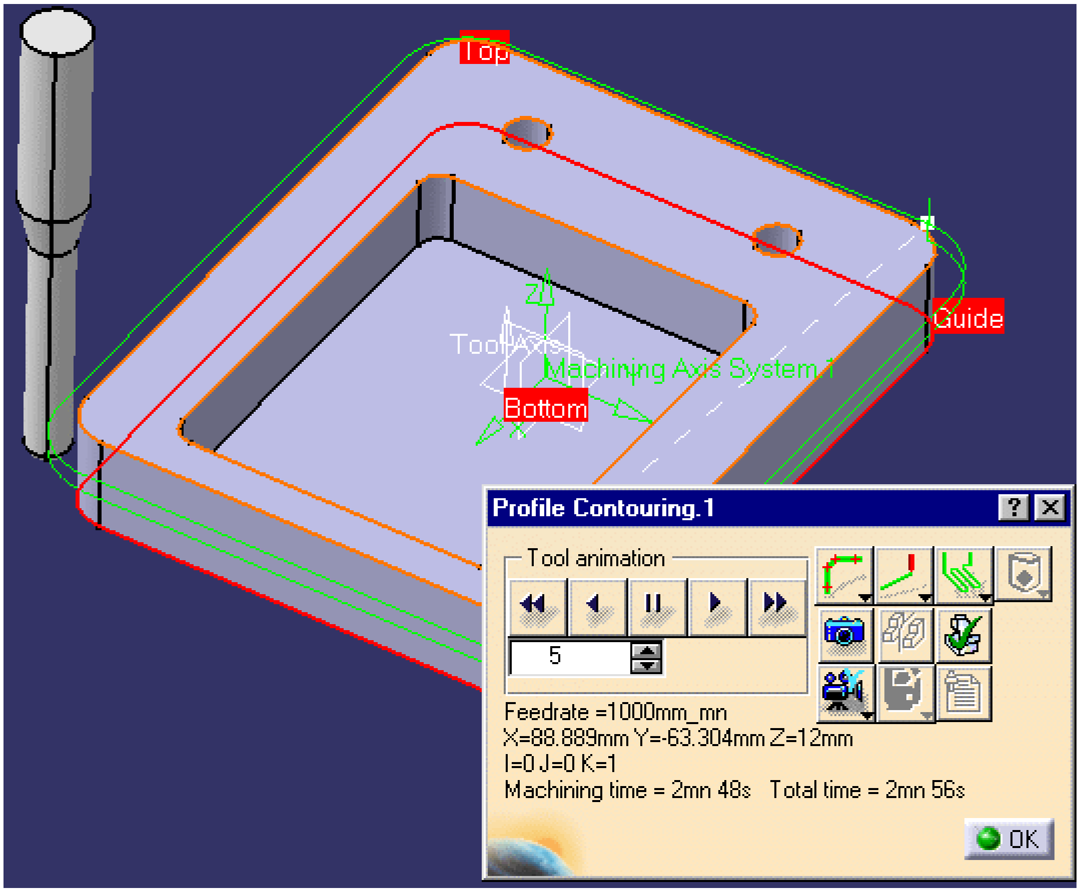


7. If needed, you can change the tool axis orientation. Just click the Tool Axis symbol then click the **Reverse Direction** button in the Tool Axis dialog box.

   You can display the tool with the specified orientation by selecting the Display tool checkbox.



8. Click **Preview** in the dialog box to request that the program verifies the compatibility of the selected tool, geometry and machining parameters. A message box appears giving feedback about this verification.
9. Click **Replay** in the dialog box to visually check the operation's tool path.

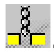


At the end of the replay click **OK** to return to the Profile Contouring dialog box.

10. Click **OK** to create the operation.

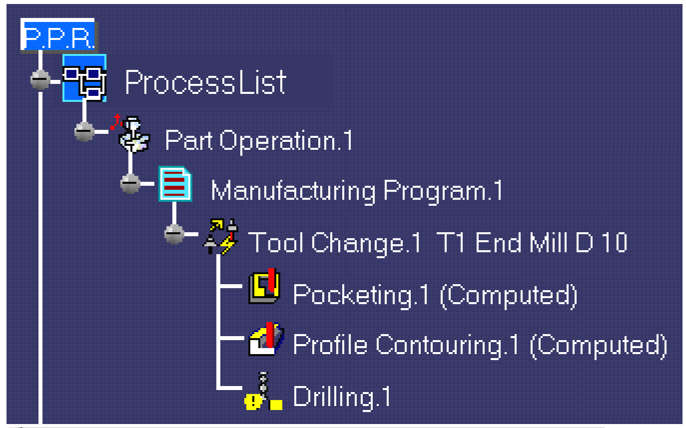
Step:5 Create a Drilling Operation

This task shows you how to insert a drilling operation in the program.

1. Select **Drilling** .

   The Drilling dialog box appears directly at the **Geometry** page .

   The program is updated to include a Drilling operation.



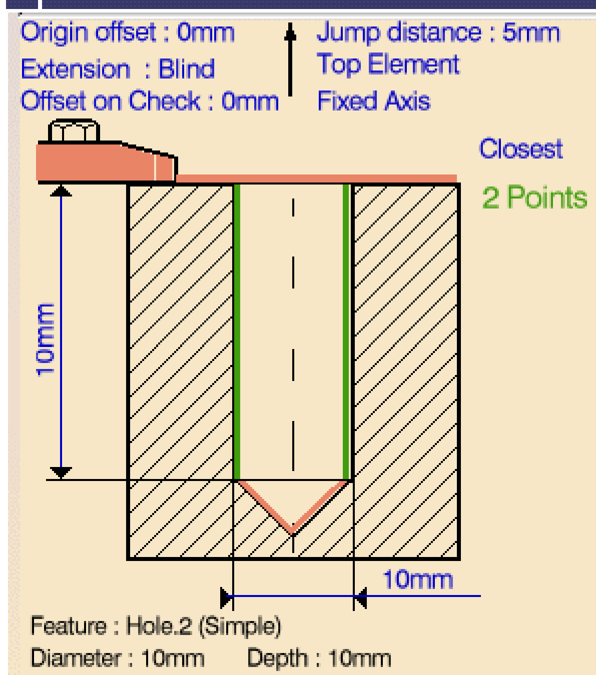
   The Drilling dialog box appears.

2. Select the red hole depth representation in the sensitive icon.

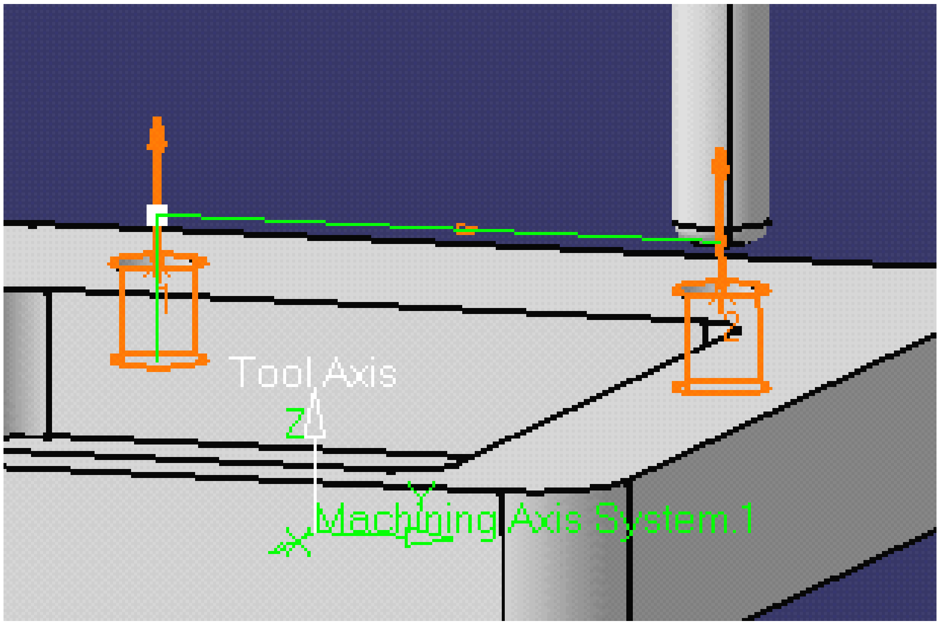
   The Pattern Selection dialog box appears to help you specify the pattern of holes to be machined.
3. Select the cylindrical feature of the first hole.
4. Select the second hole feature, then double click to end hole selection.

   The Drilling dialog box replaces the Pattern Selection dialog box.

   The icon is updated with geometric information about the first selected hole of the pattern.



5. Double click **Jump distance** in the sensitive icon, then enter a value of 5mm in the Edit Parameter dialog box.
6. Click **Preview** in the dialog box to request that the program verifies the compatibility of the selected tool, geometry and machining parameters. A message box appears giving feedback about this verification.
7. Click **Replay** to replay the operation as described previously.



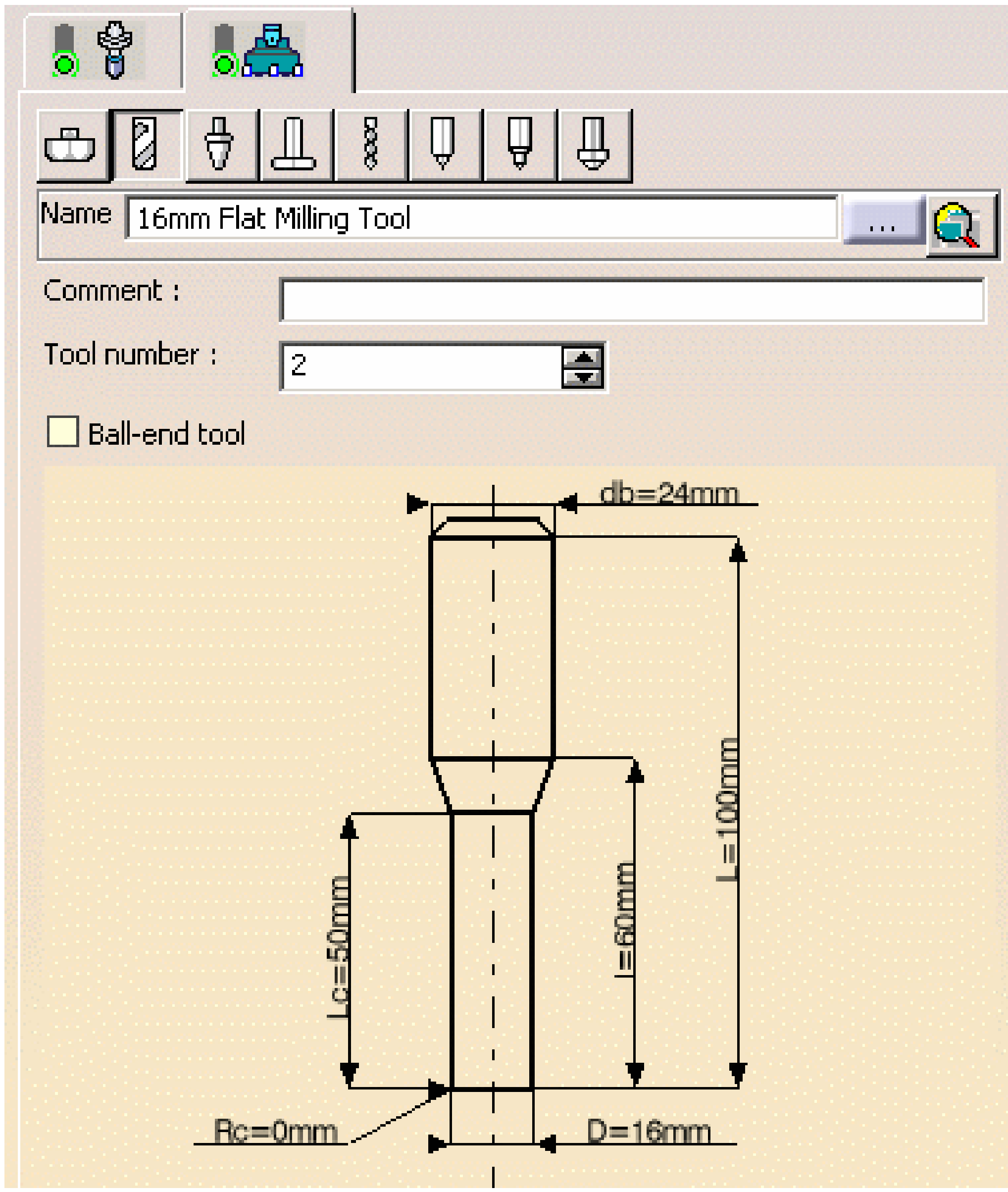
Click **OK** to return to the Drilling dialog box.

8. Click **OK** to create the Drilling operation in the program.

Step:6 Assign a Tool

This task shows you how to assign another tool to an operation.

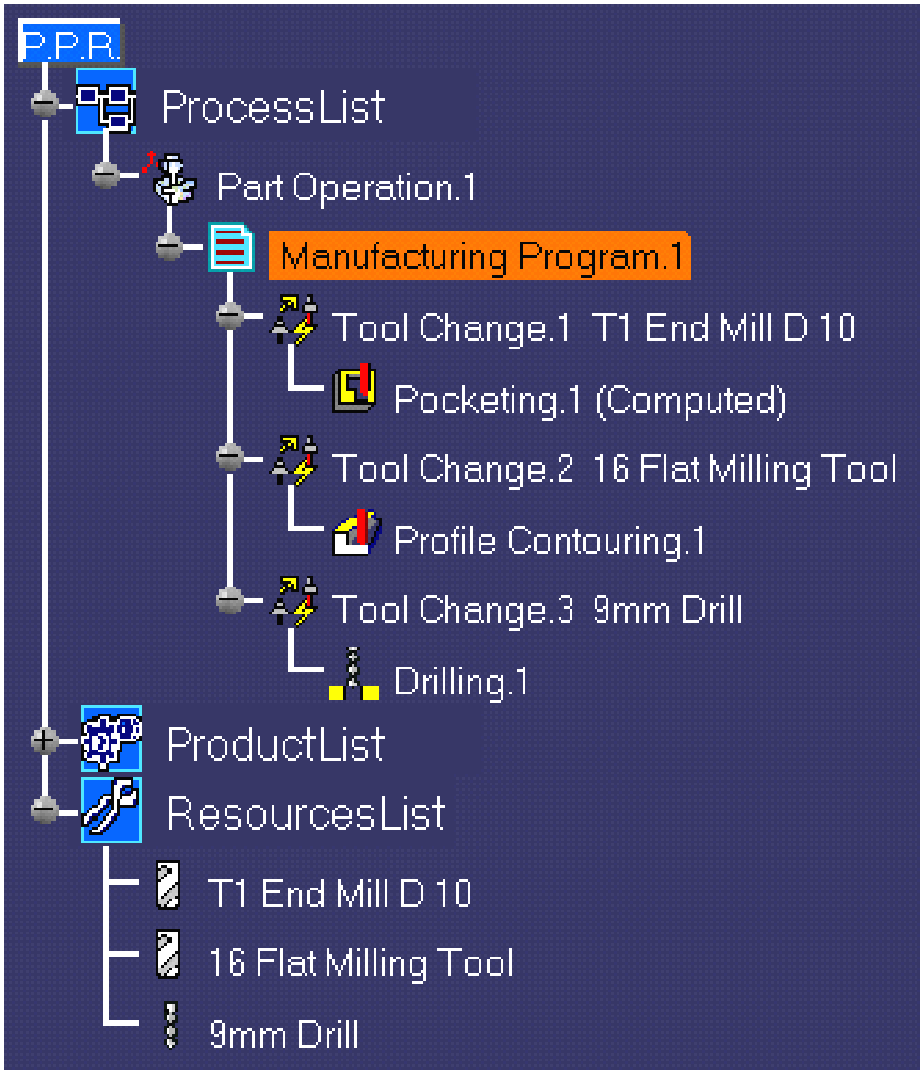
1. Double-click the Profile Contouring operation in the program, then in the **Tooling** tab page , select the **Tool** tab .
2. Enter a name of the new tool (for example, 16mm Flat Milling Tool).



3. Double-click **D** (nominal diameter) in the sensitive icon, then enter 16mm in the Edit Parameter dialog box. The tool icon is updated to take the new value into account.
4. Double-click **Rc** (corner radius) in the icon, then enter 0mm in the Edit Parameter dialog box.

   Set **db** (body diameter) to 24mm in the same way.

   The Tool number is set to 2.



5. Click **OK** to accept the new tool.
The program is automatically updated.

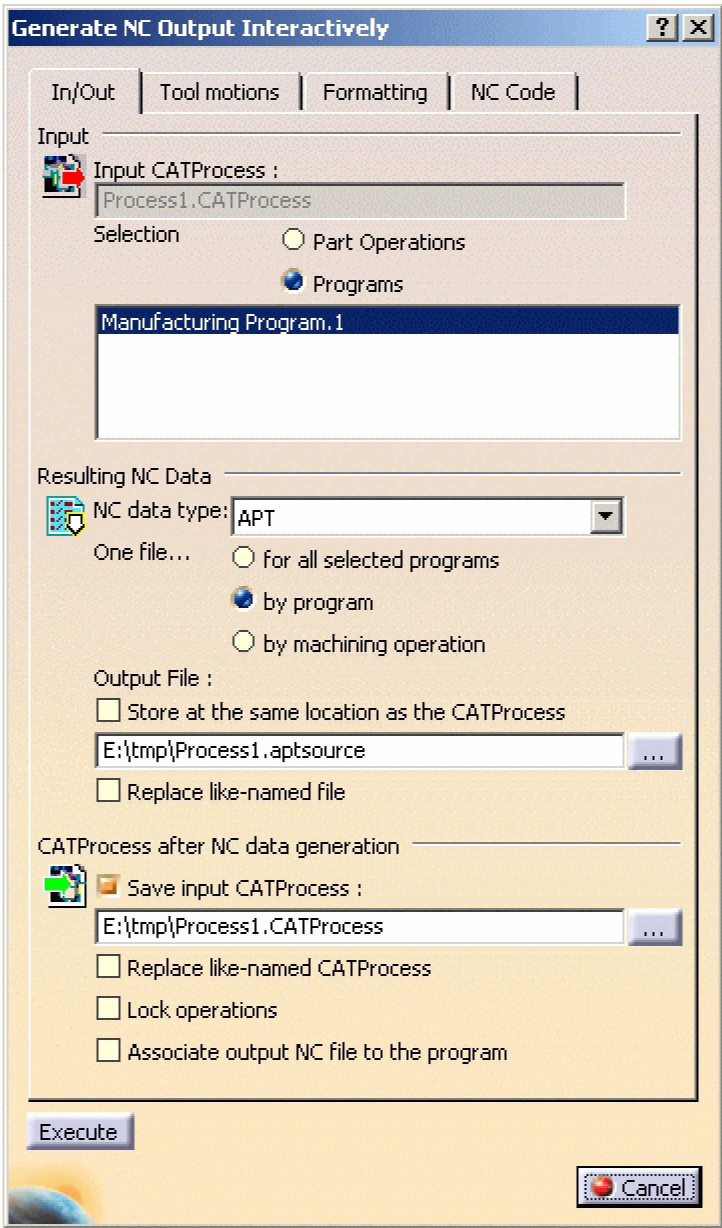
You can modify the tools of the other operations in the same way. For example, you may want to replace the End Mill by a Drill in the Drilling operation.

Step:7 Generate NC Code

This task shows you how to generate NC data in APT format from the program.

For more information about this procedure please refer to Program Output.

1. Right-click the Manufacturing Program.1 entity in the tree and select **Manufacturing Program.1 object > Generate NC Code Interactively**. The Generate NC Output Interactively dialog box appears.



2. Select APT as the desired NC data type.
3. Click **Output File** to select the folder where you want the file to be saved and specify the name of the file.

4. Click **Execute** to generate the APT source file.

An extract from a typical APT source file is given below.

```
$$ -----------------------------------------------------------------
$$ Generated on Thursday, May 10, 2001 04:58:20 PM
$$ -----------------------------------------------------------------
$$ Manufacturing Program.1
$$ Part Operation.1
$$*CATIA0
$$ Manufacturing Program.1
$$ 1.00000 0.00000 0.00000 0.00000
$$ 0.00000 1.00000 0.00000 0.00000
$$ 0.00000 0.00000 1.00000 0.00000
PARTNO PART TO BE MACHINED
COOLNT/ON
CUTCOM/OFF
PPRINT OPERATION NAME : Tool Change.1
$$ Start generation of : Tool Change.1
TLAXIS/ 0.000000, 0.000000, 1.000000
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 100.00000
CUTTER/ 10.000000, 2.000000, 3.000000, 2.000000, 0.000000$
, 0.000000, 100.000000
TOOLNO/1, 10.000000
TPRINT/T1 End Mill D 10
LOADTL/1
$$ End of generation of : Tool Change.1
PPRINT OPERATION NAME : Pocketing.1
$$ Start generation of : Pocketing.1
FEDRAT/ 1000.0000,MMPM
SPINDL/ 70.0000,RPM,CLW
GOTO / 30.61644, 2.50000, 5.00000
GOTO / 17.50000, 2.50000, 5.00000
...
GOTO / 30.61644, 35.00000, 5.00000
$$ End of generation of : Pocketing.1
PPRINT OPERATION NAME : Tool Change.2
$$ Start generation of : Tool Change.2
$$ TOOLCHANGEBEGINNING
RAPID
GOTO / 0.00000, 0.00000, 100.00000
CUTTER/ 16.000000, 0.000000, 8.000000, 0.000000, 0.000000$
, 0.000000, 100.000000
TOOLNO/2, 16.000000
TPRINT/16 Flat Milling Tool
LOADTL/2
$$ End of generation of : Tool Change.2
PPRINT OPERATION NAME : Profile Contouring.1
$$ Start generation of : Profile Contouring.1
FEDRAT/ 300.0000,MMPM
SPINDL/ 70.0000,RPM,CLW
GOTO / -69.00000, 40.00000, 46.00000
...
GOTO / -69.00000, 50.00000, 0.00000
$$ End of generation of : Profile Contouring.1
PPRINT OPERATION NAME : Tool Change.3
$$ Start generation of : Tool Change.3
$$ TOOLCHANGEBEGINNING
RAPID
```

```
GOTO / 0.00000, 0.00000, 100.00000
CUTTER/ 9.000000, 0.000000, 4.500000, 2.598076, 30.000000$
, 0.000000, 100.000000
TOOLNO/3, 9.000000
TPRINT/9mm Drill
LOADTL/3
$$ End of generation of : Tool Change.3
PPRINT OPERATION NAME : Drilling.1
$$ Start generation of : Drilling.1
LOADTL/3,1
SPINDL/ 70.0000,RPM,CLW
RAPID
GOTO / -40.00000, -30.00000, 25.00000
RAPID
GOTO / -40.00000, -30.00000, 21.00000
CYCLE/DRILL, 10.000000, 1.000000, 1000.000000,MMPM
GOTO / -40.00000, -30.00000, 20.00000
GOTO / -40.00000, 30.00000, 20.00000
CYCLE/OFF
RAPID
GOTO / -40.00000, 30.00000, 21.00000
RAPID
GOTO / -40.00000, 30.00000, 25.00000
$$ End of generation of : Drilling.1
SPINDL/OFF
REWIND/0
END
```