

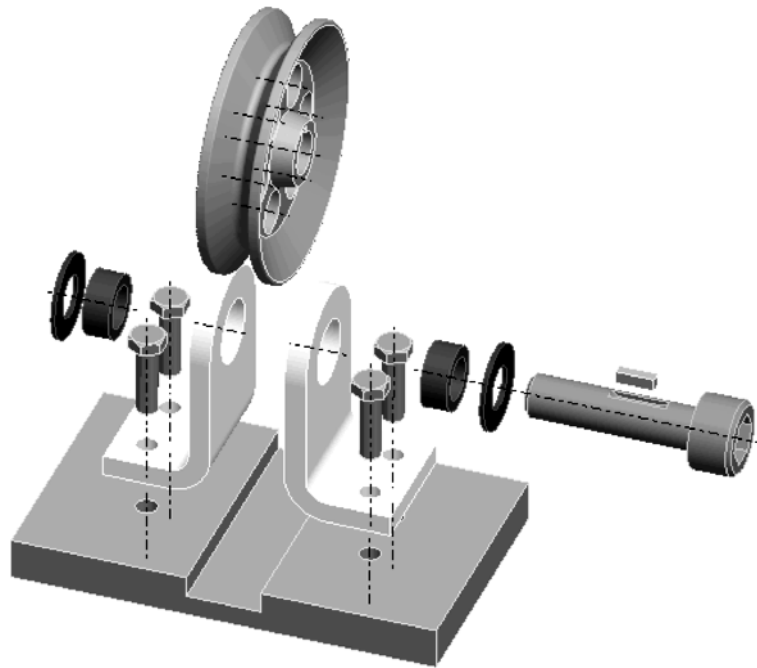
INSIDE:



Pro | ENGINEER[®] WILDFIRE[™] 3.0

Tutorial and MultiMedia CD

- ▣ LESS9.ASM
 - ▣ ASM_RIGHT
 - ▣ ASM_TOP
 - ▣ ASM_FRONT
 - ▣ BPLATE.PRT
 - ▣ SUPPORT.ASM
 - ▣ LBRACK.PRT
 - ▣ BUSHING.PRT
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 - ▣ SUPPORT.ASM
 - ▣ AXLE.PRT
 - ▣ PULLEY.PRT
 - ▣ Group BOLTS
 - ▣ Extrude 1
 - ▣ KEY.PRT
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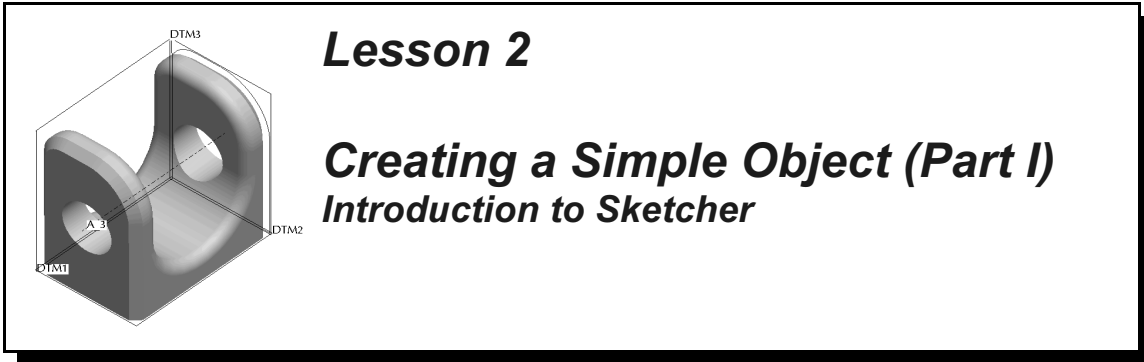
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Synopsis

Creating a part; introduction to Sketcher; Sketcher constraints; creating datum curves, protrusions, cuts; using the dashboard; saving a part; part templates.

Overview of this Lesson

The main objective of this lesson is to introduce you to the general procedures for creating sketched features. We will go at quite a slow pace and the part will be quite simple (see Figure 1 on the next page), but the central ideas need to be elaborated and emphasized so that they are very clearly understood. Some of the material presented here is a repeat of the previous lesson - take this as an indication that it is important! Here's what we are going to cover:

1. Creating a Simple Part
2. Feature Types and Menus
3. Introduction to Sketcher
 - ▶ Sketcher menus
 - ▶ Intent Manager and Sketcher constraints
4. Creating a Datum Curve
5. Creating an Extruded Protrusion
 - ▶ Using the Dashboard
6. Creating an Extruded Cut
7. Saving the part
8. Using Part Templates

It will be a good idea to browse ahead through each section to get a feel for the direction we are going, before you do the lesson in detail. There is a lot of material here which you probably won't be able to absorb with a single pass-through.

Start Pro/E as usual. If it is already up, close all windows (except the base window) and erase all objects in session using **File > Erase > Current** and **File > Erase > Not Displayed**. Close the Navigator and Browser windows.

Creating a Simple Part

In this lesson, we will create a simple block with a U-shaped central slot. By the end of the lesson your part should look like Figure 1 below. This doesn't seem like such a difficult part, but we are going to cover a few very important and fundamental concepts in some depth. Try not to go through this too fast, since the material is crucial to your understanding of how Pro/E works. We will be adding some additional features to this part in the next lesson.

We are going to turn off some of the default actions of Pro/E. This will require us to do some things manually instead of letting the program do them automatically. This should give you a better understanding of what the many default actions are. Furthermore, eventually you will come across situations where you don't want the defaults and you'll need to know your way around these options.

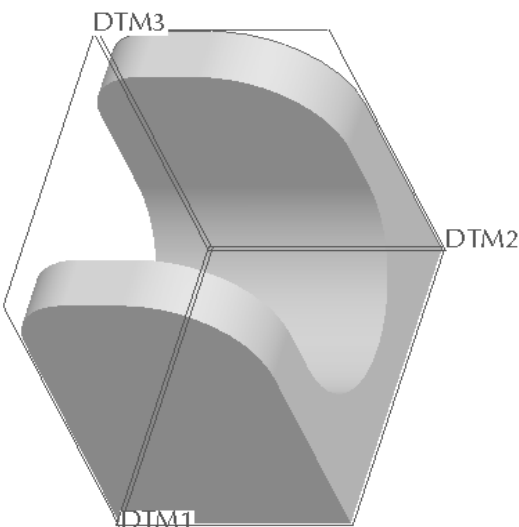


Figure 1 Part at the end of this lesson

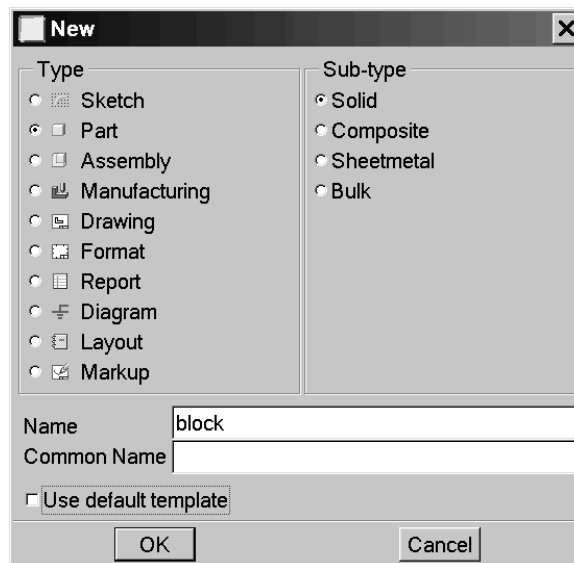


Figure 2 Creating a new part

Creating and Naming the Part

Click the **Create New Object** short-cut button, or select **File > New**. A window will open (Figure 2) showing a list of different types and sub-types of objects to create (parts, assemblies, drawings, and so on). In this lesson we are going to make a single solid object called a *part*. Keep the default radio button settings

Part | Solid

IMPORTANT: Turn off (remove the check) the **Use Default Template** option at the bottom. We will discuss templates at the end of this lesson.

Many parts, assemblies, drawings, etc. can be loaded simultaneously (given sufficient

computer memory) in the current session. All objects are identified by unique names¹. A default name for the new part is presented at the bottom of the window, something like **[PRT0001]**. It is almost always better to have a more descriptive name. So, double click (left mouse) on this text to highlight it and then type in

[block]

(without the square brackets) as your part name. The **Common Name** of the part is an option for specifying an even more descriptive name. For example, you might have a number of part files named using a part or catalog number such as “TG123_A29”. This is not very descriptive, so you could enter a common name such as “small flat rubber washer”. We will not use common names in this tutorial, so leave this blank and just press **Enter** or select **OK**.

The **New File Options** dialog window opens. Since we elected (in the previous window) to *not* use the default template for this part, Pro/E is presenting a list of alternative templates defined for your system. As mentioned previously, we are going to avoid using defaults this time around. So, for now, as shown in Figure 3 at the right, select

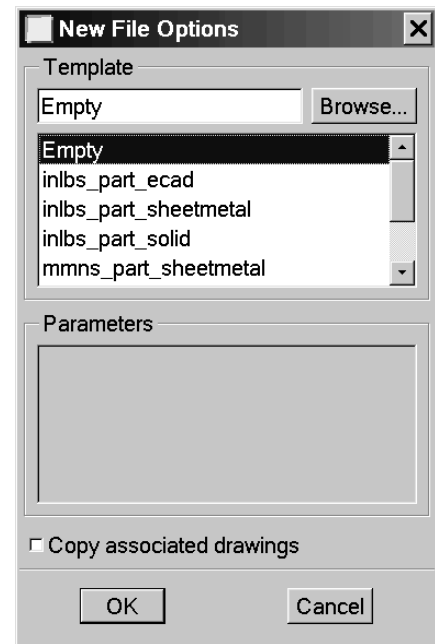


Figure 3 Options for new parts

Empty | **OK** (or middle click).

At this time, **BLOCK** should appear in the title area of the graphics window. Also, some of the toolbar icons at the right are now “live” (ie. not grayed out).

Create Datum Planes and Coordinate System

We will now create the first features of the part: three reference planes to locate it in space. It is not absolutely necessary to have datum planes, but it is a very good practice, particularly if you are going to make a complex part or assembly. Datum planes are created using the “Datum Plane” button on the right toolbar, as shown in Figure 4. Note that these icons look quite similar to the buttons on the top toolbar that control the display of datums. What’s the difference?

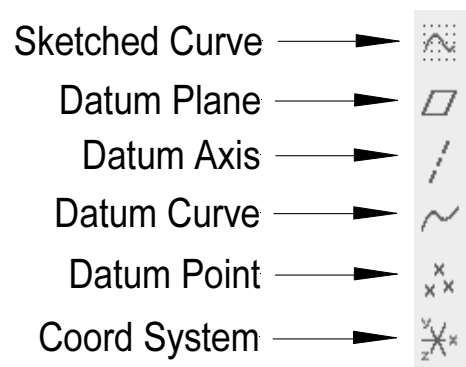


Figure 4 Toolbar buttons for creating DATUMS

¹ Pro/E can keep track of objects of different types with the same names. A part and a drawing can have the same name since they are different object types.

Select the **Datum Plane** button now. Since we currently have no features in the model, Wildfire rightly assumes that we want to create the three standard datum planes.

The datum planes represent three orthogonal planes to be used as references for features that will be created later. You can think of these planes as XY, YZ, XZ planes, although you generally aren't concerned with the X,Y,Z form or notation. Your screen should have the datum planes visible, as shown in Figure 5. (If not, check the datum display button in the top toolbar.) They will resemble something like a star due to the default 3D viewing direction. Note that each plane has an attached tag that gives its name: **DTM1**, **DTM2**, and **DTM3**. This view may be somewhat hard to visualize, so Figure 6 shows how the datum planes would look if they were solid plates in the same orientation. An important point to note is, while the plates in Figure 6 are finite in size, the datum planes actually extend off to infinity. Finally, before we move on to the next topic, notice that the last feature created (in this case DTM3), is highlighted in red. This is a normal occurrence and means that the last feature created is always preselected for you as the "object" part of the object/action command sequence.

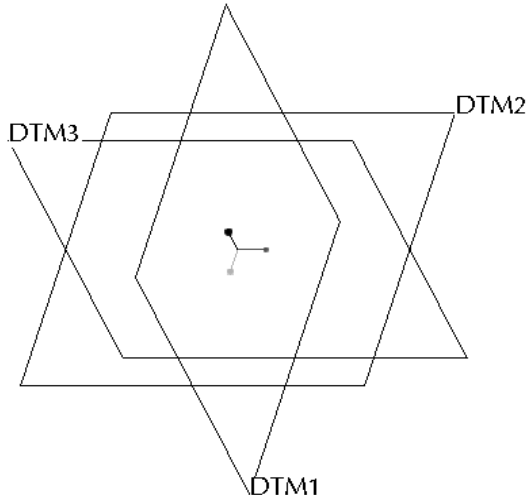


Figure 5 Default datum planes

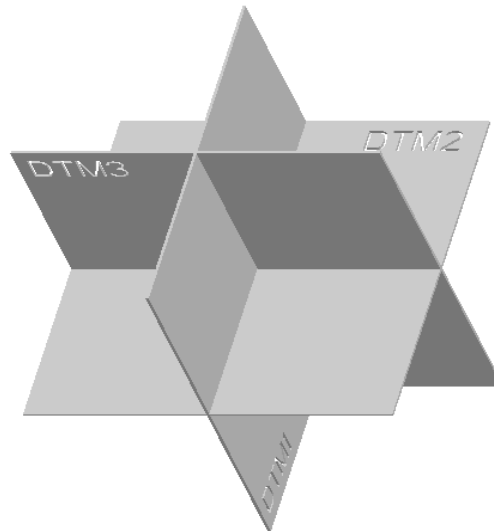


Figure 6 Datum planes as solid plates

Pro/ENGINEER Feature Overview

Below (and/or to the right of) the datum creation buttons in the right toolbar are three other groups of buttons. These are shown in Figures 7, 8, and 9. If you move the cursor over the buttons, the tool tip box will show the button name.

Two of these menus contain buttons for creating features, organized into the following categories:

Placed Features (Figure 7) - (holes, rounds, shells, ...) These are features that are created directly on existing solid geometry. Examples are placing a hole on an existing surface, or creating a round on an existing edge of a part.

Sketched Features (Figure 8) - (extrusions, revolves, sweeps, blends, ..) These features require the definition of a two-dimensional cross section which is then manipulated into the third dimension. Although they usually use existing geometry for references, they do not specifically require this. These features will involve the use of an important tool called Sketcher.

The final group of buttons (Figure 9) is used for editing and modifying existing features. We will deal with some of these commands (*Mirror* and *Pattern*) later in the Tutorial.

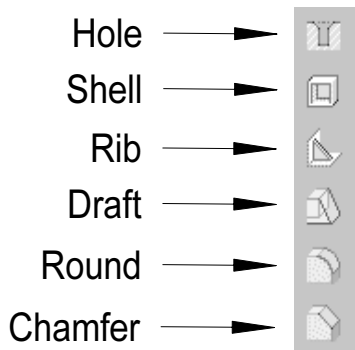


Figure 7 The Placed Features toolbar

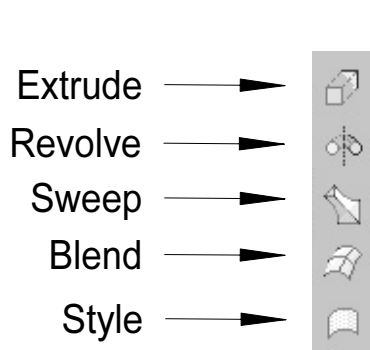


Figure 8 The Sketched Features toolbar

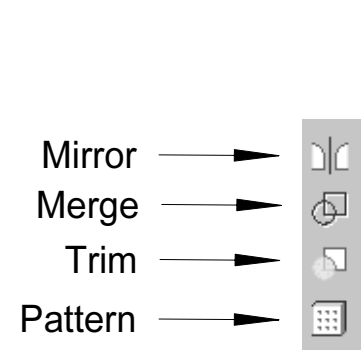


Figure 9 The Edit toolbar

In this lesson we will be using the *Extrude* command to create two types of sketched features (a protrusion and a cut). In the next lesson, we will use the *Hole*, *Round*, and *Chamfer* commands to create three placed features. Before we continue, though, we must find out about an important tool - Sketcher.

Introducing Sketcher

Sketcher is the most important tool for creating features in Pro/E. It is therefore critical that you have a good understanding of how it works. We will take a few minutes here to describe its basic operation and will explore the Sketcher tools continually through the next few lessons. It will take you a lot of practice and experience to fully appreciate all that it can do.

Basically, Sketcher is a tool for creating two-dimensional figures. These can be either stand-alone features (*Sketched Curves*) or as embedded elements that define the cross sectional shape of some solid features. The aspects of these figures that must be defined are location, shape, and size, roughly in that order. The sketching plane where we will create the 2D sketch is defined or selected first. Then, within Sketcher the location is further specified by selecting references to existing geometry. You will find the usual drawing tools for lines, arcs, circles, and so on, to create the shape. Finally, you can specify alignments or dimensions to control the size of the sketch.

Sketcher is really quite “smart”, that is, it will anticipate what you are going to do

(usually correctly!) and do many things automatically. Occasionally, it does make a mistake in guessing what you want. So, learning how to use Sketcher effectively involves understanding exactly what it is doing for you (and why) and discovering ways that you can easily over-ride this when necessary.

The “brain” of Sketcher is called the *Intent Manager*. We will be discussing the notion of *design intent* many times in this tutorial. In Sketcher, design intent is manifest not only in the shape of the sketch but also in how constraints and dimensions are applied to the sketch so that it is both complete and conveys the important design goals for the feature. Completeness of a sketch implies that it contains just enough geometric specification so that it is uniquely determined. Too little information would mean that the sketch is under-specified; too much means that it is over-specified. The function of Intent Manager is to make sure that the sketch always contains just the right amount of information. Moreover, it tries to do this in ways that, most of the time, make sense. Much of the frustration involved in using Sketcher arises from not understanding (or even sometimes realizing) the nature of the choices it is making for you or knowing how easy it is to override these actions. When you are using Sketcher, Intent Manager must be treated like a partner - the more you understand how it works, the better the two of you will be able to function².

The term *sketch* comes from the fact that you do not have to be particularly exact when you are “drawing” the shape, as shown in the two figures below. Sketcher (or rather Intent Manager) will interpret what you are drawing within a built-in set of rules. Thus, if you sketch a line that is approximately vertical, Sketcher assumes that you want it vertical. If you sketch two circles or arcs that have approximately the same radius, Sketcher assumes that’s what you want. In cases like this, you will see the sketched entity “snap” to a particular orientation or size as Intent Manager fires one of the internal rules.

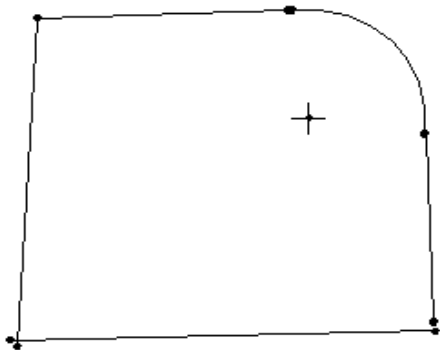


Figure 10 Geometry input by user. Note misaligned vertices, non-parallel edges, non-tangent curves.

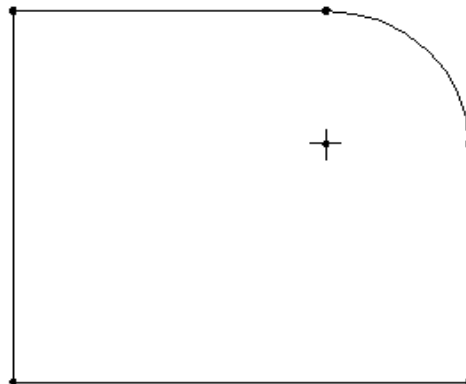


Figure 11 Geometry after processing by Sketcher. Note aligned vertices, parallel edges, tangent curves.

² Intent Manager was introduced several releases ago. Some veteran Pro/E users still have not made the switch from “the old days”. For those users, Pro/E has the ability to turn off the Intent Manager and let them do everything manually.

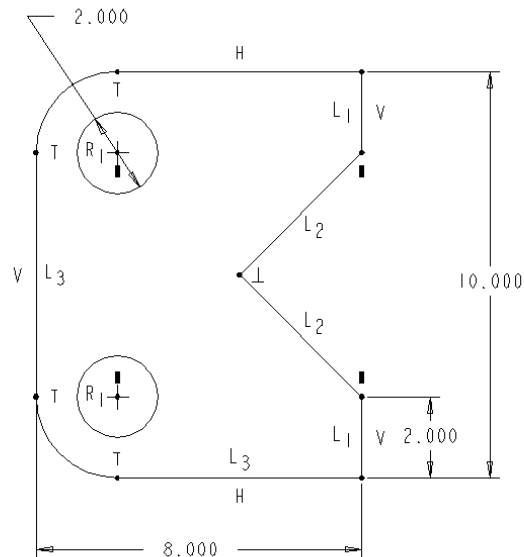
When Sketcher fires one of its internal rules (this occurs while you are sketching), you will be alerted by a symbol on the sketch that indicates the nature of the assumed condition. If you accept the condition, it becomes a *constraint* on the sketch. These constraint symbols are summarized in Table 2-1 on the next page. You should become familiar with these rules or constraints, and learn how to use them to your advantage. If you do not want a rule invoked, you must either

- (a) use explicit dimensions or alignments, or
- (b) exaggerate the geometry so that if fired, the rule will fail, or
- (c) tell Pro/E explicitly to ignore the rule (disable the constraint).

You will most often use option (a) by specifying your desired alignments and dimensions and letting Sketcher worry about whatever else it needs to solve the sketch. When geometry is driven by an explicit dimension, fewer internal rules will fire. Option (b) is slightly less common. An example is if a line in a sketch must be 2° away from vertical, you would draw it some much larger angle (like 15° or so) and put an explicit dimension on the angle. This prevents the “vertical” rule from firing. Once the sketch has been completed with the exaggerated angle, you can modify the dimension value to the desired 2°. For method (c), there is a command available that explicitly turns off the rule checking (for all rules or selected ones only) during sketching. This is very rarely used.

An example of a sketch with the geometric constraints is shown in Figure 12. Note how few dimensions are required to define this sketch. See if you can pick out the following constraints:




- ▶ vertical lines
- ▶ horizontal lines
- ▶ perpendicular lines
- ▶ tangency
- ▶ three sets of equal length lines
- ▶ equal radius
- ▶ vertical alignment (two cases)



How do you suppose Sketcher is able to determine the radius of the rounded corners (fillets) at the top and bottom on the left edge?

Figure 12 A sketch showing implicit constraints

Table 2-1 Implicit Constraints in Sketcher

Rule	Symbol	Description
Equal radius and diameter	R	If you sketch two or more arcs or circles with approximately the same radius, the system may assume that the radii are equal
Symmetry		Two vertices may be assumed to be symmetric about a centerline
Horizontal or vertical lines	H or V	Lines that are approximately horizontal or vertical may be considered to be exactly so.
Parallel or perpendicular lines	or ⊥	Lines that are sketched approximately parallel or perpendicular may be considered to be exactly so.
Tangency	T	Entities sketched approximately tangent to each other may be assumed to be tangent
Equal segment lengths	L	Lines of approximately the same length may be assumed to have the same length
Point entities lying on other entities or collinear with other entities		Point entities that lie near lines, arcs, or circles may be considered to be exactly on them. Points that are near the extension of a line may be assumed to lie on it.
Equal coordinates		Endpoints and centers of the arcs may be assumed to have the same X- or the same Y-coordinates
Midpoint of line	M	If the midpoint of a line is close to a sketch reference, it will be placed on the reference.

Two Ways to use Sketcher

In this lesson, we will use two methods to create a couple of sketched features. The two methods differ in how they use sketcher to define the cross section of the features. The method you use in your own modeling is a matter of personal preference. Both features we will make here are extrusions: one will be a *protrusion* (which adds material) and the other is a *cut* (which removes material). Either of the two methods shown here can be used to create either protrusions or cuts; for either method, whether you add or remove material is determined by a single mouse click!

In the first method, we invoke Sketcher first to create the cross sectional shape of the extrusion. This shape is defined in a *sketched curve* which becomes a stand-alone feature in the model. We then launch the extrude command, specifying the curve to define the cross section of the feature. In the second method, we do not create a separate curve but rather invoke Sketcher from inside the extrusion creation sequence.

In terms of design intent, the first method would be used if the sketched curve was going to be used for additional features, for example an extrude and a revolve. The second method (creating the sketch within the feature) is the traditional mode of operation, and would be the method of choice if the sketched shape was to be used only in a single feature.

Creating a Sketched Curve

When we left the model last, the datum plane DTM3 was highlighted in red. If that is not the case now, use preselection highlighting to select it now.

In the datum toolbar on the right of the screen, pick the *Sketched Curve* button. Be careful not to pick the datum curve button below it - that one will create a datum curve using sets of existing datum points, points read from a file, or using equations. If you accidentally pick the wrong button, you can back out with the *Quit* command.

Setting Sketch Orientation

The **Sketch** dialog window opens as shown in Figure 13. Since DTM3 was highlighted (in red) prior to the present command, it has been preselected as the Sketch Plane. It is now highlighted in the graphics window in orange. This is the plane on which we will draw the sketch. The view orientation has changed so that you are looking directly at DTM3. Two dashed lines represent sketch references that have been chosen automatically - these are the other datum planes seen on edge. A yellow circle is actually the “tail feathers” of a view direction arrow. Spin the orientation with the middle mouse button to see the arrow. The yellow arrow attached to the edge of DTM3 should be pointing back into the screen. This is the direction of view onto the sketch plane. The direction of view can be reversed with the *Flip* button in the dialog window (Figure 13). Leave it pointing towards the back. DTM1 is now highlighted in red in the graphics window. In the dialog window, DTM1 is identified as the Sketch Orientation *Reference*, with the *Orientation* set to *Right*. What is all this about?

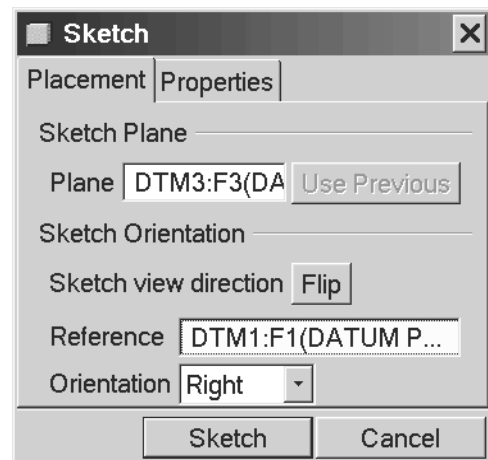


Figure 13 Defining the sketch plane and sketch orientation reference

The relation between the sketch plane and the sketch orientation reference generally causes a lot of confusion for new users, so pay attention!

The meaning of the sketch plane is pretty obvious - it is the plane on which we will draw the sketch - in this case DTM3. Our view is always perpendicular to the sketch plane³. That is not enough by itself to define our view of the sketch since we can be looking at that plane from an infinite number of directions (imagine the sketch plane rotating around an axis perpendicular to the screen). The *Orientation* option list in the dialog window (*Top, Bottom, Left, Right*) refers to directions relative to the computer screen, as in “TOP edge of the screen” or “BOTTOM edge of the screen” and so on. We must combine this direction with a chosen reference plane (*which must be perpendicular to the sketch plane*) so that we get the desired orientation of view onto the sketching plane.

In the present case, when we get into Sketcher we will be looking directly at the brown (positive) side of DTM3. So that the sketch is the right way up, we can choose either DTM2 to face the Top of the screen, or (as was chosen automatically for us) DTM1 can face the Right of the screen. Note that both DTM1 and DTM2 are both perpendicular to the sketch plane, as required. The direction a plane or surface “faces” is determined by its normal vector. The normal vector for a datum plane is perpendicular to the brown side. For a solid surface, the orientation is determined by the outward normal.

Read the last couple of paragraphs again, since new users are quite liable to end up drawing their sketches upside-down!

To illustrate the crucial importance of the reference plane, consider the images shown in Figure 14. These show two cases where the same sketching plane **DTM3** was used, the same sketched shape was drawn, the same reference orientation TOP was chosen, but where different datums were chosen as the sketching reference. On the left, the TOP reference was **DTM2**. On the right, the TOP reference was **DTM1**. The identical sketch, shown in the center, was used for both cases (rounded end of sketch towards the top of the screen). However, notice the difference in the orientation of the part obtained in the final shaded images. Both of these models are displayed in the default orientation (check the datum planes). Clearly, choosing the sketching reference is important, particularly for the base feature.

³Well, almost always. It is possible to sketch in 3D, in which case you can manipulate your view so that you are not looking perpendicularly at the sketch plane. We will not attempt that here.

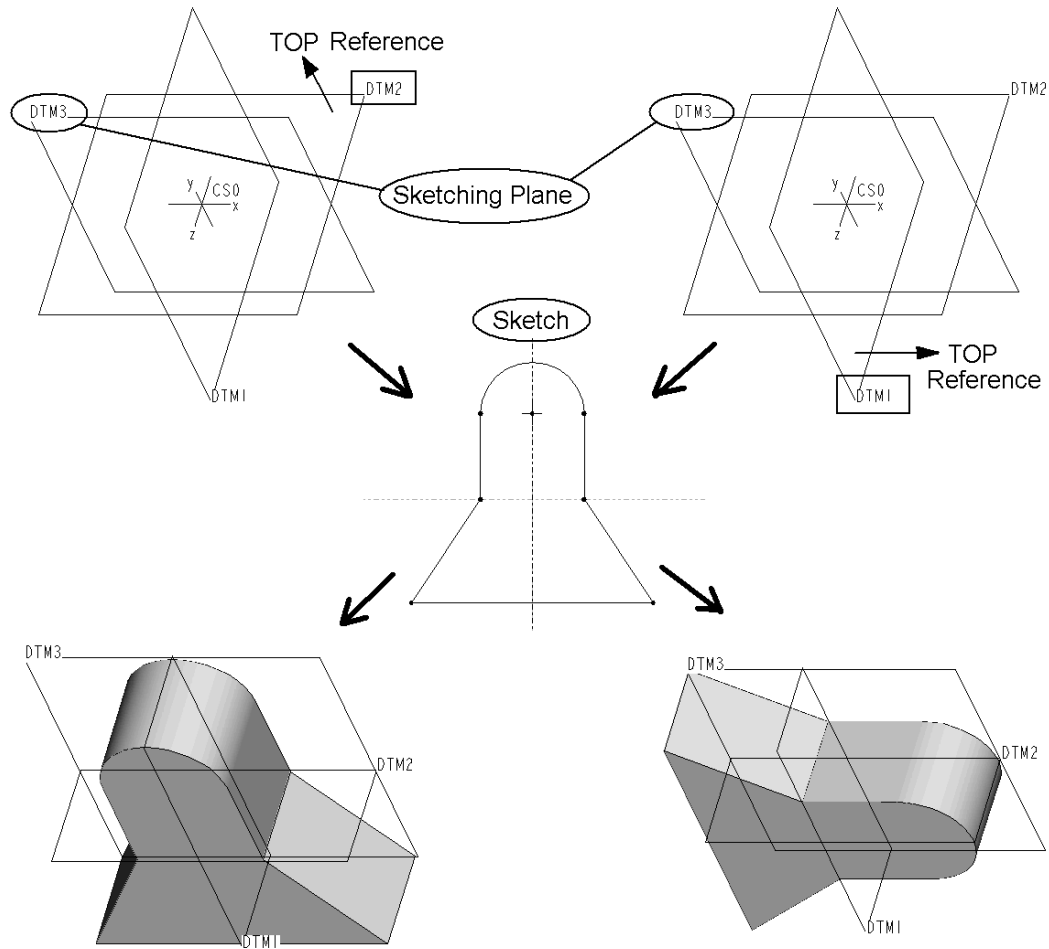


Figure 14 The importance of the sketching reference plane!

Let's continue on with creating the curve. Make sure the **Sketch** dialog window is completed as in Figure 13. Select the **Sketch** button in the dialog window.

To verify the meaning of the dashed orange lines, in the top pull-down menu, select

Sketch > References

This opens the **References** dialog window, Figure 15. In this window we can select any existing geometry to help Sketcher locate the new sketch relative to the part. In the present case, there isn't much to choose from, and two references have been chosen for us - DTM1 and DTM2. These references are responsible for the two dashed lines in the graphics window. The number of references you choose is not limited - there may be several listed here. You are also free to delete the ones chosen for you. However, notice the

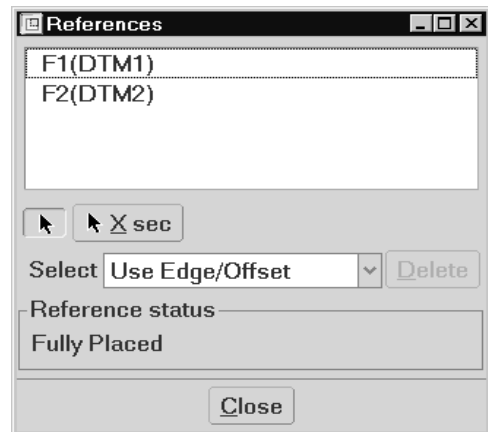


Figure 15 Choosing references in Sketcher

Reference Status at the bottom of this dialog. **Fully Placed** means enough references have been specified to allow Sketcher to locate your sketch in the model. If there are not enough references, the status will be **Partially Placed**. For now, do not proceed beyond this window unless you have a **Fully Placed** status indicated. Once you have that, select **Close** in the References window.

The drawing window is shown in Figure 16. Note that you are looking edge-on to the datums DTM1 and DTM2. The datum DTM1 (actually, its brown side) is facing the right edge of the screen, as specified in the dialog back in Figure 13. Note that we could have obtained the same orientation by selecting DTM2 to face the top of the screen.

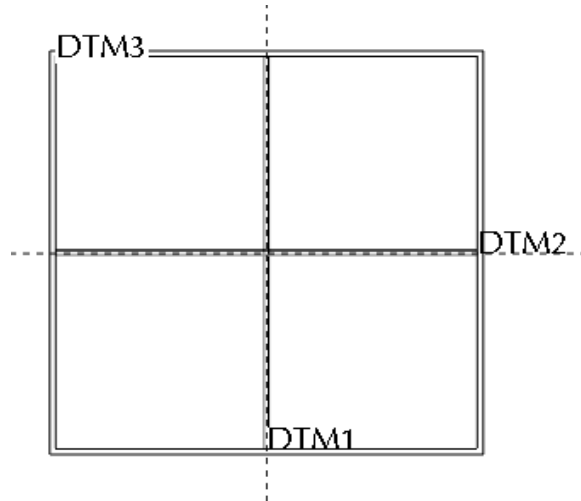


Figure 16 The drawing window

Another change is the addition of some new toolbar buttons at the top of the screen. These control the display of entities in the sketch. See Figure 17. The four buttons with the eyeballs control display of dimensions, constraints, the grid (default off), and vertices. Leave these buttons in their default position. It is seldom (if ever) that you will need to turn on the grid in Sketcher. The button at the left will return you to the default view of the sketch if you should accidentally (or intentionally) go into 3D view.

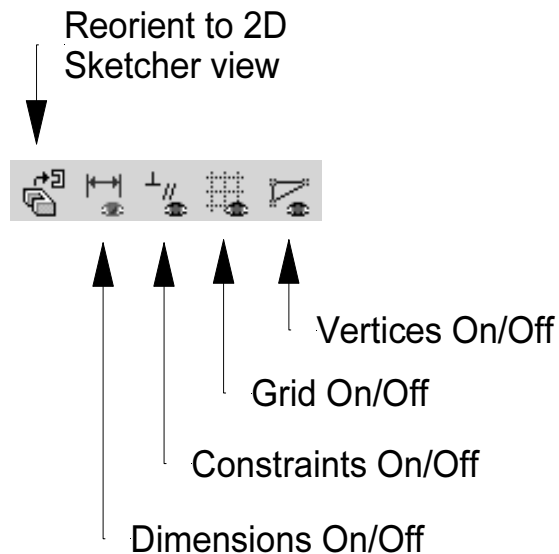


Figure 17 Sketcher top toolbar buttons

The Sketcher Toolbar

The major addition to the screen is the new toolbar on the right of the screen. This contains the Sketcher tools and is shown in Figure 18. Several buttons on this menu have fly-outs, indicated by the ► symbol on the right edge. These fly-outs lead to related buttons, and are listed in Table 2-2 on the next page. Compared to some 2D drawing programs, this doesn't seem like such a large number of drawing commands. Rest assured that there will not be much that you cannot draw with these.

When you are sketching, many of the commands in the right toolbar are instantly available (but context sensitive) by holding down the right mouse button in the graphics window. This will bring up a pop-up window of commands relevant in the current situation.

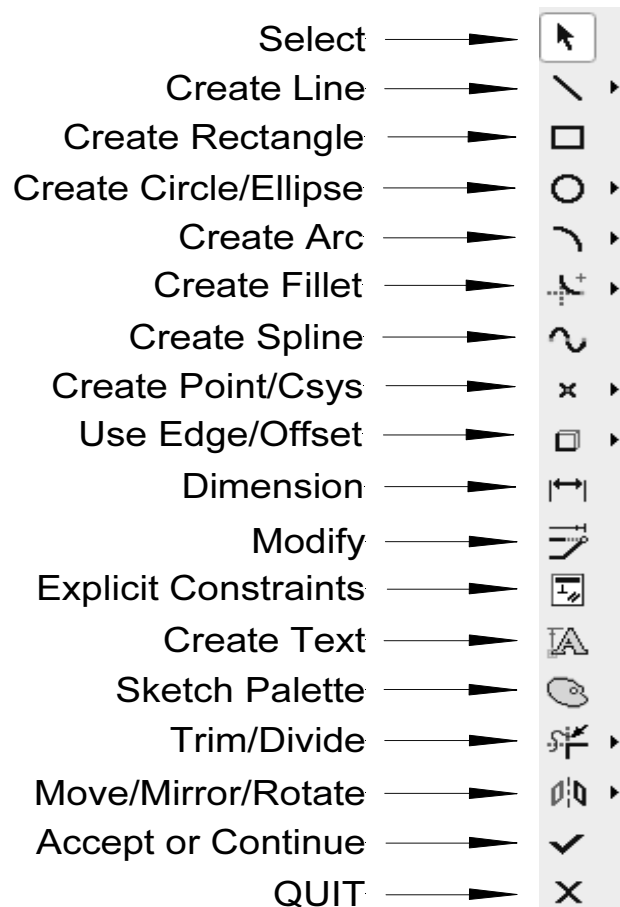






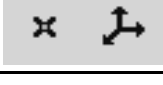
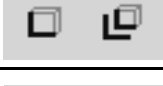


Figure 18 The Sketcher toolbar

Finally, a new Sketch pull-down menu is available at the top of the screen. Open this and have a look at the available commands there. Two commands not in the toolbar are **Data From File** and **Options**. The first of these lets you read in a previously created sketch. The second lets you change the default settings used by Sketcher. Have a look at these but don't change anything just yet.

Helpful Hint

From wherever you are in the Sketcher menu structure, a single **middle** mouse click will often abort the current command and return you to the toolbar with the **Select** command already chosen. Sometimes, you may have to click the middle button twice.

Table 2-2 Sketcher Toolbar Flyout Buttons

Button Flyout Group	Button Commands
	Line Tan-Tan Line Centerline
	Circle Concentric 3 Point 3 Tan Ellipse
	Tangent End Concentric Center 3 Tan Conic Arc
	Circular fillet Conic fillet
	Point Coordinate System
	Use Edge Offset edge
	Dynamic trim (delete) Trim(extend) Divide
	Mirror Rotate Move

Creating the Sketch

Select the **Line** tool using one of the following three methods:

- using the **Line** toolbar button on the right,
OR
- in the pull-down menus select **Sketch > Line > Line**,
OR
- hold down the right mouse button and select **Line** from the pop-up menu (Figure 19).

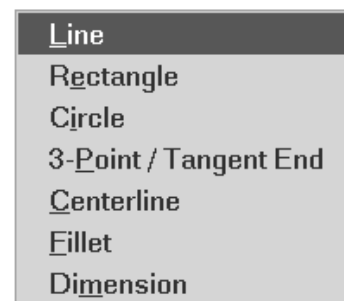


Figure 19 RMB pop-up menu in Sketcher

You will now see a small yellow X which will chase the cursor around the screen. Notice that the X will snap to the dashed references when the cursor is brought nearby. While you are creating the sketch, watch for red symbols (V, H, L) that indicate Intent Manager

is firing an internal rule to set up a constraint (Vertical, Horizontal, Equal Length). These symbols will come and go while you are sketching. The trick with Sketcher is to get Intent Manager to fire the rule you want, then click the left mouse button to accept the position of the vertex. Click the corners in the order shown in Figure 20. After each click, you will see a straight line rubber-band from the previous position to the cursor position:

1. left-click at the origin (intersection of **DTM1** and **DTM2**)
2. left-click above the origin on **DTM1** (watch for V)
3. left-click horizontally to the right (watch for H and L - we do not want L)
4. left-click straight down on **DTM2** (watch for V)
5. left-click back at the origin (watch for H)
6. middle-click anywhere on the screen to end line creation

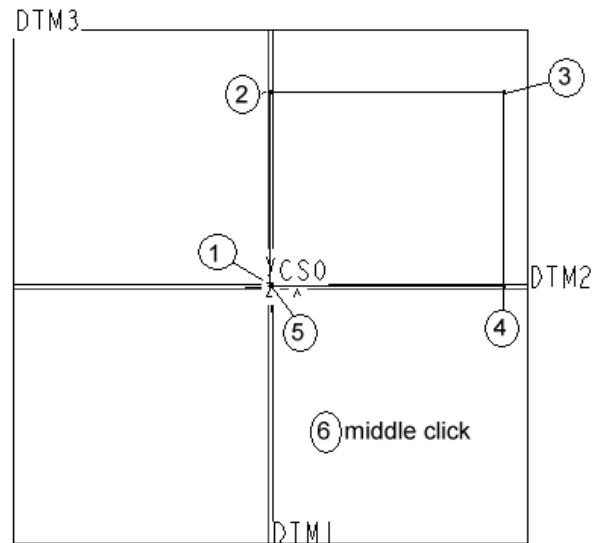


Figure 20 Drawing the Sketch


When you are finished this sequence, you are still in **Line** creation mode (notice the yellow X on screen and the **Line** toolbar button). If you middle click again, you will leave that and return to **Select** mode - the same as if you picked on the **Select** button in the right toolbar, but much faster.

The sketched entities are shown in yellow. Note that we didn't need to specify any drawing coordinates for the rectangle, nor, for that matter, are any coordinate values displayed anywhere on the screen. This is a significant departure from standard CAD programs. We also didn't need the grid or a grid snap function (although both of these are available if you want them).

You can also sketch beyond the displayed edges of the datum planes - these actually extend off to infinity. The displayed extent of datum planes will (eventually) adjust to the currently displayed object(s).

Helpful Hints

If you make a mistake in drawing your shape, here are some ways to delete entities:

1. Pick the **Select** tool in the right toolbar and left click on any entity you want to delete. Then either press the **Delete** key on the keyboard, or hold down the RMB and choose **Delete**.
2. If there are several entities to delete, hold the CTRL key down while you left click on each entity. Then pick **Delete** as before.
3. You can left-click and drag to form a rectangle around a set of entities. Anything completely inside the rectangle is selected. Use **Delete** as before.
4. Notice the **Undo** and **Redo** buttons on the top toolbar 

We will cover more advanced Sketcher commands for deleting and trimming lines a bit later.

After you have finished the sequence above, Sketcher will put two dimensions on the sketch - for the height and width of the rectangle. These will be in dark gray, so may be hard to see, but similar to those shown in Figure 21. For the first feature in a part, the numerical values of these dimensions are picked more-or-less at random (although they are in correct proportion to each other)⁴. For later features in the part, Sketcher will know the sketch size more accurately because it will have some existing geometry to set the scale.

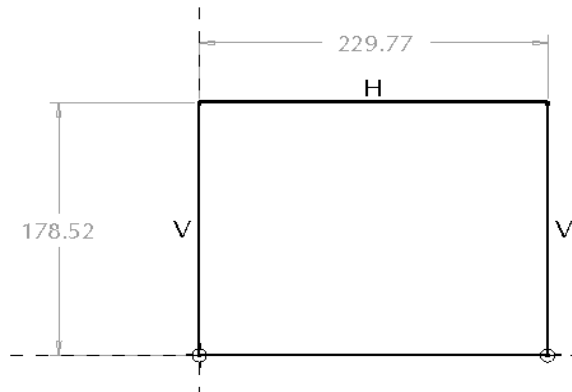





Figure 21 Completed sketch with weak dimensions

Weak vs Strong Dimensions

A dimension created by Sketcher is called “weak” and is shown in gray. Strong dimensions, on the other hand, are those that you create. You can make a strong dimension in any of three ways:

- or  modify the value of a weak dimension
- or  create a dimension from scratch by identifying entities in the sketch and placing a new dimension on the sketch
- or  select a weak dimension and promote it to strong using the RMB pop-up menu

⁴ The default datum display with no other features present is actually ± 250 units from where they cross.

Strong dimensions will be shown in white (actually a very pale yellow).

The special significance of weak and strong dimensions is as follows. When Intent Manager is “solving” a sketch, it considers the sketch references, any implicit rules that have fired (like H, V, and so on) and any existing dimensions. If there is not enough information to define the drawing (it is *underconstrained*), Sketcher will create the necessary and sufficient missing dimensions. These are the weak dimensions. If Sketcher finds the drawing is *overconstrained* (too many dimensions or constraints) it will first try to solve the sketch by deleting one or more of the weak dimensions (the ones it made itself earlier). It will do this without asking you. This is one way for you to override Intent Manager - if you don't like the dimensioning scheme chosen by Sketcher, just create your own (automatically strong) dimensions. Sketcher will remove whichever of the weak dimensions are no longer needed to define the sketch. Sketcher assumes that any strong dimensions you have created shouldn't be messed with! However, if Sketcher still finds the drawing overconstrained, it will tell you what the redundant information is (which may be dimensions or constraints), and you can choose what you want deleted. Thus, although weak dimensions can be deleted without asking you, Sketcher will never delete a strong dimension without your explicit confirmation.

We want to modify the two weak dimensions on the rectangle in a couple of ways. First, we can make a cosmetic improvement by selecting the dimension text (the number) and performing a drag-and-drop to move it to a better location. Note in passing that preselection highlighting also works with dimensions and constraints. Do that now, so that the dimensions are located as in Figure 22 (“off the sketch”).

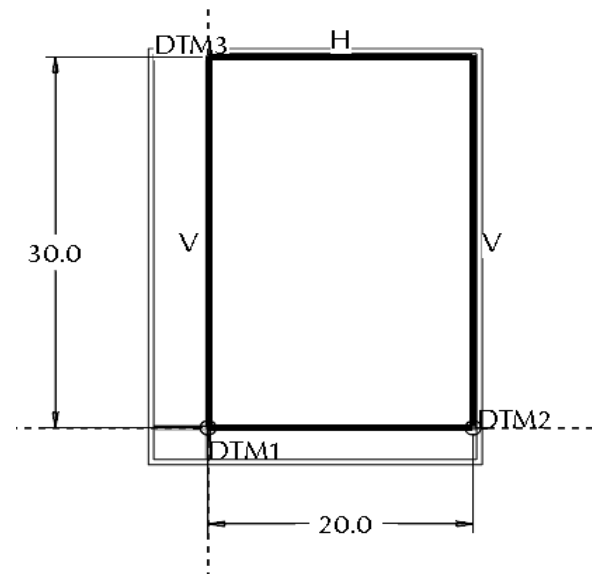


Figure 22 Modified sketch

Next we want to change the numeric value of the dimension. Double-click on the horizontal dimension. In the text entry box, enter the value **20**. When you hit Enter, the sketch geometry will be updated with this new dimension. The dimension is now strong. Change the vertical dimension to **30**. It will also now be strong. (Click anywhere on the graphics window to remove the red highlight.) See Figure 22. Notice that the indicated extent of the datum plane DTM3 adjusts to the sketch.

Now we'll add a couple of rounded corners, technically known as fillets, on the top corners of the sketch to help us "see" the orientation of the feature in 3D. Select the **Fillet** toolbar button on the right (or from the RMB pop-up menu) and pick on the top and right lines in the sketch close to but not at the corner. A circular fillet is created to the pick point closest to the corner. Two tangent constraints (T) are added, along with a weak dimension for the fillet radius. Do the same on the top and left lines. Middle click to return to Select mode.

Because our fillet command has removed two vertices on the top of the sketch, Intent Manager has removed our two strong dimensions (which used those vertices) and replaced them with weak ones (see the message window). You can make them strong by selecting them, clicking the RMB, and selecting **Strong** in the pop-up menu. Modify the cosmetics and values of the fillet radius dimensions as shown in Figure 23.

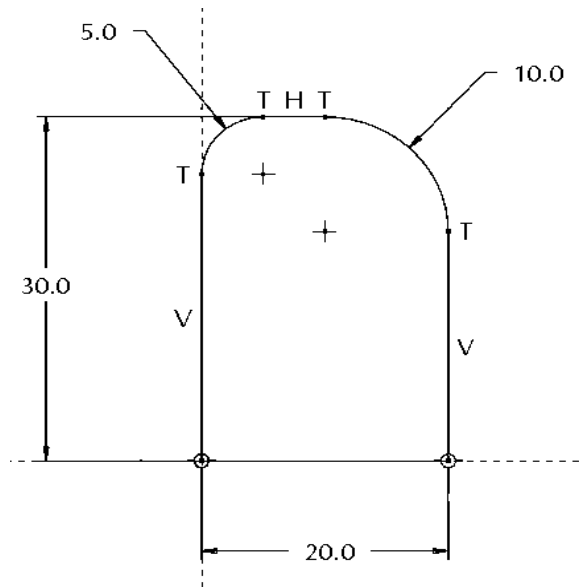


Figure 23 Completed sketch

This completes the creation of our first sketch. Select the **Accept** (or **Continue**) toolbar icon (the check mark). This returns us to the regular graphics window with our new sketched curve shown in red (last feature created). You can spin the model around with the middle mouse button to see this curve from different view points. When you are finished with this, return the model to approximately the default orientation - Figure 24.

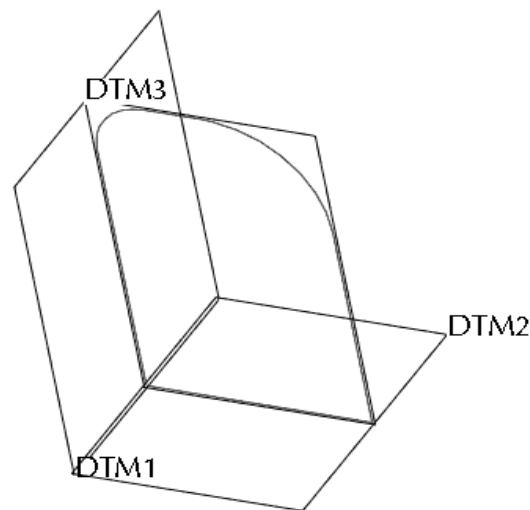


Figure 24 Sketched curve

Creating a Solid Protrusion

Most of the work to create this protrusion has been done already - creating the sketched curve that defines its shape. This curve should be highlighted in red. If you have been playing around with the model and the sketch is blue, just left click on it to select it again.

There are a number of ways to launch the protrusion creation command. With the sketched curve highlighted, the easiest way is to pick the **Extrude** button in the right toolbar.

What you will see now is a yellow shaded image of the protrusion, Figure 25. On this shape, you will see a yellow arrow that indicates the extrusion direction, which by default comes off the positive side of the sketch. There is also a dashed line ending in a white square. This is a drag handle. Click on this with the mouse and you can drag it to change the length of the extrusion. This length is also shown in a dimension symbol. You can even drag this extrusion out the back of the sketch to extrude in the opposite direction. This direct manipulation of the feature on the screen is called, in Pro/E vernacular, *Direct Modeling*. Bring the protrusion out the front and double click on the numeric dimension, and enter the value **30**.

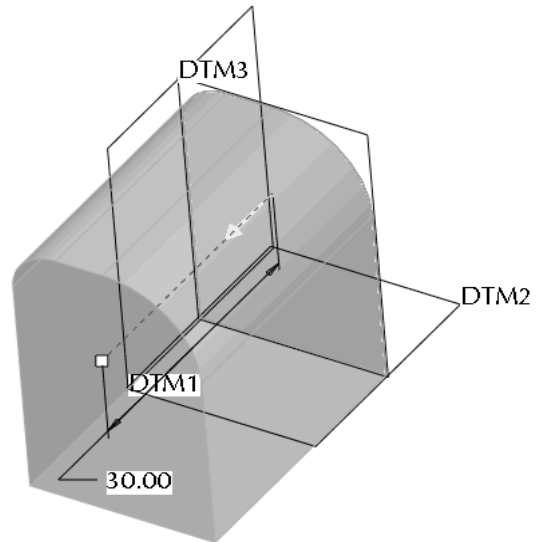


Figure 25 The protrusion preview

At the bottom of the graphics window is a new collection of tools. These comprise the *Dashboard*. Many features are constructed with tools arranged using this interface element. It is worth spending some time exploring this one in detail, since you will probably be using it the most.

Helpful Hint

You may accidentally leave the dashboard with an inadvertent click of the middle mouse button. Remember that this is a short cut for “Accept.” If that happens, with the protrusion highlighted in red, hold down the right mouse button and select Edit Definition. This will bring you back to the dashboard. The Undo command, if executed immediately, will delete the feature.

The Extrude Dashboard

The dashboard collects all of the commands and options for feature creation in an easily navigated interface. Moreover, most optional settings have been set to default values which will work in the majority of cases. You can change options at any time and in any order. This is a welcome and significant departure from releases of Pro/E prior to Wildfire.

The dashboard contains two areas. On the left (Figure 26) are commands, settings, and so on for the particular feature under construction. On the top row, the feature is identified with the toolbar icon - extrude in this case - and several slide-up panels which do the following:

Placement - allows you to select, create, or modify the 2D section to be used for the feature. Since we preselected the sketched curve, it is now listed on this panel. If we had not preselected the curve, we could have chosen it now, or launched Sketcher from this panel to create a new sketch. This would involve selecting the sketching plane, sketcher reference, and so on. We will go this route in the next

feature. If you wanted to change the sketch for the extrude, this is how you access it. The **Unlink** button is currently displayed on the **Placement** panel. This button appears if, like now, you have preselected a curve to serve as the sketch for the extrude. Thus, the extrude is linked to the previous curve feature; changes to the geometry or dimensions of the curve would drive changes in the shape of the extrude. The curve itself is a separate entry on the model tree. The purpose of unlinking is to break this (parent/child) connection to the original curve. If you were to select this command (**don't do this now**), a copy of the original curve will be brought into the extrude feature. In that case, a change to the original curve would not affect the extrude. The original curve could be modified, moved, or even deleted, and the extrude would still be able to regenerate. The use of external curves to drive feature geometry is an important aspect of an advanced modeling technique that makes use of *skeleton models*.

Options - information about the depth specification for the feature. We will find out what is meant by "Side 2" in a later lesson. For a simple extrude, the depth specification is easiest to set using one of the icons in the lower dashboard area (see below).

Properties - specify the name of the feature

The icons on the second row operate as follows:

Solid and **Surface** buttons - these are an either/or toggle set. The default button is to create a solid. If you pick the next button, **Surface**, the sketch will be extruded as an infinitely thin surface (Figure 27). Return this to the **Solid** selection.

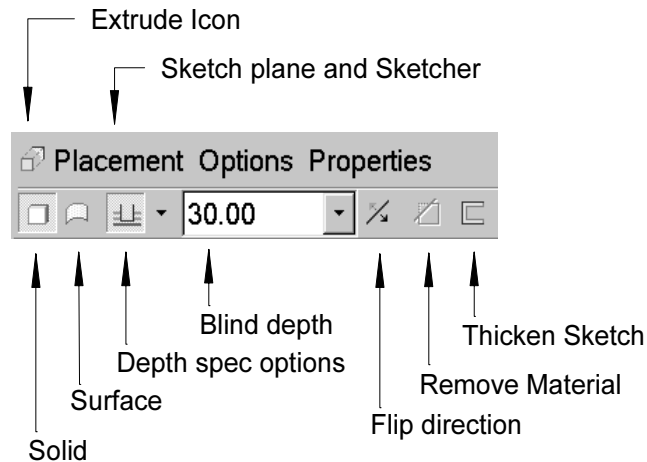


Figure 26 The **Extrude** Dashboard

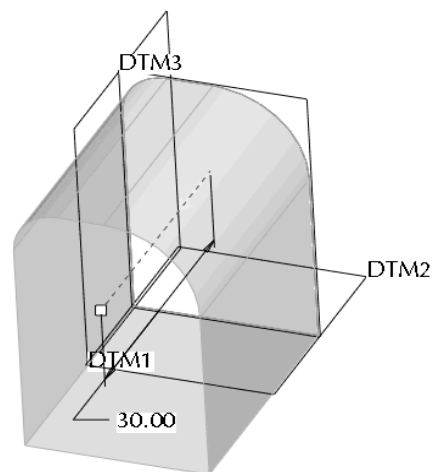


Figure 27 Extruded surface

Depth Spec Options - the next button is a pull-up list of all the possibilities for setting the depth of the extrusion. These are indicated in Figure 28. The default is a **Blind** extrusion, which means the extrusion is for a fixed distance. Other options may appear here as more part geometry appears (as in the cut which we will do next).

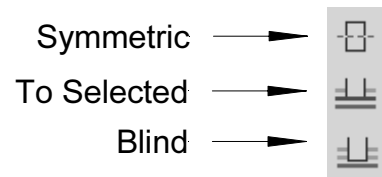


Figure 28 Depth Spec options

Blind Depth - this contains the numeric value of the length of the protrusion. If the depth specification on the button to the left is not Blind, this text input area is grayed out.

Flip - selecting this will reverse the direction of the protrusion (the yellow arrow).

Remove Material - this allows you to change the meaning of the solid feature from a protrusion (which adds solid material) to a cut (which removes solid material). Since there is nothing to remove at this time, this command is grayed out - all we can do is add.

Thicken Sketch - press this to see the solid block replaced by a thin-walled extrusion (formerly called a *Thin Solid*). A new dimension appears in the graphics window and on the dashboard. This is the thickness of the solid wall. Try changing this thickness to something like 1.0. On which side of the sketched curve has this been added? Another **Flip** button has also appeared. Press this a couple of times - it controls which side of the sketch the material is added to. Actually, it is a three way switch since you can also add material equally on both sides of the sketch. Press the **Thicken** button again to return to a full solid protrusion.

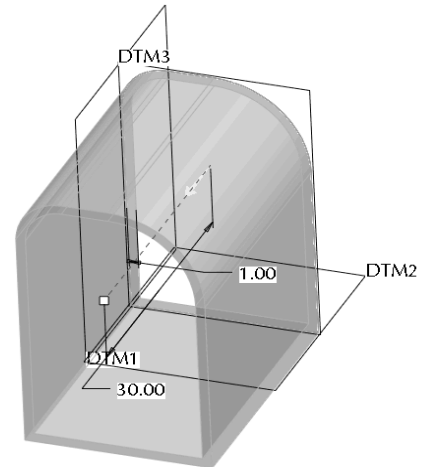


Figure 29 A **Thick** extruded solid

As you explore the creation of new features in Pro/E you should investigate what is in each of these menus. They are context sensitive, so there is a lot of variety in what you will find.

On the right end of the dashboard are several common tools that appear for all features. See Figure 30. These function as follows:

Pause - allow you to temporarily suspend work on this feature so that you can, for example, create a missing reference like a datum plane, measure something in the model, etc. When you are finished with the side trip, press the symbol ▶ that appears here to continue where you left off.

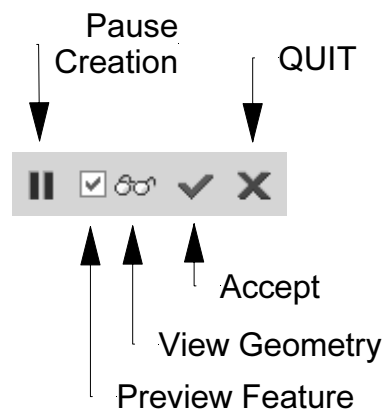


Figure 30 Common dashboard controls

Preview - (default on = checked) this is responsible for the shaded yellow display of the feature under construction. Uncheck this - all you will see is the feature creation direction, drag handle, and depth dimension. Turn this back on.

View Geometry (or Verify) - this shows what the geometry will look like when the feature is fully integrated into the part. Not much happens with this first protrusion. Press again to return to preview.

Accept and **Quit** - do just what you expect!

Select **Accept** (or middle click). The message window informs you that the feature has been created successfully. The block now appears, Figure 31, with its edges highlighted in red (last feature created).

We spent a lot of time discussing the creation of this feature. Let's create it again without the discussion. Select the **Undo** button at the top - this will delete the protrusion. Now create it again:

- ✓ highlight the sketched curve
- ✓ select the **Extrude** tool
- ✓ change the depth to **30**
- ✓ middle click.

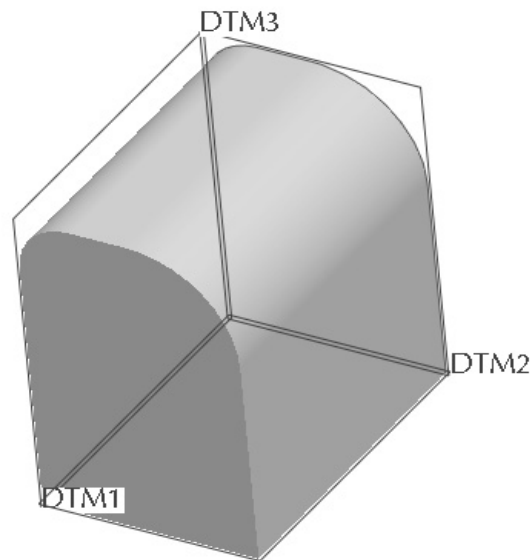


Figure 31 Completed protrusion

You might open the Navigator pane to see the model tree. The three default datum planes are listed. Then comes the sketched curve SKETCH 1 (hidden) and the extrude EXTRUDE 1. Open the listing for the extrude and you will see the internal sketch (also SKETCH 1) that uses the hidden curve and is currently linked to it. If you have used the **Unlink** button in the **Placement** panel, the model tree will appear somewhat differently - the sketch stored inside the extrude is called S2D0001 and is independent of SKETCH 1. Close the Navigator.

We will now add another extruded feature - this time we will create a cut that removes material. Furthermore, instead of creating the sketch first, as we did for the solid protrusion, we will create the sketch within the feature itself. This is actually the more common way to use Sketcher.

Creating an Extruded Cut

Start by launching the *Extrude* command from the right toolbar. The extrude dashboard at the bottom of the screen opens. Open the *Placement* slide-up panel in the dashboard and select *Define*. The *Sketch* dialog window appears. This time, however, nothing has been preselected for us as it was for the previous sketch. We'll have to enter the data ourselves.

First, the dialog is waiting for you to select the sketching plane (notice the pale yellow data entry field). Pay attention to preselection here. Notice the preselection filter setting (should be *Surface*); you will not be able to pick an edge or a curved surface (both of these would be illegal). Pick on the right side surface of the block (see Figure 32).

As soon as you pick the sketching plane (it highlights in orange), a yellow arrow will appear showing the default direction of view relative to the surface. The *Flip* button can be used to reverse this direction, but leave it as it is. Pro/E makes a guess at a potential reference plane for you to use. This may depend on the current orientation of your view, and might result in a strange view orientation in sketcher (like sideways or even upside down). We want to be a bit more careful and specific here. Pick on the top planar surface (Figure 32), between the two tangent lines of the rounded corners; the surface will highlight in red. In the *Orientation* pull-down list, select *Top* so that the reference will face the top of the screen. We now have our sketch plane and reference set up, so select *Sketch* at the bottom of the dialog window.

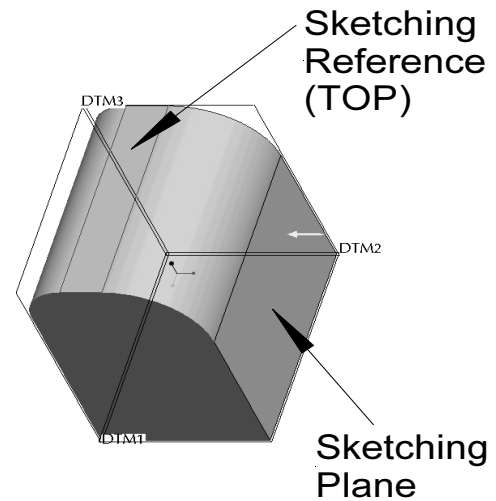


Figure 32 Setting up to sketch the cut

We are now in Sketcher (Figure 33). Two references have been chosen for us (the back and top surfaces of the object). These will be OK for now, so select *Close* in the *References* window. We are going to create the U-shaped figure shown in Figure 34. Note that there is no sketched line across the top of the U - there is no inside or outside. Thus, it is technically called an *open* sketch (as opposed to a *closed* sketch for our previous feature). There are some restrictions on the use of open sketches which we will run across in a minute or two.

You might prefer to set your display mode to *Hidden Line* at this point.

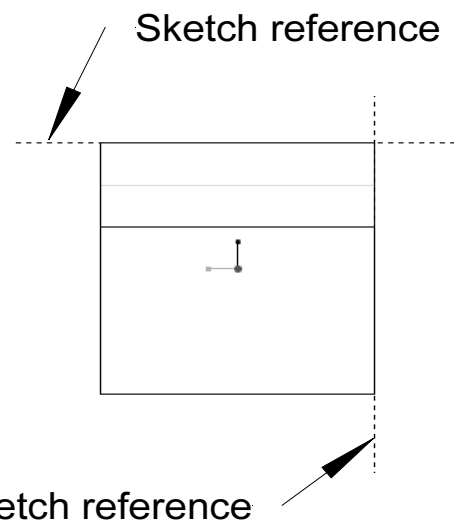


Figure 33 References for cut sketch

Helpful Hint

In general, try to keep your sketches closed - you will have fewer problems that way.

Use the RMB pop-up menu to select the **Line** command. Start your sketch at vertex 1 in Figure 34 - the cursor will snap to the reference. Then drag the mouse down and pick vertex 2 (note the V constraint), and middle click twice to end the **Line** command. Some weak dimensions will appear. Do nothing about them yet because, since they are weak, they are liable to disappear anyway. If we make them strong, this will cause us extra work dealing with Intent Manager.

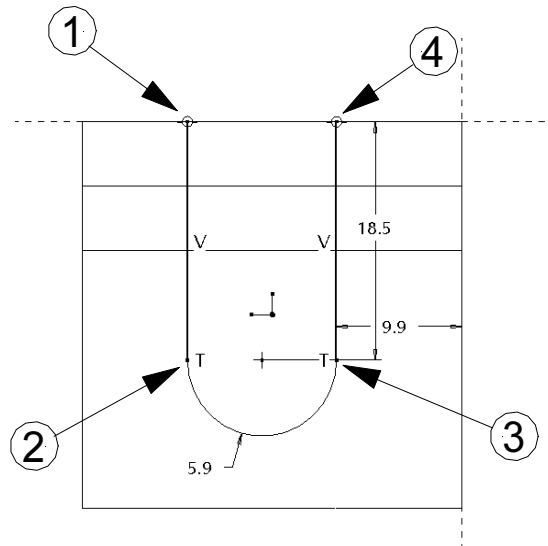


Figure 34 Sketch geometry

Helpful Hint

Wait until the shape of the sketch is finished before you start worrying about the dimensioning scheme or dimension values.

Use the RMB pop-up menu again and select the **3-Point/Tangent End** command. Pick on the end of the sketched line (vertex 2) and drag the mouse downwards in the direction of tangency. Once the arc has been established, drag the cursor over to the right (the arc will rubber-band while maintaining the tangency constraint) and click at vertex 3. (If you drag straight across to vertex 3 you will get a 3-point arc which is not automatically tangent at vertex 2.) You should see two small blips that indicate when vertex 3 is at the same height as the center of the arc. Use the RMB menu to pick **Line** again.

Now left click at vertex 3 and draw a vertical line up to snap to the reference at vertex 4. Our sketch is complete. Use the middle mouse button to return to Select mode. Your dimension values may be different from those shown in Figure 34. Your dimensioning scheme may even be slightly different. It will be easier to see this if you go to hidden line display instead of shading.

All the dimensions should be weak. Drag them to a better location if necessary (off the

part). Don't be afraid to resize your display so that you can see everything clearly. Compare the dimensioning scheme with the one in Figure 36. We want to have a horizontal dimension of 15 from the reference at the back of the part to the center of the arc of the U. If you do not have that dimension, we'll have to add one manually. This will illustrate a case where we will override the Intent Manager.

To create your own dimension, select the **Dimension** command from the right toolbar (or the RMB pop-up). Click on the vertex at the center of the arc (it will highlight) then click again on the dashed reference line at the right. Now middle click in the space above the part where you want the dimension text to appear. It's that easy! Note that this dimension shows immediately in pale yellow (almost white) since it is strong. One of the weak linear dimensions should be gone. Middle click to get back to **Select**.

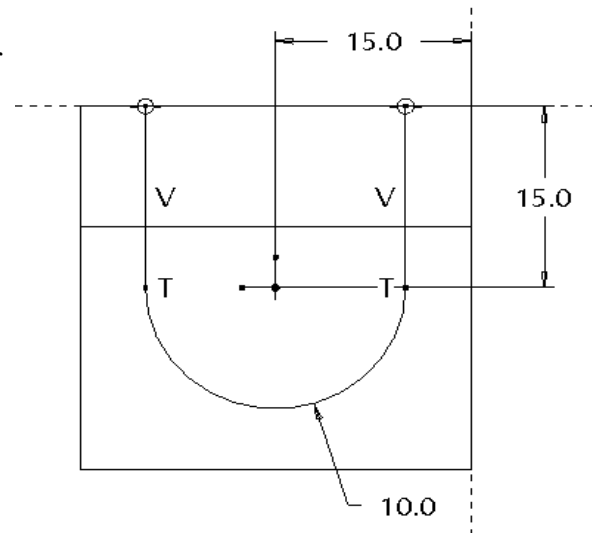


Figure 35 Final sketch for cut

We now have the implicit constraints and dimensioning scheme we want. So now we can worry about the dimension values. Get in the habit of dealing with your sketches in that order (shape, constraints, dimension scheme, dimension values). Modify the values of the dimensions to match those in Figure 35.

The sketch is now complete, so click on the **Accept** button in the toolbar⁵. If you are in hidden line display, return to shading display. In the **Sketch** dialog window, select **OK**.

The feature will now be previewed. A couple of new buttons have appeared on the dashboard. First, in the **Depth Spec** pull-up list, there are a few more options available (Figure 36). For this cut, we would like the sketch to be extruded through the entire part, so pick the **Through All** option. Note that the dimension for a blind extrusion disappears from the screen. To the right of this area, click the **Flip** button to make the extrusion go through the part. The **Remove Material** button needs to be selected. Now, there are two yellow arrows attached to the sketch. The one perpendicular to the plane of the sketch shows the direction of the extrusion. The other shows which side of the sketched line we want to remove material from. These should be set as shown in Figure 37.

⁵ This is one of the few times when a middle click does not mean "Accept", which is a good thing since inadvertent middle clicks happen often when you are in Sketcher.

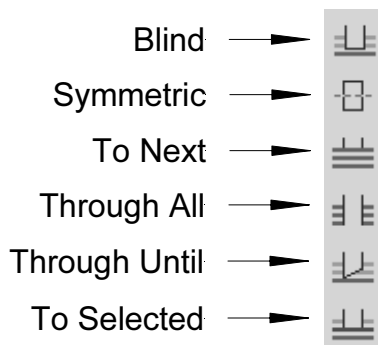


Figure 36 More Depth Spec options in the dashboard

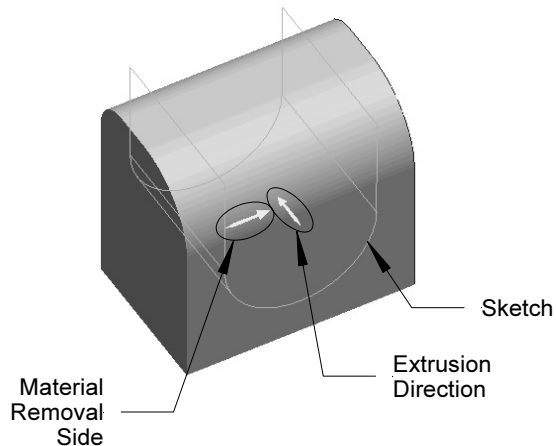


Figure 37 Defining cut attributes (direction and material removal side)

Now select the **Verify** button in the right area of the dashboard. If you have the **Remove Material** button set wrong, that is for a protrusion instead of a cut, Pro/E will not be able to create the feature. You will be launched into a **Troubleshooter** dialog window. You might come back and explore this later. For now, if you are in this window, select **Cancel**, turn **Verify** off, and select **Remove Material** again. The source of this problem is the open sketch for the U. This sketch is ambiguous since when the sketch starts out from the sketch plane, the vertices at the ends are out in the open air; Pro/E does not know how to create the solid to attach it to the existing part. This problem does not occur with a cut as long as the open ends of the sketch stay outside or on the surface of the part.

Another common error with cuts is having the material removal side set wrong (the second yellow arrow in Figure 37). If you do that for this part, you will end up with Figure 38. Make sure the material removal arrow points to the inside of the U. Plus, you should explore the **Placement**, **Options**, and **Properties** menus on the dashboard before you leave.

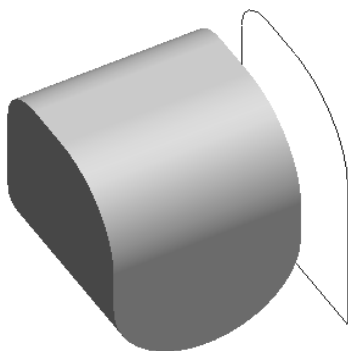


Figure 38 Removing from the wrong side of the sketch

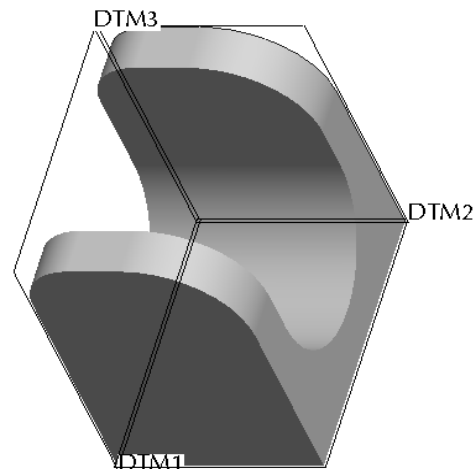


Figure 39 Cut feature completed

We are finished creating this feature, so select the *Accept* button at the right end of the dashboard. The part should now look like Figure 39 when in default orientation. The cut will be highlighted in red as usual, as the last feature created.

Saving the Part

It is a good idea when you are just getting started to save your model quite frequently, just in case something serious goes wrong. If you have to bail out of the program, you can always reload the most recently saved copy of the part and continue from there.

There are (as usual!) several ways to save the part:

- in the top toolbar, select the *Save* button, or
- in the pull-down menus select *File > Save*, or
- use the keyboard shortcut CTRL-S.

Make sure that the **Save Object** dialog is showing the desired directory at the top. One of the icons to the right of this will select the current working directory. At the bottom, you enter the name of the object to be saved (remember that you can have more than one object loaded into memory at a time). Accept the default model name [**block.prt**] (this is the *active* part) by pressing the enter key or the middle mouse button. Pro/E will automatically put the part extension (*prt*) on the file. If you save the part a number of times, Pro/E will automatically number each saved version (like block.prt.1, block.prt.2, block.prt.3, and so on). Be aware of how much space you have available. It may be necessary to delete some of the previously saved versions; or you can copy them to a diskette. You can do both of these tasks from within Pro/E - we'll talk about that later.

IMPORTANT NOTE:

The *Save* command is also available when you are in Sketcher. Executing this command at that time will *not* save the part, but it will save the current sketched section with the file extension *sec*. This may be useful if the sketch is complicated and may be used again on a different part. Rather than recreate the sketch, it can be read in from the saved file (using *Data from File*). In these lessons, none of the sketches are complicated enough to warrant saving them to disk.

Using Part Templates

You will recall that in the block part created earlier, the first thing we did was to create default datum planes. These (plus the named views based on them, which we didn't create this lesson) are very standard features and aspects of part files, and it would be handy if this was done automatically. This is exactly the purpose of part templates.

A template is a previously created part file that contains the common features and aspects of almost all part files you will ever make. These include, among other things, default datum planes and named views. Pro/E actually has several templates available for parts, drawings, and assemblies. There are variations of the templates for each type of object. One important variation consists of the unit system used for the part (inches or millimeters). Templates also contain some common model parameters and layer definitions⁶.

A template can be selected only when a new model is first created. Let's see how that works. Create a new part (note that you don't have to remove the block - Pro/E can have several parts "in session" at the same time) by selecting

File > New

or using the "Create New Object" button. The **New** dialog window opens. The

Part | Solid

options are selected by default. Enter a new name, like *exercise_1*. Remove the check mark beside *Use default template* and then select **OK**.

In the **New File Options** dialog window, the default template is shown at the top. It is likely "inlbs_part_solid" (unless your system has been set up differently). This template is for solid parts with the units set to inch-pound-second. It seems strange to have force and time units in a CAD geometry program. Actually, this is included so that the part units are known by downstream applications like Pro/MECHANICA which perform finite element analysis (FEA) or mechanism dynamics calculations. These programs are very picky about units!

Note that there are templates available for sheet-metal parts and for metric units (millimeter-Newton-second). While we are mentioning units, be aware that if you make a wrong choice of units here, it is still possible to change the units of a part after it has been created (see *Edit > Setup > Units*).

There are only two model parameters in the default template. *DESCRIPTION* is for an extended title for the part, like "UPPER PUMP HOUSING". This title can (eventually) be called up and placed automatically on a drawing of the part using, you guessed it, a drawing template. Similarly, the *MODELED_BY* parameter is available for you to record your name or initials as the originator of the part. Fill in these parameter fields and select **OK**.

The new part is created which automatically displays the default datums. They are even named for you (we will see how to name features in lesson 3): instead of DTM1, we have **RIGHT**. **TOP** replaces DTM2, and **FRONT** replaces DTM3. The part also contains a coordinate system, named views (look in the Saved Views List), and other data that we'll

⁶ Model parameters and layers are discussed in the *Advanced Tutorial*.

discover as we go through the lessons. The named views correspond to the standard engineering views. Thus, it is important to note that if you are planning on using a drawing template, your model orientation relative to the default datums is critical. The top-front-right views of the part are the ones that will be automatically placed on the drawing later. If your model is upside down or backwards in these named views, then your drawing will be too. This is embarrassing and not likely to win favor with your boss or instructor!

Now, having created this new part, you are all set up to do some of the exercises at the end of the lesson. Do as many of these as you can. Perhaps do some of them in different ways by experimenting with your sketch orientation, Sketcher commands, and so on.

This completes Lesson #2. You are strongly encouraged to experiment with any of the commands that have been presented in this lesson. Create new parts for your experiments since we will need the block part in its present form for the next lesson.

In the next lesson we will add some more features to the block, discover the magic of relations, and spend some time learning about the utility functions available to give you information about the model.

Questions for Review

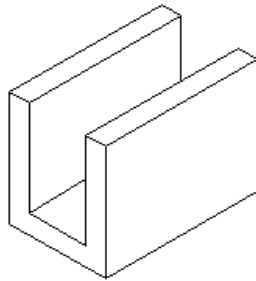
Here are some questions you should be able to answer at this time:

1. What is meant by a blind protrusion?
2. What is the purpose of the sketching reference?
3. How do you specify the name of a part?
4. Give as many of the Sketcher implicit rules as you can.
5. How do you save a part?
6. What is a template?
7. What is your system's default template?
8. Where does your system store your part files when they are saved?
9. What is meant by the *active* part?
10. How does Sketcher determine the radius of a fillet created on two lines?
11. What happens if you delete any of the constraints (H, V, etc.) on a sketch?
12. In an extrude, what happens if you set the thickness of a thickened sketch greater than the radius of a filleted corner of the sketch?
13. What is meant by *Linking* to a sketch?
14. In Sketcher, what is the difference between a gray and a white (pale yellow) dimension?
15. In Sketcher, how do you create an explicit dimension?
16. In Sketcher, how do you indicate where you want the dimension text placed?

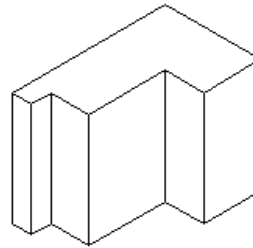
Exercises

Here are some simple shapes that you can make with simple extrusions. They should give you some practice using the Sketcher drawing tools and internal rules. Choose your own dimensions and pay attention to alignments and internal constraints. The objects should appear in roughly the same orientation in default view. Have a contest with a buddy to see who can create each object with the fewest number of dimensions. This is not necessarily a goal of good modeling, but is a good exercise! Feel free to add additional features to these objects.

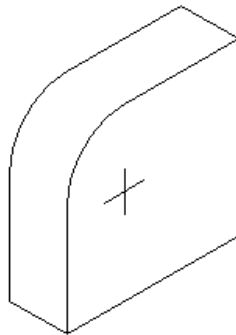
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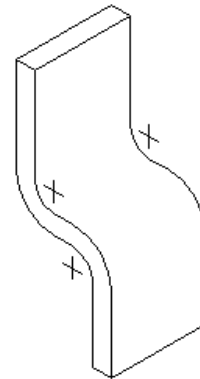
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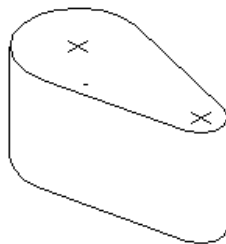
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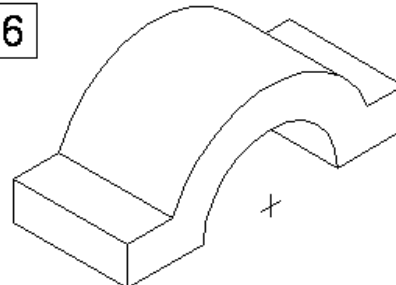
4



5

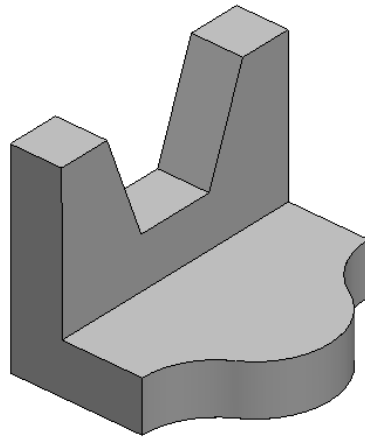


6

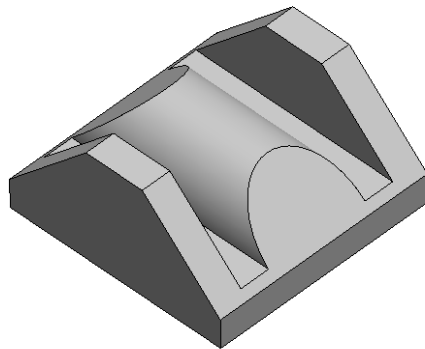


These parts are a bit more complicated, requiring two or more simple extruded features (protrusions or cuts). Think about these carefully before you try to make them.

1.



2.



3.

