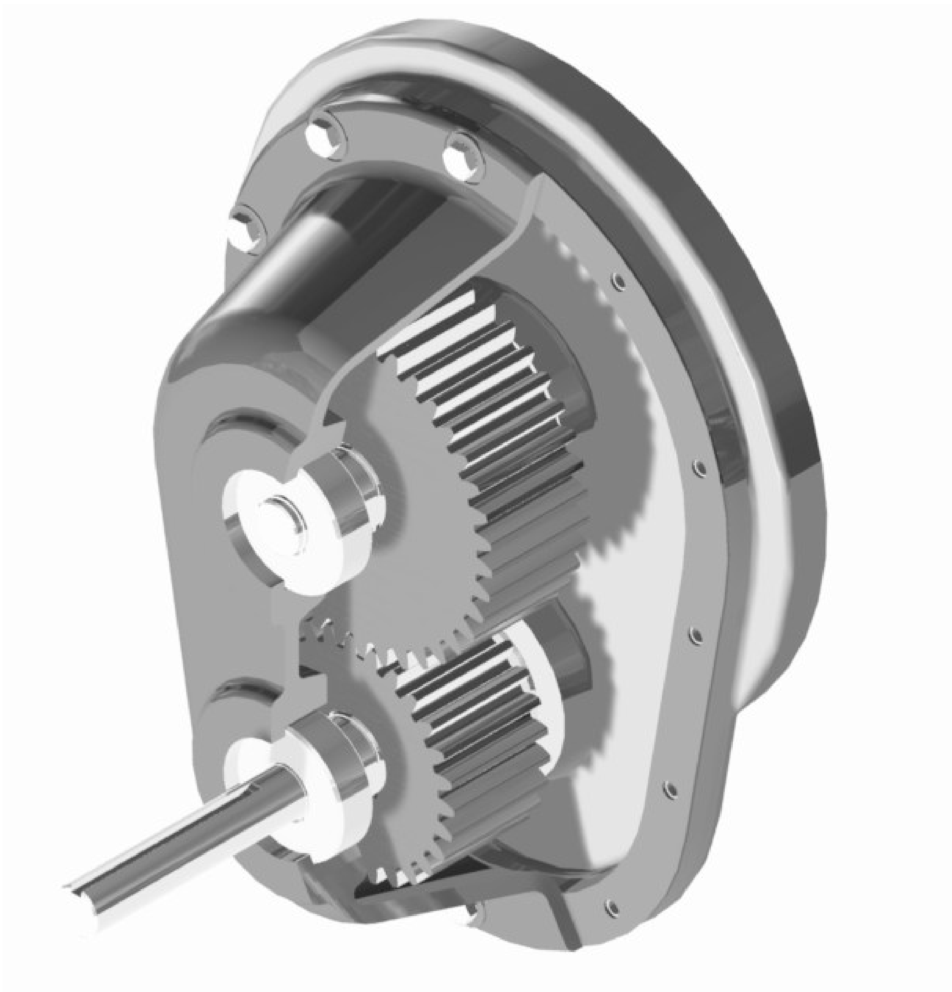
Mechanical Engineering Design with Pro/ENGINEER Release 2000i

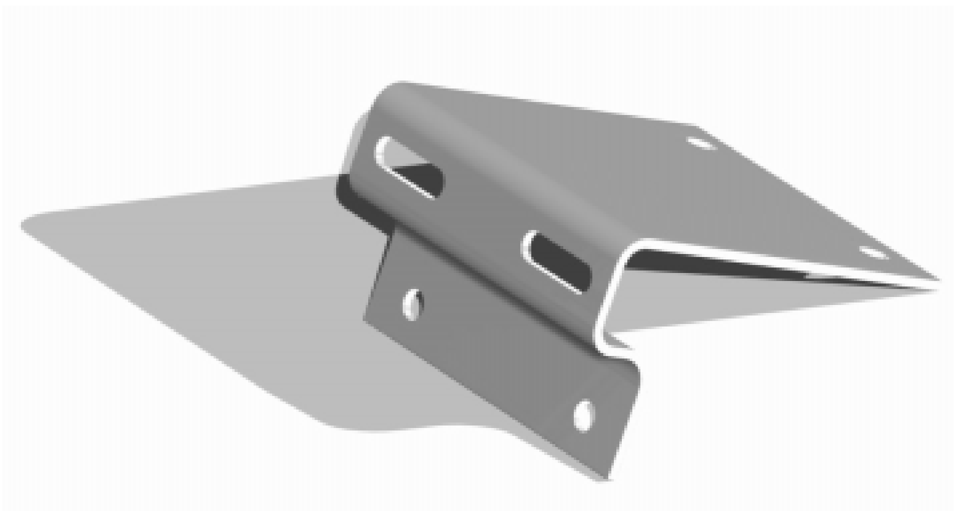


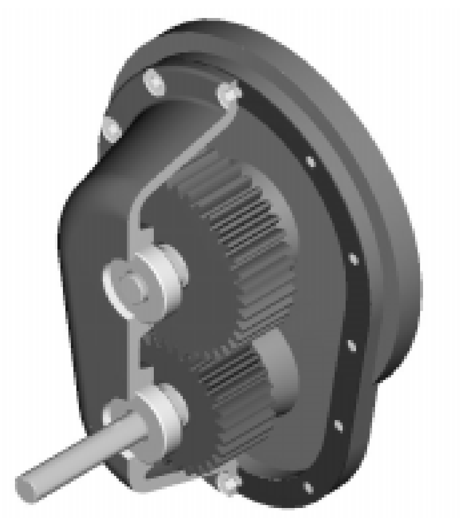
Dr. Mark Archibald

Grove City College
Mechanical Engineering

Please note: This file contains the first 23 pages of Chapter 2. The entire chapter is 53 pages long.

CHAPTER 2
BASIC FEATURE CREATION AND MODEL MANIPULATION





Chapter Objective:

- To teach students basic feature creation techniques.
- To teach students the importance of model structure.
- To teach students to manipulate and save model views.

The Pro/E Interface

The Main Window

Pro/E uses a graphical user interface that combines menus, toolbars, and windows to provide an efficient working environment. The active model appears in the graphics area of the **main window** (Fig 1).

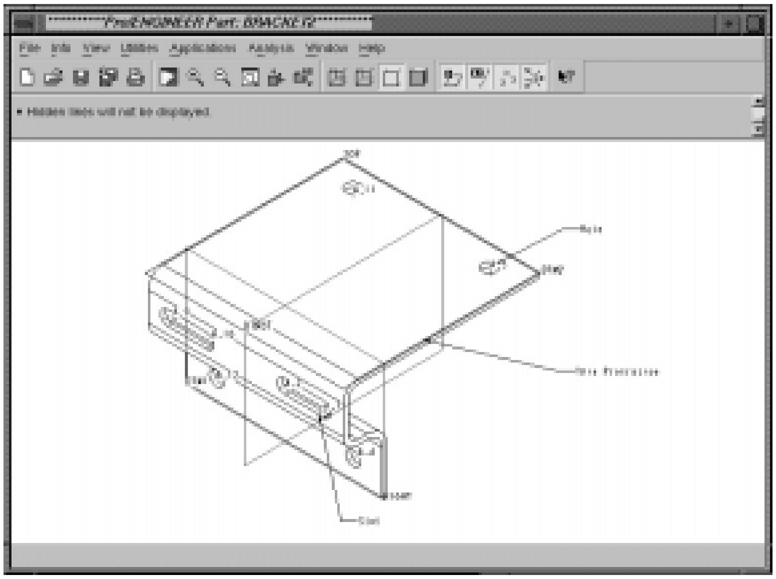


Figure 3 Main Window

The **header bar** at the top of this window displays the model type and name, and indicates that the model is active. (Active models display ***** before and after the model name.)

Figure 4 Header bar

The figure shows the header bar for a Pro/E part model named *bracket2*. The stars indicate that this window is active.

The **menu bar**, containing non-model specific menus, lies just below the header bar.

Figure 5 Menu Bar

Following is a brief description of the menu bar options. Most of these menus are treated extensively in later chapters.

File	File commands, such as set working directory, save, open, create new object, erase from RAM, delete either old versions or all versions of a file. **Note 1:** When Pro/E saves a file, it does not overwrite the previously saved version, but creates a new file with an incremented version number, such as frame.prt.4 . Old versions of model files must be deleted, or purged, periodically to prevent excessive numbers of files on disk. **Note 2:** When a window is closed (see window menu), the model remains in memory. To remove it from memory, use the erase command (be sure to save the file first!)
Info	Access model information commands, such as bills of material, model or feature information, parent-child relations, etc.
View	Access image viewing commands such as repaint screen, shade image, model orientation, model colors and lights.
Utilities	Set environmental variables, modify configuration options, set system colors, etc.
Applications	Select other applications, such a Pro/MECHANICA, Pro/SHEETMETAL, or Pro/NCPOST.
Analysis	Analyze the model and obtain measurements such as distance between entities, lengths and areas; curvature, etc.
Window	Activate, close or select a window. (Note that Pro/E can have multiple windows open at any time, but only one will be active. In order to work on a model, its window must be activated from this menu.)
Help	Activate on-line help, either general or context-sensitive.

A **toolbar**, with icons for frequently-used commands, follows the menu bar. It contains icons of frequently used commands, such as create new object, save model, open model, repaint screen, orient model, and blank datums, axes, points or coordinate systems. The toolbar is easily customized to contain icons for most Pro/E commands.

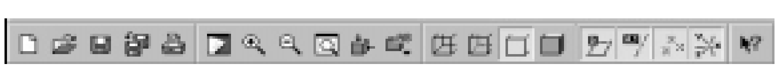


Figure 6 Toolbar

The **main graphics area** contains the Pro/E model, and is where most of the modeling work occurs. A **message area** is located just above the graphics area. Important information and prompts are provided in this area.

✔ ***Important:*** *Always check the message area to avoid missing important information.*

A **one-line help area** appears on the bottom line of the window. When the cursor is placed over a menu item or icon, a succinct description of the command is provided here.

When a model is opened or created, additional menus appear that are specific to the type of model. These menus will be discussed in the exercises. Most menus have multiple levels, which usually remain open as you move down the menu tree.

Model Tree

An additional window, called the **Model Tree** (Fig 2), also opens when a model is activated or created. The model tree displays the hierarchy of the model. For part models, features are listed in order, along with an icon indicating the feature class (solid feature, datum feature, or surface.) (By default, only components are shown for assembly models, although assembly and part features can be easily shown on the tree.) Additional columns can be added for more information. For example, it is often useful to see feature IDs. The model tree is a "roadmap" for your model. Good designers quickly learn to use the model tree frequently and effectively.

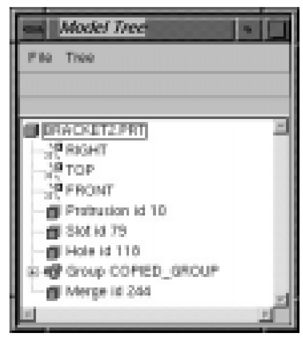


Figure 7 Model Tree

Usually, model features can be selected from either the model tree or the model itself. Also, if the highlight option is activated (it is by default), the model feature is highlighted when the cursor is placed over a feature on the model tree. This is very useful for identification of features in complex models.

When Pro/E is started, an additional window may appear at the top of the screen. This window is called the **application manager**, and is used to navigate between various applications and windows. It contains a button for each open window. To pop a window to the foreground, simply click on its button. When ending your work session, first exit Pro/E, then exit the application manager.

Figure 8 Application Manager

Pro/E model views can be manipulated with the mouse easier than from the view menu. **Mouse view control** -- zoom, spin, and pan -- is accomplished by pressing the control key while simultaneously pressing one of the mouse keys and dragging.

Mouse View Control	
Zoom	Press **Ctrl** and the **left mouse key** while dragging right or left.
	For a window zoom, press **Ctrl** and **left click** the mouse on opposite corners of the zoom box.
Spin	Press **Ctrl** and **center mouse key** while dragging.
Pan	Press **Ctrl** and **right mouse key** while dragging

Good Design Practice

Design Intent

To effectively use Pro/E as a design tool, designers must not only know and understand the software functionality, they must also know how to build models that behave as desired during modifications or downstream applications. This is known as **capturing design intent**, and is extremely important for reducing design cycle time. Unlike many CAD programs, Pro/E requires the designer to think beyond basic geometry. Most parts designed in Pro/E will be modified, sometimes drastically. Most will also be used with other applications, such as Pro/MECHANICA, NC machining, mold design, injection molding simulation, stereolithography, etc. If the part model is poorly constructed, modifications will be difficult, perhaps requiring that the part be completely remodeled. Also, much time may be required to repair or remodel the part prior to using any of the downstream applications. Some of the prime benefits of the Pro/E package can be nullified by poor modeling.

Throughout this book, emphasis is placed on capturing design intent. Examples and tutorials show good modeling practice and illustrate how the design intent is realized. Emphasis is placed on understanding the model and what it will be used for prior to modeling. The importance of model structure, especially parent-child relationships, is treated extensively. The Pro/E student should strive not just to understand how to obtain desired geometry, but how to obtain the desired geometry with a robust model.

Good practice starts at the feature creation level, where parameters and parent-child relationships are defined. The structure of the model -- as reflected in the model tree -- is the second tier in developing good models. Building robust assemblies is the third tier. Attention, planning, and foresight will ensure that good design practice is obtained at all three levels.

Model Structure

Understanding how to structure a model is the first step to good design practice. Pro/E models are hierarchical. Each feature (except the base feature) references earlier features in the model. When the model is changed in any way, features regenerate in sequence. If the model has changed in such a way as to delete the references for a feature, the regeneration process will fail when it gets to that feature. (What to do then is covered in Chapter 6.) Thus it is very important to understand the relationships between model features. These are known as parent-child relationships.

Pro/E models are also parametric. Model geometry is defined by a set of parameters. The two most common ways to define parameters are dimensions and alignments. When a dimension is defined, it becomes a parameter of the model. It is important to consider this when creating geometry, as the model parameters define how the model will behave when modified. Alignments indicate that new geometry should be aligned with existing features (essentially a dimension that always has a value of zero.)

Every Pro/E model should start with **default datum planes**. These are three orthogonal datum planes that provide references, either directly or indirectly, for all subsequent features in the model. Recall that datum planes are infinite planes and have both a red and a yellow side. Datum planes provide excellent references for other features, and frequently models will contain many of them. However, the default datum planes are special, in that they provide a three-dimensional anchor for the entire model.

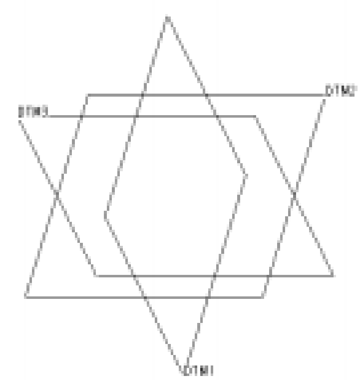


Figure 9 Default Datum Planes

The first feature in ***any*** *model should be default datum planes.*

Thought should be given to the order of subsequent features. Sometimes the order is obvious: to model a cylindrical shaft with a keyway, create a cylindrical protrusion followed by a cut. Other times careful thought is required to ensure a robust model results. In addition, there are usually many different ways to obtain a particular geometry. Learn as many of these methods as possible, then select the ones that will best capture the design intent.

As features are added, parent-child relationships are created. For example, when a hole is placed in a flat plate, three references are required: the surface on which the hole is placed and two edges or surfaces used to define the location of the hole on the placement surface. All three references could be to a single feature -- the flat plate, or they could be to different features, say the flat plate, a datum plane and the surface of a cut in the plate. In the former case, the hole would have only one parent. In the latter case, the hole would have three parents. It is usually good to minimize the number of parent-child relationships in the model. Thus the first method is usually, but not always, the best. It is always important to know what you are using for references during feature creation.

In short, a Pro/E model is comprised of an ordered set of features held together in a web of dimensions, alignments, and references. The web defines the parent-child relationships between features.

Data Base Management

Although model structure is the most important aspect of good design practice, data base management is also important. The **File** menu contains most of the commands needed. Selected commands are discussed here, along with some tips for good file management.

New Creates a new model. A dialog box opens with the type of model (part is the default) and the name of the new model (Fig 3). Select the model type by clicking on the appropriate button, and enter the new name. Note that only the base name should be provided -- Pro/E will append the correct extension based on the type of file (For example, if the model is a part and *bracket* is entered, the actual file name is bracket.prt.1.) There is also a button called **copy from**. This loads an existing model into the new model (without affecting the old model.) It is very useful if the new part is similar to an existing part. The tutorials show how this feature is used to expedite model creation for all models.

Open Opens an existing model. A dialog box appears showing file names of objects in the current working directory. Click on the desired file and click the **open** button. Note that the Type box permits files to be filtered by type, such as part, assembly, manufacturing, etc. Icons provide shortcuts to navigate through directories.

Working Directory Sets the working directory. The working directory is the directory that Pro/E uses to look for files or to write new files (unless otherwise specified.) At the start of each work session, set the working directory to ensure the files are read and written correctly.

Erase Erases models from memory. When a window is closed, the model remains in memory, or **In Session**. To clear memory, use the erase function. Two options are available: **Current** or **Not Displayed**. Current will erase the model in the active window from memory. Not Displayed brings up a dialog box of all items in memory that are not displayed on the screen -- select all or some for erasure. Note that files are NOT saved prior to erasing from memory.

Delete Deletes files from the hard drive. The two options are to delete **All Versions** or delete **Old Versions**. The latter purges all versions of the file except the most recent.

Save Save the current model to the hard drive. Note that you are prompted for the object to save, which must exist in session (in memory.) The current model is the default.

Other options include **Save As** (save model as a different object), **Backup** (save to a different directory), and **Rename** (change the name of the model.)

Tips for good data management...

1. Create a new directory for each new project, with subdirectories as needed. Pro/E can generate a large number of files and good project organization is imperative.

2. Create subdirectories for each Mechanica analysis and each manufacturing model.

3. Purge old versions frequently to prevent using excessive disk space. One approach is to always purge prior to saving -- then you will always have just one backup version. (Clearly, do not delete old versions if they are needed for archives.)

4. Typing "*purge*" from a system window will delete old versions of all files in the directory.

5. Remember that closing a window removes the model from the screen, but does not erase it from memory or save it. To clear memory, use the **Erase** command.

6. Always start Pro/E from the home directory, then use the working directory command to change to the desired directory.

Pro/E Customization

There are many ways in which Pro/E can be customized. While some of these methods should only be attempted by advanced users, many others are both important enough and simple enough for the novice Pro/E user to use. The most important is the configuration file, typically called config.pro. This file should reside in the same directory from which Pro/E is started (the home directory.) It is an ASCII text file that contains configuration commands. When Pro/E is started, it looks for this file, and automatically executes all the commands. This configures Pro/E for an individual designer. The config.pro file can be modified during a session, but must be loaded before changes take effect. The **Edit Config** and **Load Config** commands accomplish this (they are found under **Preferences** on the **Utilities menu**.)

It is also quite simple to customize the toolbar. While the default toolbar is usually fine for beginners, experienced Pro/E users may find that some additional commands are used very often. It is convenient to place these commands as icons on the toolbar. In fact, several different toolbars can be created, each with its own set of commands. To modify the toolbar, position the cursor on the toolbar and press the right mouse button.

Pro/Help

This manual provides an introduction to Pro/E and is designed to get the student productive as rapidly as possible. However, no tutorial or lab manual can include all the details of the many Pro/E commands. Students should form the habit of using the on-line help to obtain more information. On-line help, available through a web browser, is accessed in several ways. To obtain access to all on-line manuals, use the **Pro/HELP** command on the **Help** menu. This method permits browsing through any help manual desired. To obtain context-sensitive help on a particular topic, use the **What's This** command on the **Help** menu. The cursor becomes a question mark. Select a command from any menu, and the help screen for that topic appears. Alternately, position the cursor over any command and press the right mouse button to bring up context-sensitive help.

Develop the habit of using on-line help on a regular basis. This helps beginners master Pro/E much more quickly.

Basic Feature Creation

A sound understanding of feature creation is crucial for effective modeling in Pro/E. Frequently several feature types can be used to create the desired geometry. The designer must choose the types that best capture his or her design intent. Then he or she must know the steps required to create each feature.

This section presents an overview of four of the most fundamental solid features -- protrusions, cuts, slots, and holes. (Solid features either add or remove solid "chunks" of material to the model.) The intent is to familiarize the student with the nature of each type of feature so that appropriate choices can be made regarding which to use for a particular task. The lab exercises demonstrate implementation steps for each feature. Subsequent sections address additional feature types, and are accompanied by appropriate lab exercises. A separate section describes sketcher.

☛ **Hint:** Use context-sensitive online help to learn more details about each of the following features.

Protrusions

A **protrusion** adds solid material to the model. Most protrusions are sketched features, meaning that 2D geometry is first created using sketcher, and then swept through space in such a way as to create a solid. There are several types of protrusions, depending on how the sketch is moved to form the solid. A brief description of the protrusion menu picks:

Extrude	The 2D sketch is moved in a straight line perpendicular to the sketch plane. The resulting solid is prismatic.
Revolve	The 2D sketch is rotated about an axis, through any desired angle. If the angle is 360°, the resulting solid is axisymmetric
Sweep	The 2D sketch is moved along a 3D path called a trajectory. The resulting solid may be complex, but will always have a constant cross-section.
Blend	Several 2D sketches are used, located on parallel planes separated by distances prescribed by the user. The resulting solid changes cross-sectional shape as it goes from one sketch to the next. The cross sections may have any shape, but must be comprised of the same number of segments. The resulting geometry is often complex.
From Quilt	The solid is generated from a surface quilt (several connected surfaces.) This type of protrusion is not sketched, and will be discussed in Chapter 12.
Advanced	Several advanced methods of creating protrusions are also available, such as the **Variable Section Sweep** and the **Swept Blend**. These features are quite powerful, but somewhat more complex than the basic protrusions. Some of these will be treated in later chapters.

Cuts

A **cut** removes solid material from the model. This menu option is only available if the model contains solid geometry. Cuts are created exactly like protrusions, except that where the protrusion adds material, the cut removes material. The menu options are identical to those for protrusions -- **Extrude**, **Revolve**, **Sweep**, **Blend**, **From Quilt**, and **Advanced**. Like protrusions, most cuts are sketched features. Once the section has been sketched, Pro/E prompts the user for the which side of the section (sketch) should be removed.

Holes

Holes are pick-and-place features, meaning that they are not sketched. A hole always removes material from the model, and always has a circular cross section. To create a hole, Pro/E only needs to know on what surface the hole is to be placed, the location of the center of the hole on the surface, the hole diameter, and the depth of the hole. Prompts are provided for all these items.

There are two types of holes: **Straight** and **Sketched**. A straight hole is always cylindrical in shape. A sketched hole is similar to a 360° revolved cut in that the axial cross section can be sketched. Counterbores, countersinks, tapered holes, etc. can be created more easily with the sketched hole feature than with a revolved cut.

Sketcher and Intent Manager

Sketcher, along with an enhancement called Intent Manager, is used for creating most Pro/E features. It is a powerful tool for generating 2D sections, which are subsequently used to generate 3D geometry. <u>Sections</u> are the resulting 2D entities which are produced by sketcher. Typically, sketcher is invoked within a feature creation sequence, such as a protrusion, cut, or slot. (It can also be used independently to create and save sections for later use, but that functionality is not discussed here.) Mastery of sketcher is essential for efficient and effective use of Pro/E. Fortunately, it is easy and intuitive – and smart. This section provides a brief overview of sketcher. Chapter 5 describes sketcher assumptions and use in much greater detail.

Sections are created by sketching a rough approximation of the desired geometry. Dimensions, alignments, and sketcher assumptions refine the sketch. The values of each dimension can then be modified to obtain the exact desired geometry.

Sketcher automatically makes assumptions as the sketch is created. The Intent Manager automatically places “weak” dimensions and alignments. If a user understands how sketcher and intent manager work, he or she can usually ensure that most of the automatic dimensions, alignments, and assumptions are correct. (Those that are not correct can easily be changed, however.) This is the key to using sketcher effectively.

In order to create solid geometry, a section must be located with respect to the part. This is done by first defining a sketching plane, then by aligning or dimensioning the section to the part. The <u>sketch plane</u> is defined prior to entering sketcher mode. It can be either a datum plane or a flat surface. The section is sketched on this plane. A point or an edge of a sketch can be forced to always lie directly on an existing part entity, such as a datum plane, edge, side, or curve. This is called <u>alignment</u>, and is one way to locate the section to the part within the sketch plane. Alternatively, dimensions can be placed between the sketch an part entities. In either case, sufficient alignments and dimensions must be provided to locate the section in both the vertical and horizontal directions.

Intent manager automatically creates alignments and dimensions. However, the user must tell intent manager what references should be used to do this. A sketch can not be started until sufficient references are selected. Choosing references is important! Choosing correct references will ensure that desired alignments and most dimensions are obtained automatically. This affects not only the current section, but also parent/child relationships within the model. Some guidelines for choosing references:

- ✔ *Any and all* part entities to be aligned to the sketch must be selected.
- ✔ Part entities to be used for dimensioning should be selected.
- ✔ Choose references to obtain desired parent/child relationships. For example, choose references that all belong to one feature to minimize the number of parents.

Sketching can begin once references are defined. Sketches consist of lines, arcs, circles, points, and several advanced entities such as splines. Geometry types are selected either from the **GEOMETRY** menu or from a pop-up menu activated by the right mouse button. Note that entities snap to the references. Also, lines that are nearly vertical or horizontal snap to vertical or horizontal respectively, and symbols V or H appear. These are two of the assumptions that sketcher makes. (If you want a line at a very small angle from the horizontal, sketch it at a large angle, and modify the angle value later.) Other assumptions are described below. As each segment of the sketch is completed, dimensions appear in white.

☛ **Note**: Do not confuse a dimension (a parameter in the Pro/E model) with its value. It is very important to create the correct dimension scheme, but values can be changed very easily – within sketcher or later, in part mode.

These are “weak” dimensions, because they were created by the Intent Manager. Frequently some of these need to be replaced with more desirable dimensions. Desirable dimensions can be strengthened – that is, the user tells Pro/E that these dimensions should not be deleted. “Strong” dimensions are shown in yellow. New dimensions can then be created. As they are created, weak dimensions are removed. User-created dimensions are always “strong.” If a new dimension conflicts with an existing strong dimension, Pro/E asks which dimension should be deleted.

Understanding assumptions is important for effective sketcher use. Typically, assumptions are obvious during a sketch because the pointer snap to the appropriate entity and a symbol for the assumption appears. Sketcher assumptions are summarized below:

Assumption	Description	Sym.
Equal radius	Circles or arcs sketched with approximately equal radii are assumed to have exactly the same radii. The radius snaps to the assumed value.	R (index)
Symmetry	Entities approximately symmetric about a sketched centerline are assumed to be symmetric. Vertices snap to symmetric positions.	→
Horizontal & vertical lines	Nearly horizontal or vertical lines are assumed to be so. Lines snap to horizontal or vertical.	H or V (index)
Parallel or Perpendicular lines	Lines nearly parallel or perpendicular to existing lines are assumed parallel or perpendicular. Lines snap to parallel or perpendicular.	// or ⊥ (index)
Tangency	Entities sketched approximately tangent to each other are assumed tangent. Entities snap to tangency.	T (index)
Equal segment length	Lines of approximately the same length are assumed to have the same length. Line snaps to length.	L (index)
Point entities lying on other entities	Point entities that lie near other entities (lines, arcs, circles) are assumed to lie on them. Point snaps to entity. (Note: point entities include end points of lines and arcs and center points of arcs and circles.)	○
Equal coordinates	Center points of arcs and circles with nearly the same X and Y coordinates are assumed to have the same coordinates. Centers snap to X or Y coordinate.	▌ or ▬ (pairs)

Sketcher assumptions can be used very effectively during sketching to quickly obtain the desired section. If an assumption needs to be avoided, exaggerate the sketch. For example, place two circles that should lie near, but not on, the same horizontal line well away from each other. This forces sketcher to place a dimension rather than make an assumption. The value of the dimension can be modified later to any desired value (even zero, but that is poor practice!)

After the sketch is complete, the values of the dimensions must be modified, and the sketch regenerated. Additions, deletions, and further modification may then take place.

Some rules of thumb for sketcher:

- ✔ Choose references carefully in order to achieve desired alignments, dimensions, and parent/child relationships.
- ✔ Exaggerate the sketch – avoid very small entities and undesirable assumptions. Use **Modify** to achieve the desired geometry.

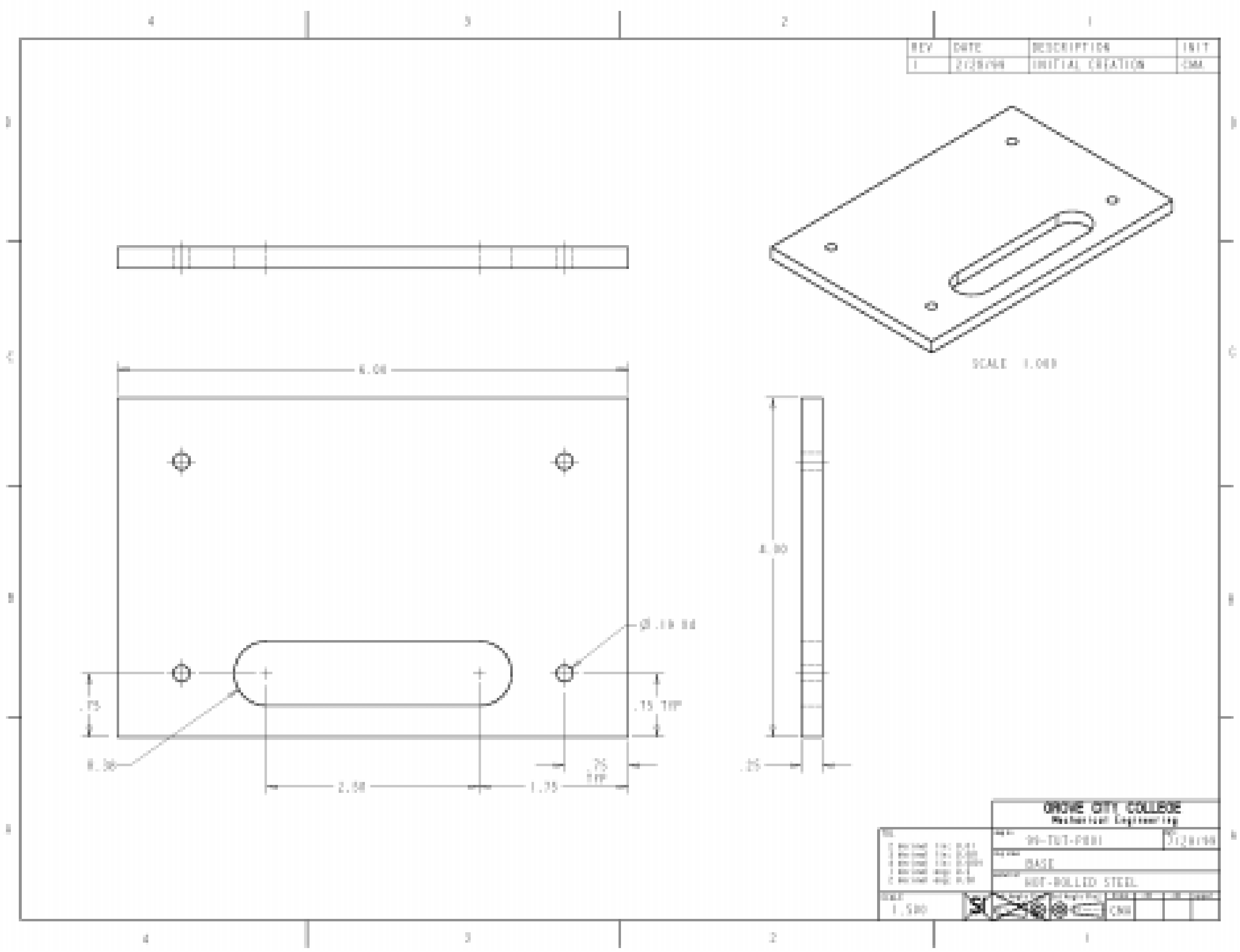
Exercise 2.1 Base

Objective: To introduce students to fundamental feature creation techniques, including extruded protrusions, slots, and holes.

This exercise involves modeling the base part shown below. The best way to learn Pro/E software is to dive right in and create a part model. That is exactly what this exercise involves.

☛ Note 1: Prior to starting this lesson, create a new directory called *tutorial*. (Do this in a system window.)

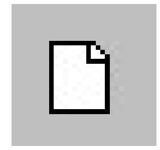
☛ Note 2: Strive to complete the exercise as presented, however, explore the menus and toolbar to become familiar with their functionality.



1. Change the working directory to the *tutorial* directory. Select **File>Working Directory**.

   Select the *tutorial* directory to highlight it and select **okay**.

2. Create a new part named *base*. Select the create new object icon (blank paper). (Or use **File>New**.)

   Create New

   Note that the radio button for part is selected by default. Enter the name *base* and select **OK**.

3. Create default datums. From the Part menu, Select **Feature>create>datum>plane>default**. The datums appear.

4. Create the base feature. Select **Feature>Create>Protrusion**.

   Accept the defaults of **Extrude** and **Solid**. Select **Done**.

   Accept the default of **One Side**. Select **Done**.

   At the prompt to create a sketching plane pick on DTM3. (**Note:** To select a datum plane pick either on the name tag or the border of the datum plane.) Accept the default direction by selecting **Okay**.

   For the second reference select **Top** and then pick on DTM2.

   The model now reorients and Pro/E enters sketcher mode. The first task is to specify references for the sketch. DTM2 and DTM3 will be used for references.

   Verify that **Specify Refs** is highlighted and pick **DTM1** and **DTM2**.

   To create the section geometry, select **Sketch** (Note: the middle mouse key also selects **Sketch**.)

   For this example, it is important to have the plate symmetric about DTM2 and DTM3. To ensure this symmetry, create two centerlines, aligned to DTM2 and DTM3.

   From the **LINE TYPE** menu, select **Centerline**, accepting the default of **2 Points**. Pick once on DTM2 to begin the centerline. Note that this becomes a pivot point for the line. Drag the line so that it aligns with DTM2 (a pair of small solid rectangles appear when aligned.) Pick a second time on DTM2. The first centerline is created. Repeat for the second centerline, using DTM3 as the alignment reference.

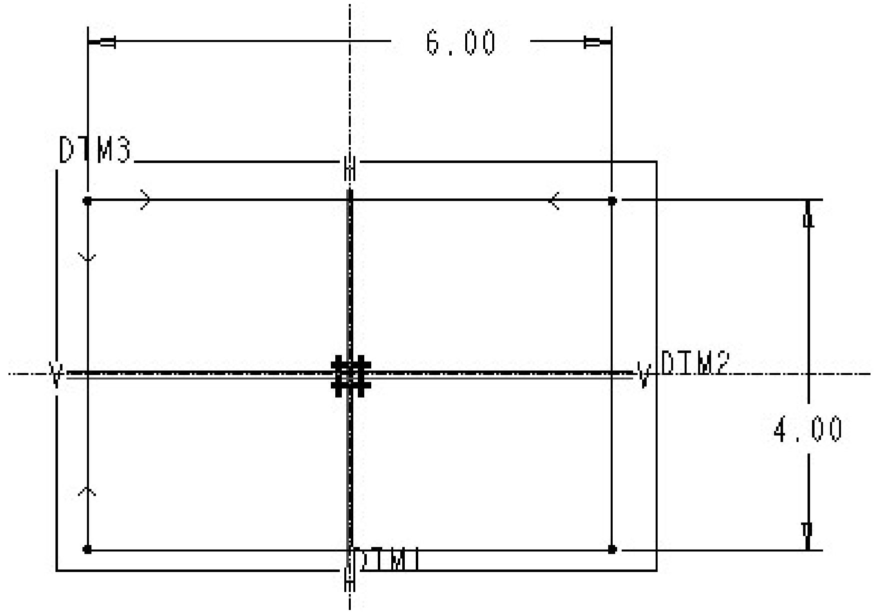


Figure 12 Sketch of rectangular base feature.

Sketch a rectangle symmetric about the two centerlines. From the **GEOMETRY** menu, select **Rectangle**. Use the grid and pick two diagonal corners of a rectangle symmetric about the two centerlines. Note that the rectangle snaps to symmetry. Also, small arrows at the vertices indicate the symmetry constraints. (Do not sketch a square!)

Dimensions for the width and height of the rectangle appear in white. These are called weak dimensions (system-supplied dimensions.) The values for these dimensions may be quite large at this point. We will now modify those dimensions.

From the **SKETCHER** menu, select **Modify** . Pick the horizontal dimension and enter a value of ***6.0***. Pick the vertical dimension and enter a value of ***4.0***. To update the model to conform to these new dimensions, it must be regenerated.

From the **SKETCHER** menu, select **Regenerate**. The sketch is now complete.

From the **SKETCHER** menu, select **Done**.

The depth of the solid rectangle must now be defined. From the **SPEC TO** menu, select **Blind>Done**. Enter a value of ***.25***.

To preview the base select **Preview** from the extrude dialog box. Press the control key while dragging the mouse with the middle mouse key depressed to spin the model and view it from different angles. Select **OK** from the *PROTRUSION:Extrude* dialog box. The protrusion is now complete.

5. Create a cut in the protrusion. Select **Feature>Create>Cut**. Accept the defaults of **Extrude** and **Solid**. Select **Done**.

Accept the default of **One Side**. Select **Done**.

At the prompt to create a sketching plane pick on the front side of the base. Ensure that the arrow points into the base. If so, select **Okay**. If not, select **Flip** and check that the arrow flips direction into the base, then select **Okay**. (The slot feature will begin on the sketching plane and extend into the direction shown by the arrow. Clearly, in this case we want the slot to extent into the part, rather than out into empty space.)

For the second reference select **Top** and then pick the top side of the part.

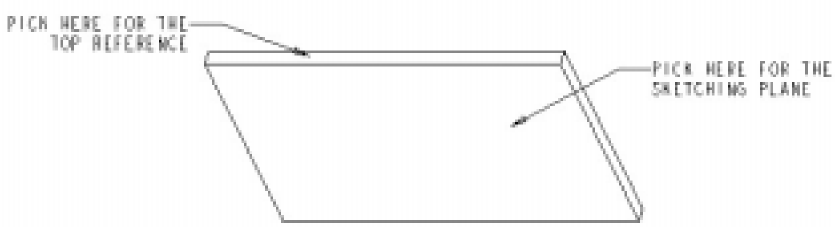
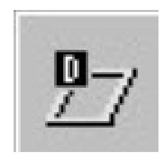


Figure 13 Orientation references for sketching the slot feature.

Turn off datum plane display for clarity. Select the *Datum Planes on/off* icon from the toolbar to toggle the display.



Datum
Planes

Verify that **Specify Refs** is highlighted and pick the bottom edge of the part and the right edge of the part.

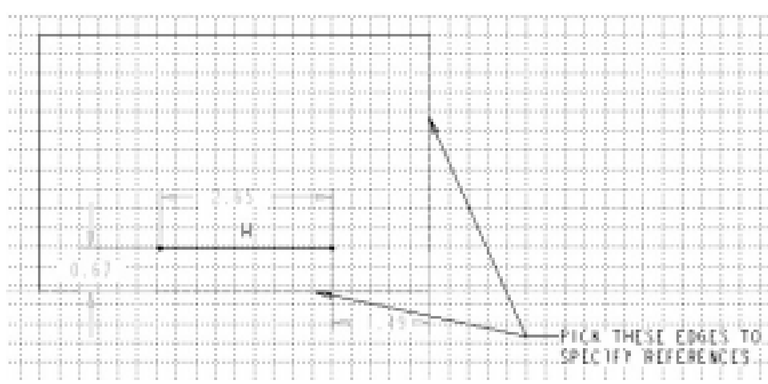


Figure 15 Sketcher references and the first line

Select **Sketch**. Accept the defaults for **Line** and **Geometry**. To sketch a single line as shown (do not worry about dimensions yet) use a left pick to start and end the line, and middle pick to terminate drawing lines. Note the *H* symbol, denoting a horizontal line.

Select **Arc|Tangent End** and create the 180° arc by a left pick on the end of the line followed by a left pick when the sketch indicates 180°. The arc will snap to 180°. Note the *T* indicating a tangency constraint and the two small solid rectangles representing the 180° constraint.

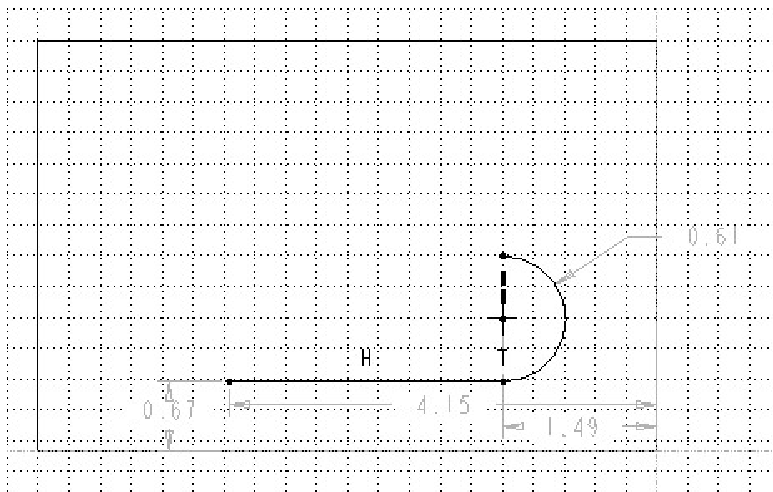


Figure 16 Line and arc showing constraint symbols (Horizontal, Tangent, and 180° arc.)

Select **Line** to sketch a second straight line. Make sure the second line snaps to the same length as the first line -- look for the L_1 symbol indicating the equal length constraint. Finish with a second tangent arc. It should complete the loop.

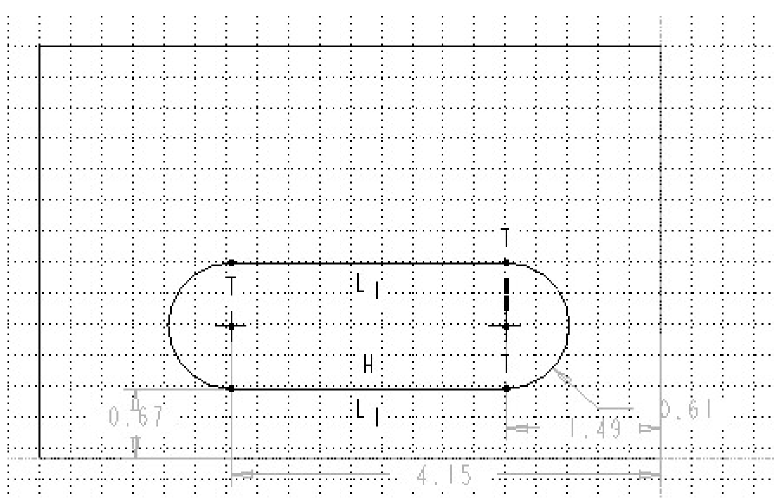


Figure 17 Completed sketch, prior to dimensioning.

Shortcuts ☛ In sketcher, the middle mouse key toggles between the **Line** and **Arc** menus.

☛ The right mouse key initiates a pop-up menu with common sketcher commands.

The default dimension scheme is not what is desired, so create new dimensions to replace the undesired ones. Select **Dimension|Normal**. Pick the two center marks (center of the arcs), place the pointer below the part and click the middle mouse key to place the dimension. Select **Horizontal** and the dimension appears.

Now select the center mark of the left arc and select the bottom edge of the part (or the dashed brown line representing the reference.) Middle mouse pick to the left of the slot to place the dimension. The remaining default dimensions are acceptable.

To move any dimension, select **Move** and drag to the new position.

Add axis points at the centers of the arcs. Sketched axis points will become datum axes when the slot feature is complete. Select **Sketch>Adv Geometry>Axis Point**.

Pick the center mark for each slot. A small **x** indicated the axis point is placed.

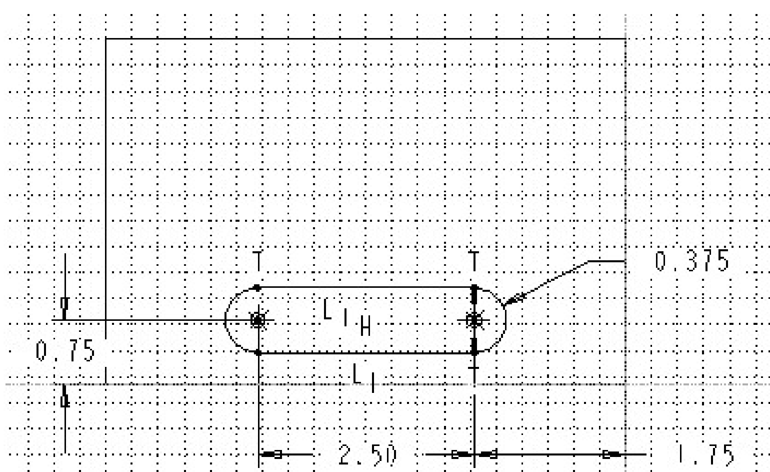


Figure 18 Completed sketch of cut after modifying dimensions and regenerating.

Modify the dimensions. Select **Modify**.

Pick the dimension from the right edge to the center and enter ***1.75***.
Pick the center-to-center dimension enter ***2.5***.
Pick the dimension from the bottom edge to the center mark and enter ***.75***.
Pick the radius and enter ***.375***.

Select **Regenerate**. The section is now complete. Select **Done**.

For the direction, the arrow should point toward the inside of the cut. If it is correct, select **Okay**, otherwise select **Flip>Okay**.

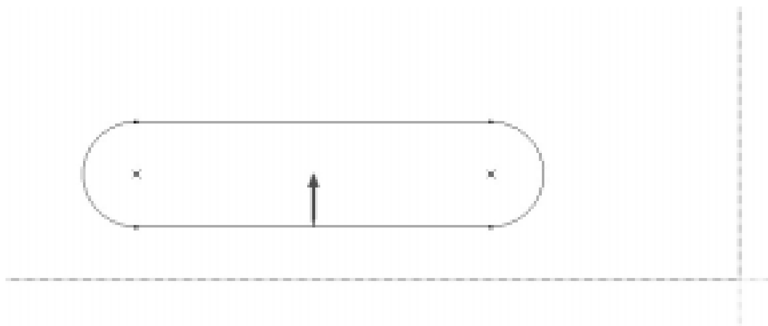


Figure 19 Correct arrow direction for cut feature.

The *SPEC TO* menu defines the depth of the slot. Select **Through All** and **Done**.

Select **Preview** to review the slot, then select **Okay** to accept it. Shading the part by selecting the **Shaded View** icon may be helpful.

Note the model tree and how the features appear in the model tree.

6. Create four holes for mounting feet to the base. Select **Feature>Create>Solid>Hole**. Select **Straight** and **Done**.

   Accept the default **Linear** and select **Done**.

   The placement plane is the surface where the hole starts. Spin the model and select the top surface of the part approximately 3/4" from the corner.

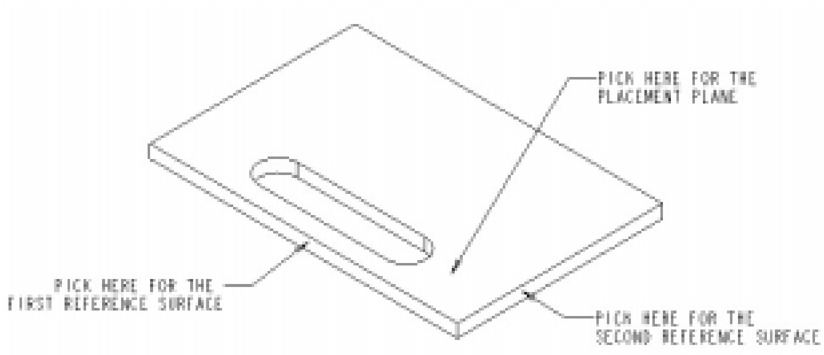


Figure 20 References for hole placement.

Two references must be selected to provide dimensions to locate the hole on the placement surface. The two side surfaces forming the corner of the plate should be chosen. Spin the model and use **Query Select** to pick the first of these surfaces. Accept it when the entire surface highlights, not just a single edge.

> ☛ **Recall:** right mouse key initiates Query Select; left mouse key picks, right mouse key goes to next entity, middle mouse key accepts)

Enter ***.75*** for the distance. Query select the second side and enter ***.75***.

Accept the default of **One Side** and select **Done**. From the **Spec To** menu select **Through All** and **Done**. Enter the diameter **.25**.

Preview the hole and select **OK** from the dialog box to complete the hole feature.

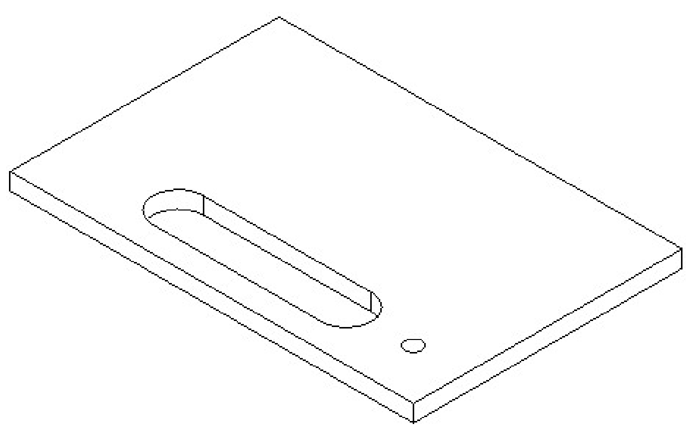


Figure 21 The completed hole feature

7. Turn on datum plane display by selecting the *Datum Planes on/off* icon.

   Mirror the hole to create dependent copies. Select **Feature>Copy>Mirror|Select|Dependent** and **Done**.

   Select the hole (Query Select may be helpful.) Select **Done Select** and then **Done**.

   The command line prompts for a plane or datum to mirror about. Query select DTM2. The second hole now appears.

   Mirror both holes about DTM1. Select **Feature>Copy>Mirror|Select|Dependent** and **Done**. Select both holes. Select **Done Select** and then **Done**.

   The command line prompts for a plane or datum to mirror about. Select DTM1. The second set hole now appears.

   Selecting the **Dependent** option, all three of the copied holes have the same parameters as the initial hole. Thus, if the hole is modified, say to change the diameter, all four holes change together. Subsequent lessons will address Independent copies.

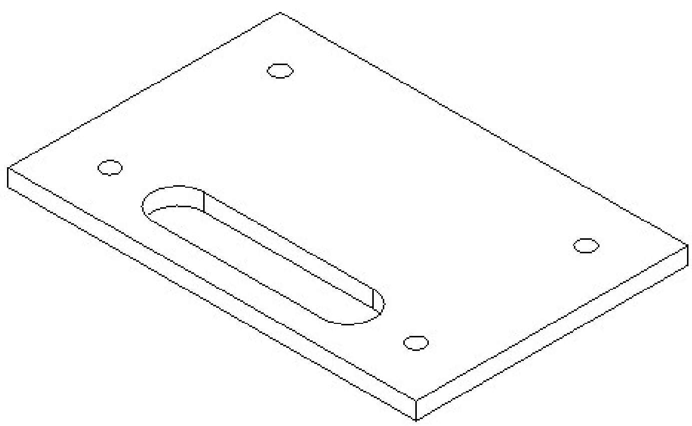
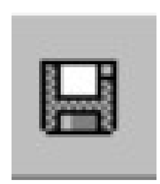


Figure 22 Holes after using *Copy>Mirror* twice.

8. Save the part. From the toolbar, select the **Save** icon. Press **Enter** to accept the default part name.



Save icon

9. Review the model tree. From the PTC Application Manager window, select **Model Tree**. The model tree window appears. Note the features on the tree: the three default datums, the base protrusion, the slot, the hole and the two mirror features (listed as Group COPIED_GROUP.) Picking the **+** sign by either copy feature explodes the feature to show the individual elements copied. Additional model tree functionality is discussed in later lessons.

10. Modify the hole diameters. From the **PART** menu (select **Done** to return to this menu) select **Modify**.

    Pick any one of the four holes. The two linear placement dimensions and the diameter dimension appear. Pick the .25 diameter dimension and enter ***.1875***. Select **Regenerate**.

    All four holes change to the new diameter. Any model parameter can be easily changed in this way. Note that our design intent – ensuring that the four holes remain the same diameter and the same distances from their respective corners – is captured in this model.

11. Experiment with the icons on the toolbar. Try shaded image, hidden line, and no hidden views. Blank the datum planes and axes. The **Saved View** icon includes only one view, the default view. The next lesson will include saving additional views.

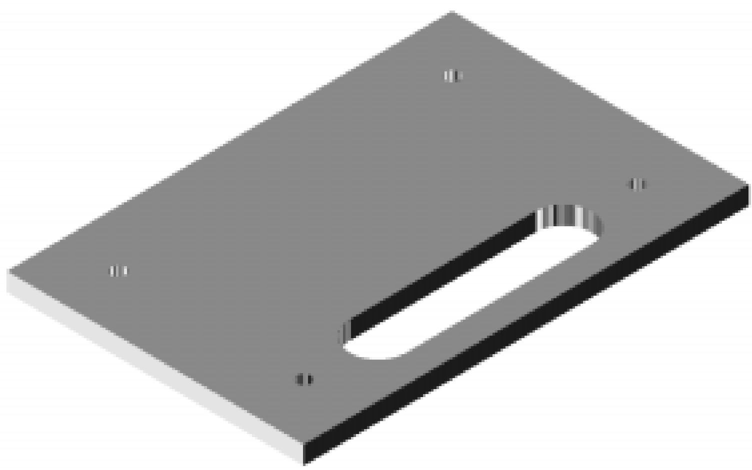


Figure 24 The completed part.

➢ End Exercise 2.1