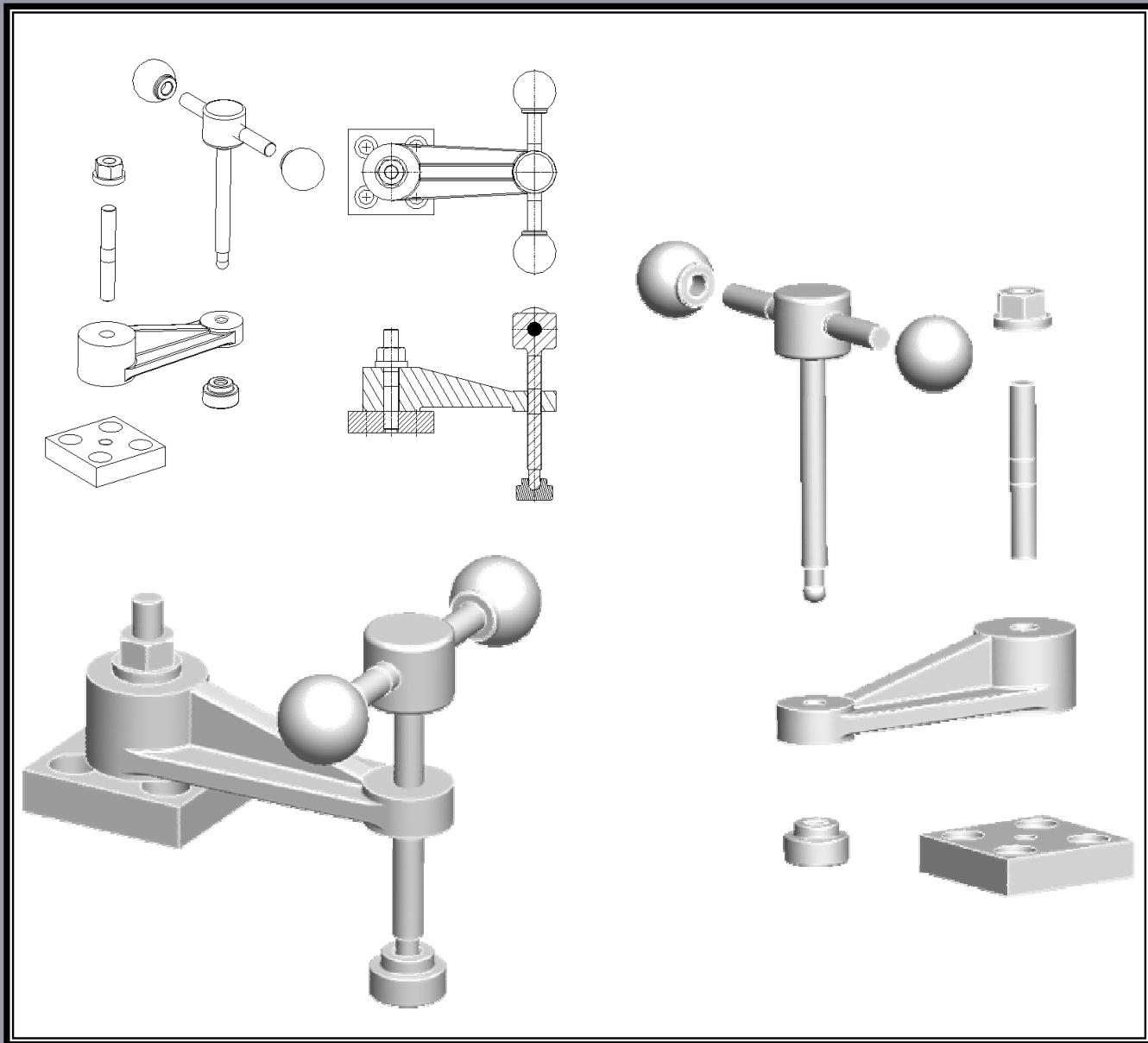


Introduction to Pro/ENGINEER® Wildfire™ 2.0

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Lesson 3 Extrusions

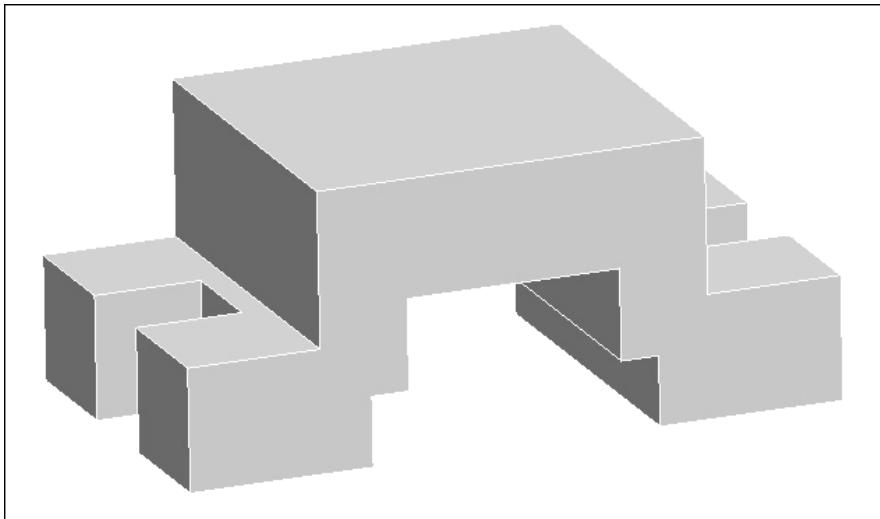


Figure 3.1 Clamp

OBJECTIVES

- Create a feature using an **Extruded** protrusion
- Understand **Setup** and **Environment** settings
- Define and set a **Material** type
- Create and use **Datum** features
- Sketch protrusion and cut feature geometry using the **Sketcher**
- Understand the feature **Dashboard**
- **Copy** a feature
- **Save and Delete Old Versions** of an object

Extrusions

The design of a part using Pro/E starts with the creation of base features (normally datum planes), and a solid protrusion. Other protrusions and cuts are then added in sequence as required by the design. You can use various types of Pro/E features as building blocks in the progressive creation of solid parts (Fig. 3.1). Certain features, by necessity, precede other more dependent features in the design process. Those dependent features rely on the previously defined features for dimensional and geometric references.

The progressive design of features creates these dependent feature relationships known as *parent-child relationships*. The actual sequential history of the design is displayed in the Model Tree. The parent-child relationship is one of the most powerful aspects of Pro/E and parametric modeling in general. It is also very important after you modify a part. After a parent feature in a part is modified, all children are automatically modified to reflect the changes in the parent feature. It is therefore essential to reference feature dimensions so that Pro/E can correctly propagate design modifications throughout the model.

An **extrusion** is a part feature that adds or removes material. A protrusion is *always the first solid feature created*. This is usually the first feature created after a base feature of datum planes. The **Extrude Tool** is used to create both protrusions and cuts. A toolchest button is available for this command or it can be initiated using Insert ⇒ Extrude from the menu bar. Figure 3.2 shows four different types of basic protrusions.

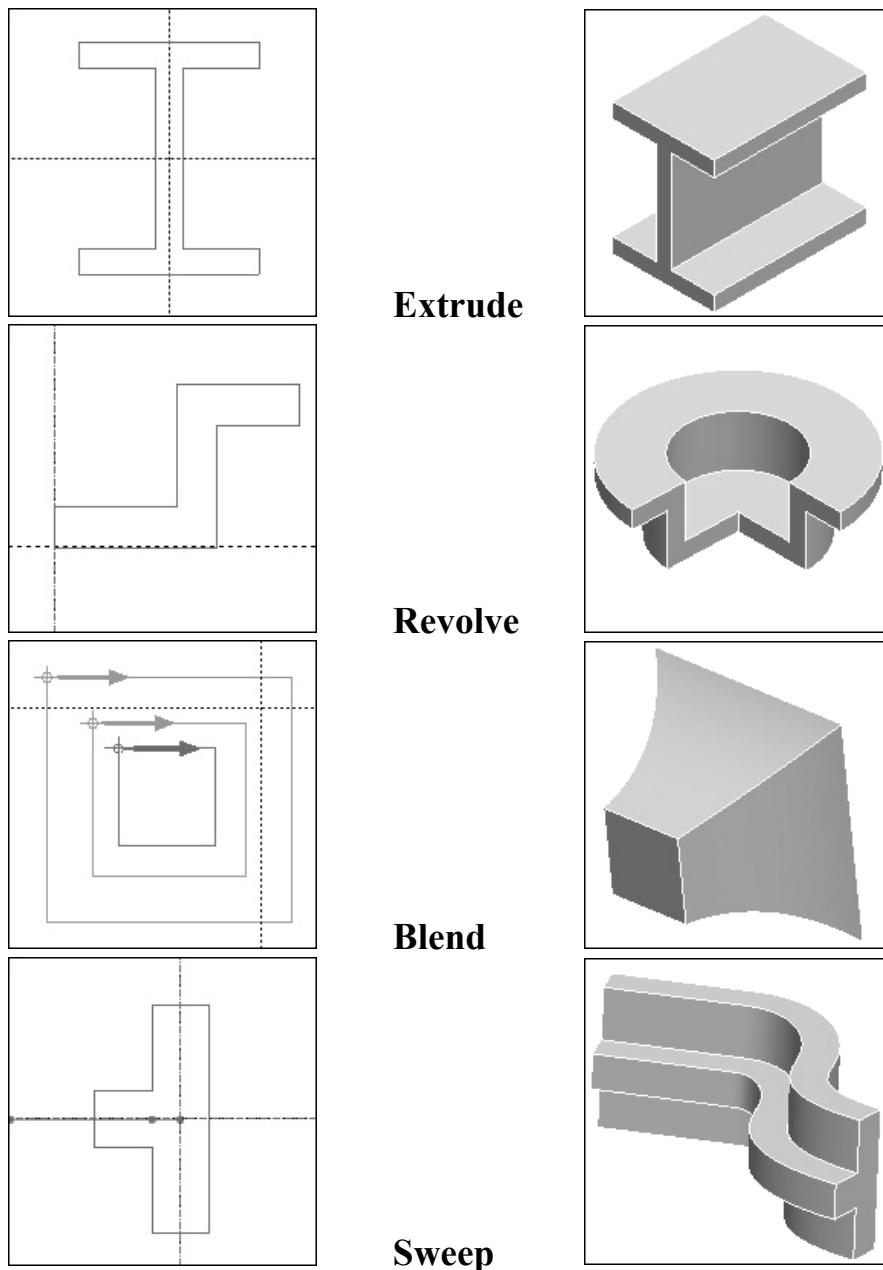


Figure 3.2 Basic Protrusions

The Design Process

It is tempting to directly start creating models. Nevertheless, in order to build value into a design, you need to create a product that can keep up with the constant design changes associated with the design-through-manufacturing process. Flexibility must be “built in” to the design. Flexibility is the key to a friendly and robust product design while maintaining design intent, and you can accomplish it through planning. To plan a design, you need understand the overall function, form, and fit of the product. This understanding includes the following points:

- Overall size of the part
- Basic part characteristics
- The way in which the part can be assembled
- Approximate number of assembly components
- The manufacturing processes required to produce the part

Lesson 3 STEPS

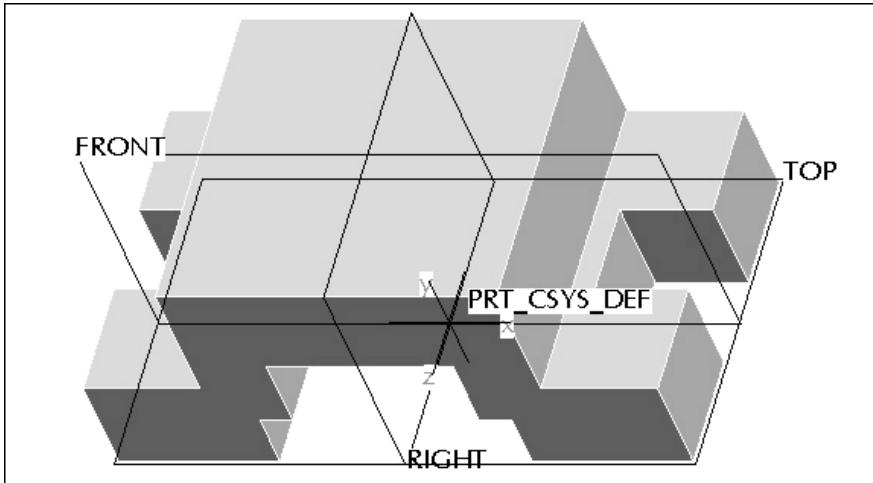


Figure 3.3 Clamp and Datum Planes

Clamp

The clamp in Figure 3.3 is composed of a protrusion and two cuts. A number of things need to be established before you actually start modeling. These include setting up the *environment*, selecting the *units*, and establishing the *material* for the part.

Before you begin any part using Pro/E, you must plan the design. The **design intent** will depend on a number of things that are out of your control and on a number that you can establish. Asking yourself a few questions will clear up the design intent you will follow: Is the part a component of an assembly? If so, what surfaces or features are used to connect one part to another? Will geometric tolerancing be used on the part and assembly? What units are being used in the design, SI or decimal inch? What is the part's material? What is the primary part feature? How should I model the part, and what features are best used for the primary protrusion (the first solid mass)? On what datum plane should I sketch to model the first protrusion? These and many other questions will be answered as you follow the systematic lesson part. However, you must answer many of the questions on your own when completing the *lesson project*, which does not come with systematic instructions.

```

Launch Pro/ENGINEER WILDFIRE 2.0 => File => Set Working Directory => select the
working directory => OK =>  Create a new object => ●Part => Name CLAMP =>
 Use default template => OK => Edit => Setup => Units => Units Manager millimeter Newton
Second (mmNs) => Set => ●Convert dimensions [Figs. 3.4(a-b)] => OK => Close => Material =>
Define => type STEEL  STEEL =>  [Fig. 3.4(c)] => File from the material
table => Save => File => Exit => Assign => pick STEEL => Accept => MMB =>  => MMB (or
Enter or OK)

```

The material file, STEEL, is without any file information [Fig. 3.4(c)]. As an option, if your instructor provides you with the specifications, or you are familiar with setting up material specs, you can edit the file using: *Edit* => *Setup* => *Material* => *Edit* => *Steel* => *Accept* => *fill in the
information* => *File* => *Save* => *File* => *Exit* => *Done*.

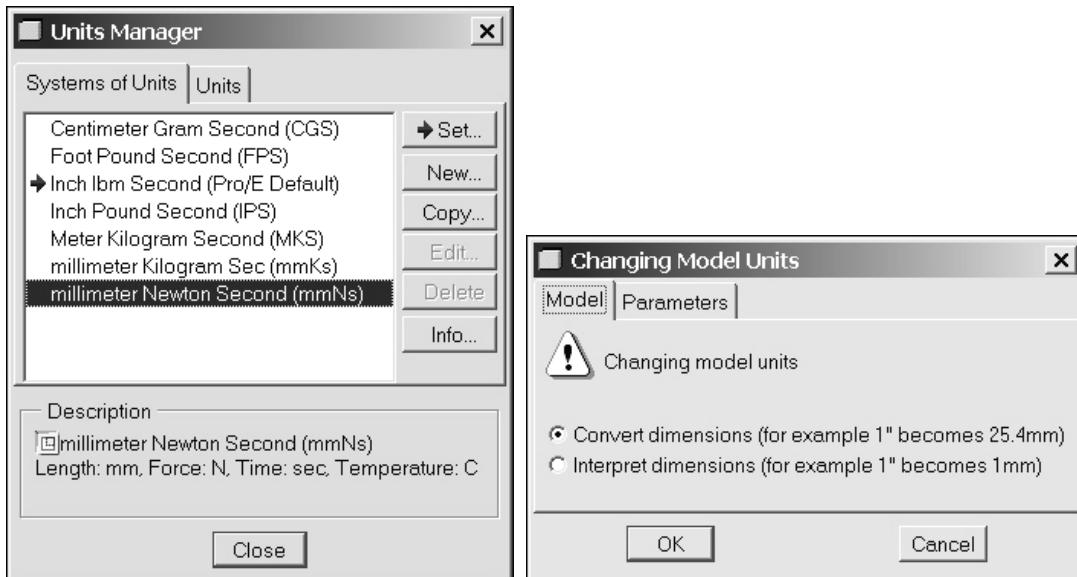


Figure 3.4(a-b) Units Manager Dialog Box and Changing Model Units Dialog Box



Figure 3.4(c) Material File

Since Use default template was selected, the default datum planes and the default coordinate system are displayed in the graphics window and in the Model Tree (Fig. 3.5). *The default datum planes and the default coordinate system will be the first features on all parts and assemblies.* The datum planes are used to sketch on and to orient the part's features. Having datum planes as the first features of a part, instead of the first protrusion, gives the designer more flexibility during the design process. Picking on items in the Model Tree will highlight that item on the model (Fig. 3.5).

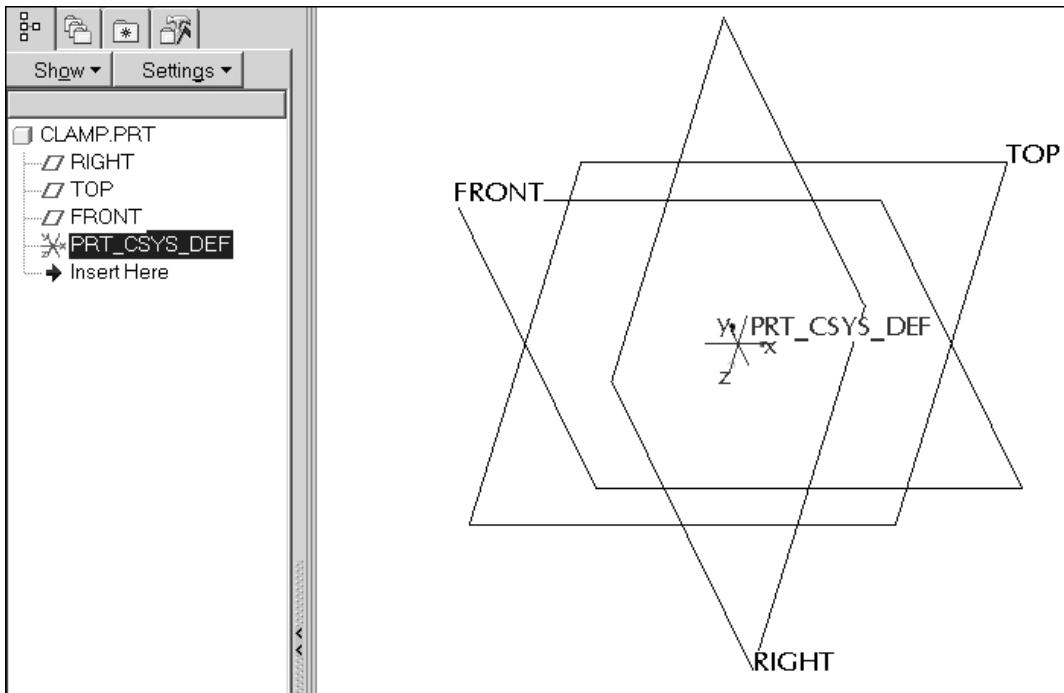


Figure 3.5 Default Datum Planes and Default Coordinate System

Pick the **FRONT** datum plane in the Model Tree \Rightarrow Sketch Tool from the right Toolbar \Rightarrow Sketch dialog box opens [Fig. 3.6(a)], click:

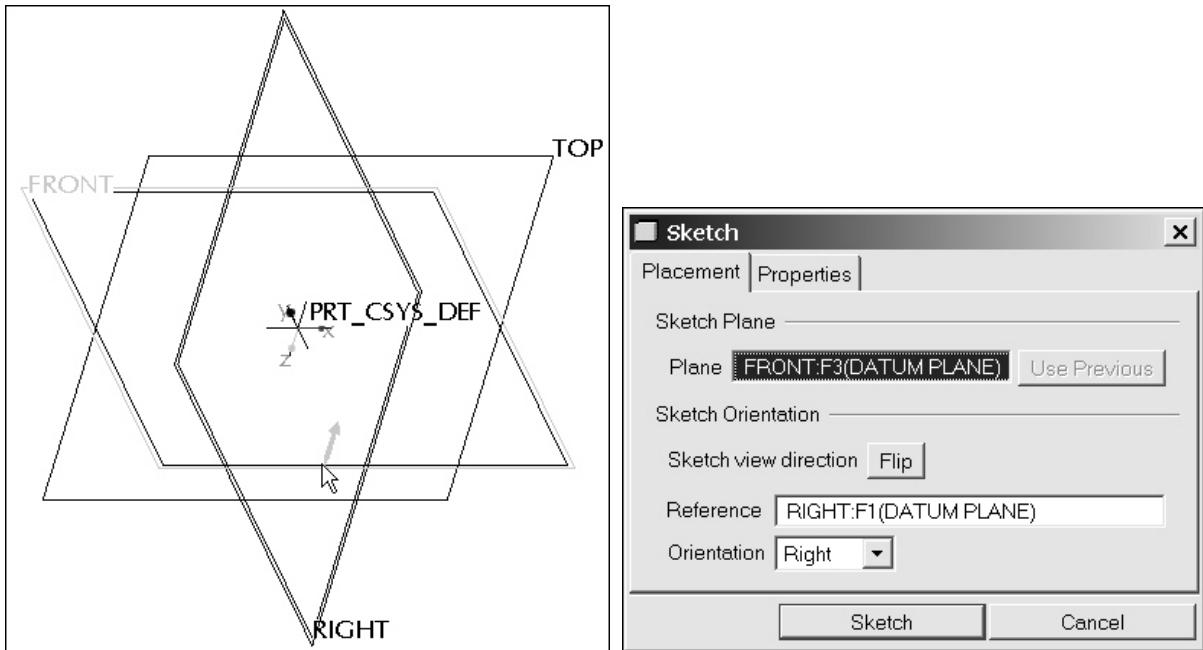


Figure 3.6(a) Sketch Dialog Box

Click: **Close** to accept the References [Fig. 3.6(b)] \Rightarrow  **Toggle the grid on** from the top toolbar [Fig. 3.6(c)]

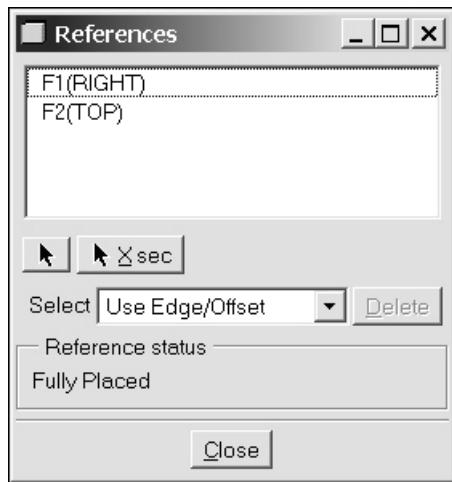


Figure 3.6(b) References Dialog Box

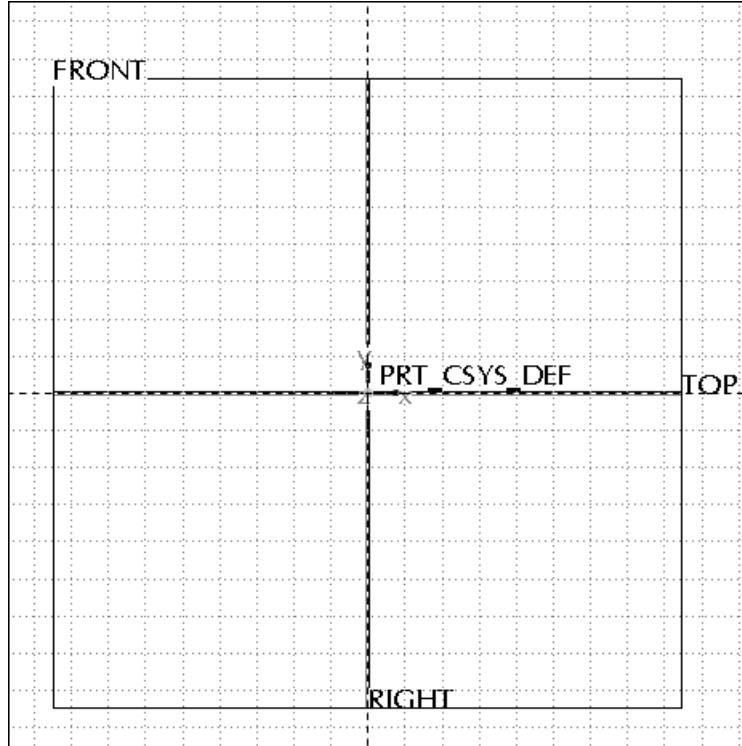


Figure 3.6(c) Grid On

The sketch is now displayed and oriented in 2D [Fig. 3.6(c)]. The coordinate system is at the middle of the sketch, where datum RIGHT and datum TOP intersect. The X coordinate arrow points to the right and the Y coordinate arrow points up. The Z arrow is pointing toward you (out from the screen). The square box you see is the limited display of datum FRONT. This is similar to sketching on a piece of graph paper. Pro/E is not coordinate-based software, so you need not enter geometry with X, Y, and Z coordinates as with many other CAD systems.

Use **Shift MMB** and **Ctrl MMB** to reposition and resize the sketch as needed. Since you now have a visible grid, it is a good idea to have your sketch picks snap to the grid. Click: **Tools** from the menu bar \Rightarrow  **Environment** \Rightarrow  **[Fig. 3.6(d)] \Rightarrow Apply \Rightarrow OK**

You can control many aspects of the environment in which Pro/E runs with the Environment dialog box. To open the Environment dialog box, click Tools \Rightarrow Environment on the menu bar or click the appropriate icon in the toolbar. When you make a change in the Environment dialog box, it takes effect for the current session only. When you start Pro/E, the environment settings are defined by your configuration file, if any; otherwise, by Pro/E configuration defaults.

Depending on which Pro/E Mode is active, some or all of the following options may be available in the **Environment** dialog box:

Display:

Dimension Tolerances Display model dimensions with tolerances

Datum Planes Display the datum planes and their names

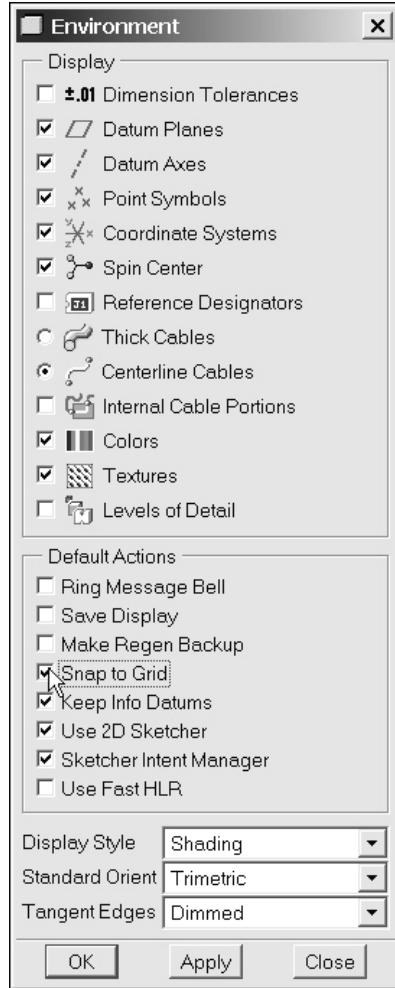
Datum Axes Display the datum axes and their names

Point Symbols Display the datum points and their names

Coordinate Systems Display the coordinate systems and their names

Spin Center Display the spin center for the model

3D Notes Display model notes



Notes as Names Display the note as a name, not the full note

Reference Designators Display reference designation of cabling, ECAD, and Piping components

Thick Cables Display a cable with 3-D thickness

Centerline Cables Display the centerline of a cable with location points

Internal Cable Portions Display cable portions that are hidden from view

Colors Display colors assigned to model surfaces

Textures Display textures on shaded models

Levels of Detail Controls levels of detail available in a shaded model during dynamic orientation

Default Actions:

Ring Message Bell Ring bell (beep) after each prompt or system message

Save Display Save objects with their most recent screen display

Make Regen Backup Backs up the current model before every regeneration

Snap to Grid Make points you select on the Sketcher screen snap to a grid

Keep Info Datums Control how Pro/E treats datum planes, datum points, datum axes, and coordinate systems created on the fly under the Info functionality

Use 2D Sketcher Control the initial model orientation in Sketcher mode

Sketcher Intent Manager Use the Intent Manager when in Sketcher

Use Fast HLR Make possible the hardware acceleration of dynamic spinning with hidden lines, datums, and axes

Display Style:

- **Wireframe** Model is displayed with no distinction between visible and hidden lines
- **Hidden Line** Hidden lines are shown in gray
- **No Hidden** Hidden lines are not shown
- **Shading** All surfaces and solids are displayed as shaded

Figure 3.6(d) Environment Dialog Box

Standard Orient:

- **Isometric** Standard isometric orientation
- **Trimetric** Standard trimetric orientation
- **User Defined** User-defined orientation

Tangent Edges:

- **Solid** Display tangent edges as solid lines
- **No Display** Blank tangent edges
- **Phantom** Display tangent edges in phantom font
- **Centerline** Display tangent edges in centerline font
- **Dimmed** Display tangent edges in the Dimmed Menu system

Because you checked Snap to Grid, you can now sketch by simply picking grid points representing the part's geometry (outline). Because this is a sketch in the true sense of the word, you need only create geometry that *approximates* the shape of the feature; the sketch does not have to be accurate as far as size or dimensions are concerned. No two sketches will be the same between those using these steps, unless you count grid spaces (which is not necessary). Even with the grid snap off Pro/E, constrains the geometry according to rules, which include but are not limited to the following:

- **RULE: Symmetry**
DESCRIPTION: Entities sketched symmetrically about a centerline are assigned equal values with respect to the centerline
- **RULE: Horizontal and vertical lines**
DESCRIPTION: Lines that are approximately horizontal or vertical are considered exactly horizontal or vertical
- **RULE: Parallel and perpendicular lines**
DESCRIPTION: Lines that are sketched approximately parallel or perpendicular are considered exactly parallel or perpendicular
- **RULE: Tangency**
DESCRIPTION: Entities sketched approximately tangent to arcs or circles are assumed to be exactly tangent

The outline of the part's primary feature is sketched using a set of connected lines. The part's dimensions and general shape are provided in Figure 3.6(e). The cut on the front and sides will be created with separate sketched features. Sketch only one series of lines (8 lines in this sketch). ***Do not sketch lines on top of lines.***

It is important not to create any unintended constraints while sketching. Therefore, remember to exaggerate the sketch geometry and not to align geometric items that have no relationship. Pro/E is very smart: If you draw two lines at the same horizontal level, Pro/E thinks they are horizontally aligned. Two lines the same length will be constrained as so.

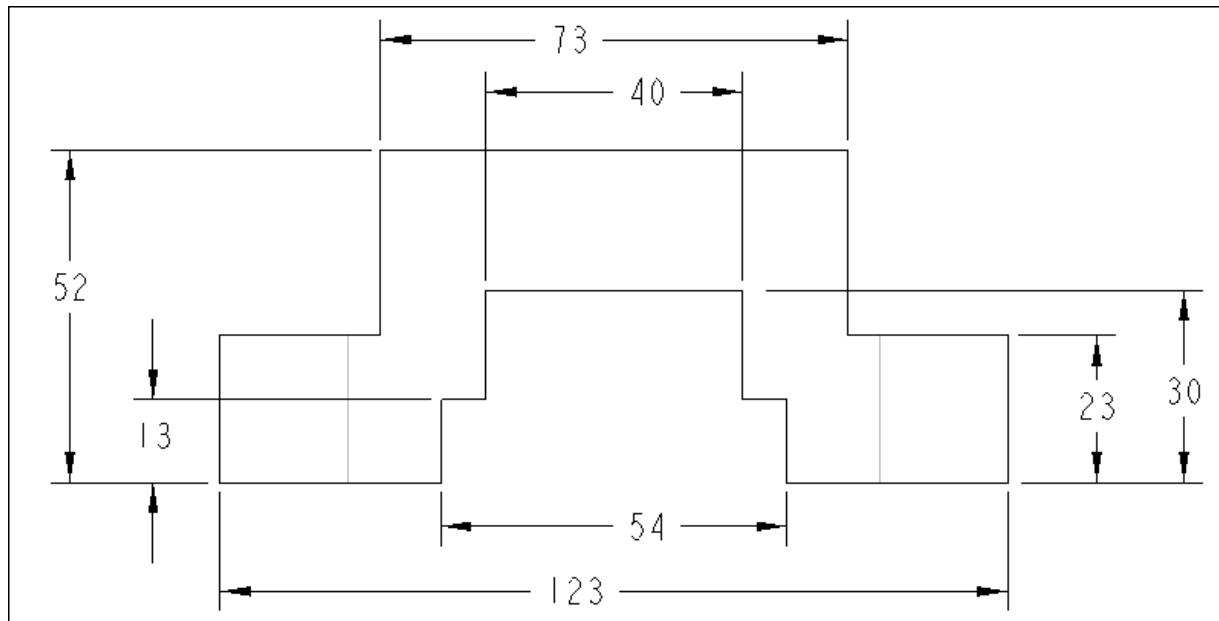


Figure 3.6(e) Front View of Drawing Showing Dimensions for the Clamp

Click: **RMB** [Fig. 3.6(f)] \Rightarrow **Centerline** [Fig. 3.6(g)] \Rightarrow pick two positions to create the vertical centerline

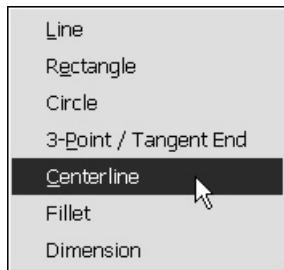


Figure 3.6(f) RMB Options

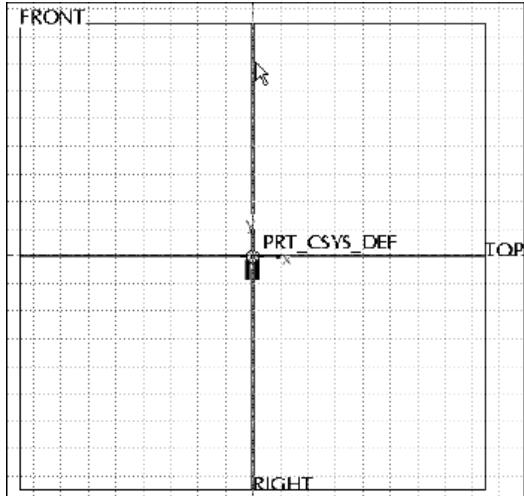


Figure 3.6(g) Create the Centerline

Click: \Rightarrow **RMB** \Rightarrow **Line** \Rightarrow sketch the eight lines of the outline [Fig. 3.6(h)] \Rightarrow **MMB** to end the line sequence [Fig. 3.6(i)] \Rightarrow **MMB**

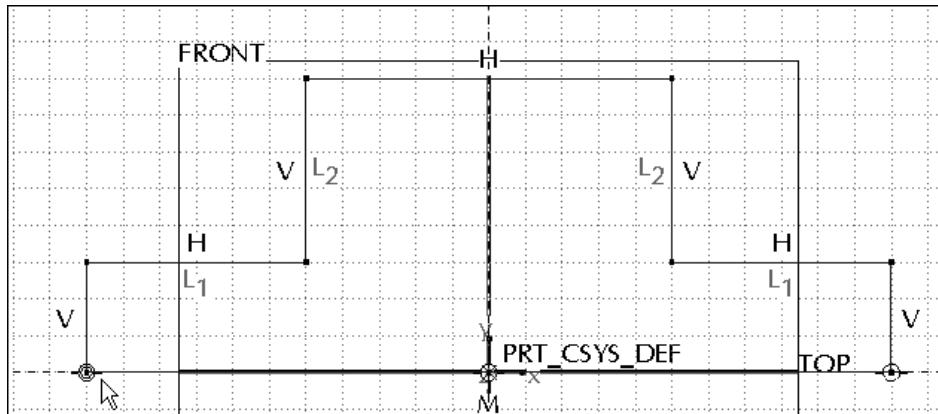


Figure 3.6(h) Sketching the Outline

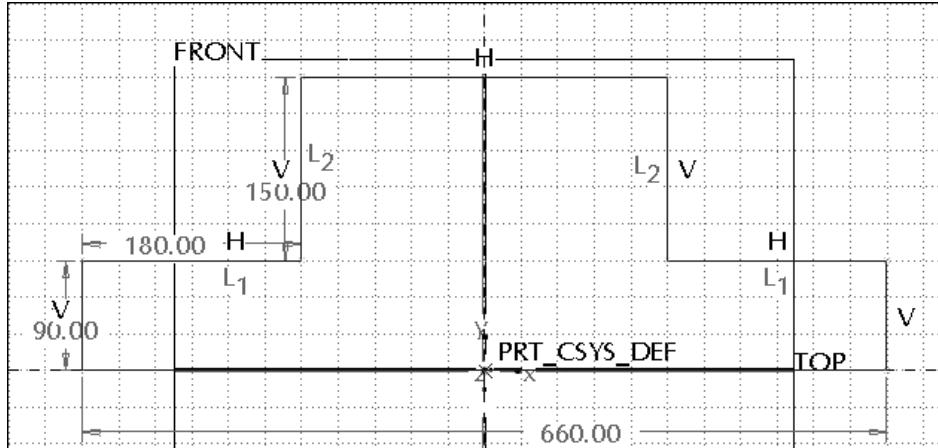


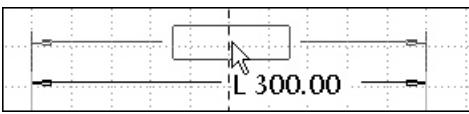
Figure 3.6(i) Default Dimensions Display

Dimensions, Constraints, Grid, and Vertices can be toggled on and off, as needed using the toolbar buttons . A sketcher constraint symbol appears next to the entity that is controlled by that constraint. Sketcher constraints can be turned on or off (enabled or disabled) while sketching. An **H** next to a line means horizontal; a **T** means tangent. Dimensions display, as they are needed according to the references selected and the constraints. Seldom are they the same as the required dimensioning scheme needed to manufacture the part. You can add, delete, and move dimensions as required. *The dimensioning scheme is important, not the dimension value, which can be modified now or later.*

Place and create the dimensions as required [Fig. 3.6(j)]. Do not be concerned with the perfect positioning of the dimensions, but try to, in general, follow the spacing and positioning standards found in the **ASME Geometric Tolerancing and Dimensioning** standards. This saves you time when you create a drawing of the part. Dimensions placed at this stage of the design process are displayed on the drawing document by simply showing all the dimensions.

To dimension between two lines, simply pick the lines with the left mouse button (LMB) and place the dimension value with the middle mouse button (MMB). To dimension a single line, pick on the line (LMB), and then place the dimension with MMB.

Click: **Tools** **Environment** **Snap to Grid** (it is easier to position the dimensions with **Snap to Grid** off) **Apply** **OK** to see a clearer sketch, your may toggle off off off (Note that the textbook leaves these items on) **RMB** **Dimension** Add and reposition dimensions as needed [Fig. 3.6(j)]



(To move a dimension – click: **pick a dimension** hold down the LMB move it to a new position release the LMB)

If any of the dimension values are *light gray* in color, they are called *weak* dimensions. If a weak dimension matches your dimensioning scheme, make them *strong* **RMB** **Strong** [Fig. 3.6(j)]

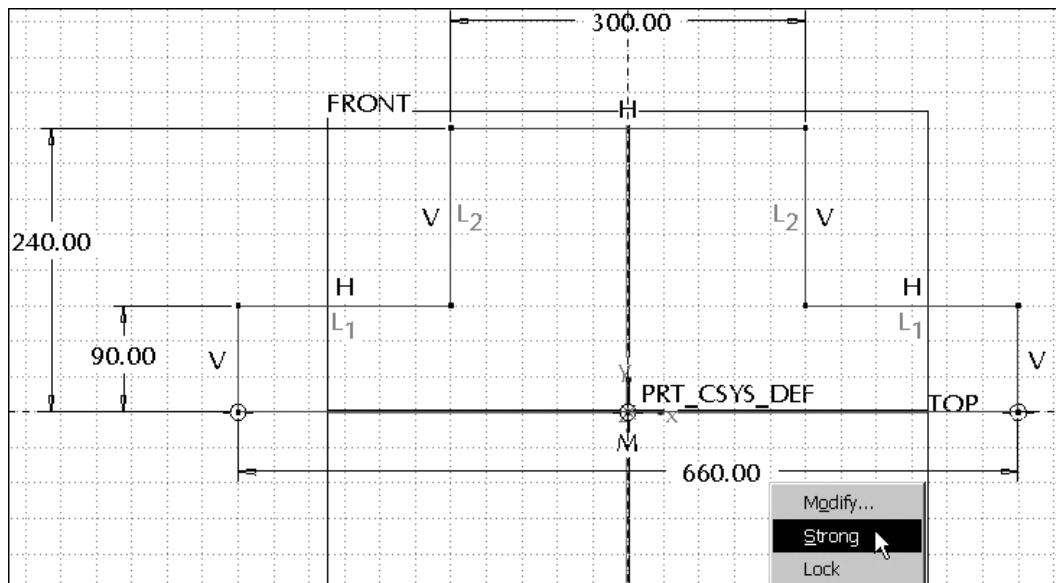


Figure 3.6(j) Dimensioned Sketch (your initial dimensions will be different)

Next, control the sketch by adding symmetry constraints, click:  **Impose sketcher constraints on the section** \Rightarrow  **Make two points or vertices symmetric about a centerline** [Fig. 3.6(k)] \Rightarrow pick the centerline and then pick two vertices to be symmetric  [Fig. 3.6(l)] \Rightarrow repeat the process and make the sketch symmetrical [Fig. 3.6(m)] \Rightarrow **Close**

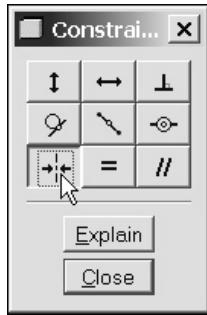


Figure 3.6(k) Constraint Dialog

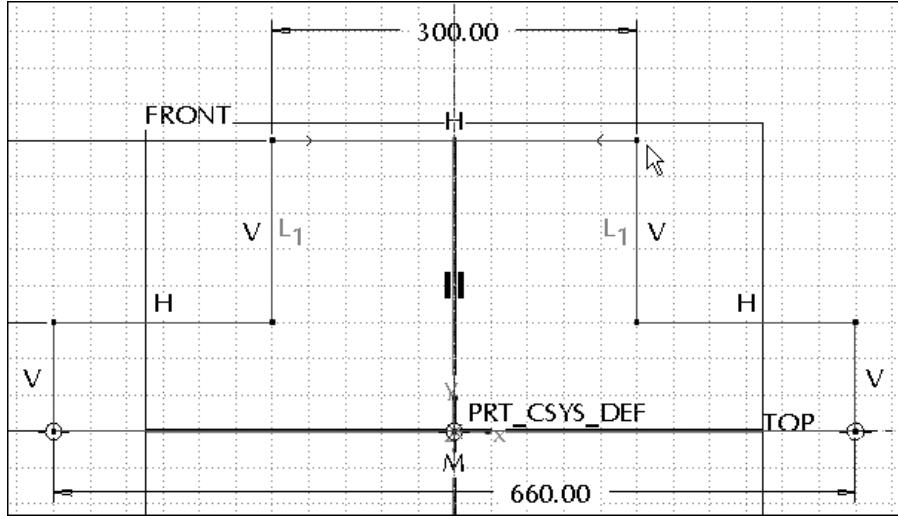


Figure 3.6(l) Adding Symmetry Constraint

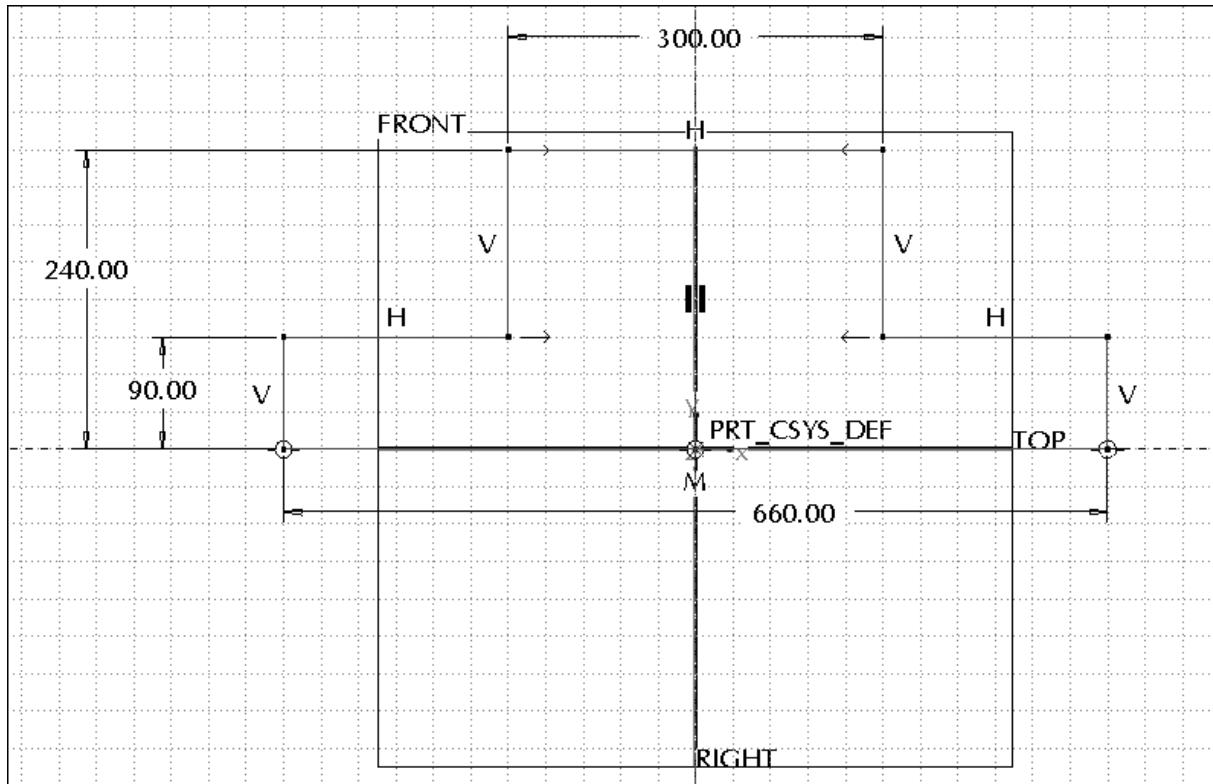


Figure 3.6(m) Sketch is Symmetrical

You can now modify the dimensions to the *design sizes*. Your original sketch values will be different from the example, but the final design values will be the same.

Click:  ⇒ Window-in the sketch (place the cursor at one corner of the window with the **LMB** depressed, drag the cursor to the opposite corner of the window and release the **LMB**) to capture all four dimensions. They will turn red. ⇒ **RMB** ⇒ **Modify** ⇒ **Lock Scale** ⇒ **Regenerate** [Fig. 3.6(n)] ⇒ double-click on length dimension (*here it is 660, but your dimension will be different*) in the Modify Dimensions dialog box and type the design value at the prompt **(123)** [Fig. 3.6(o)] ⇒ **Enter** ⇒  **Regenerate the section and close the dialog** ⇒ double-click on another dimension and modify the value [Fig. 3.6(p)] ⇒ **Enter** ⇒ continue until all of the values are changed to the design sizes [Fig. 3.6(q)]

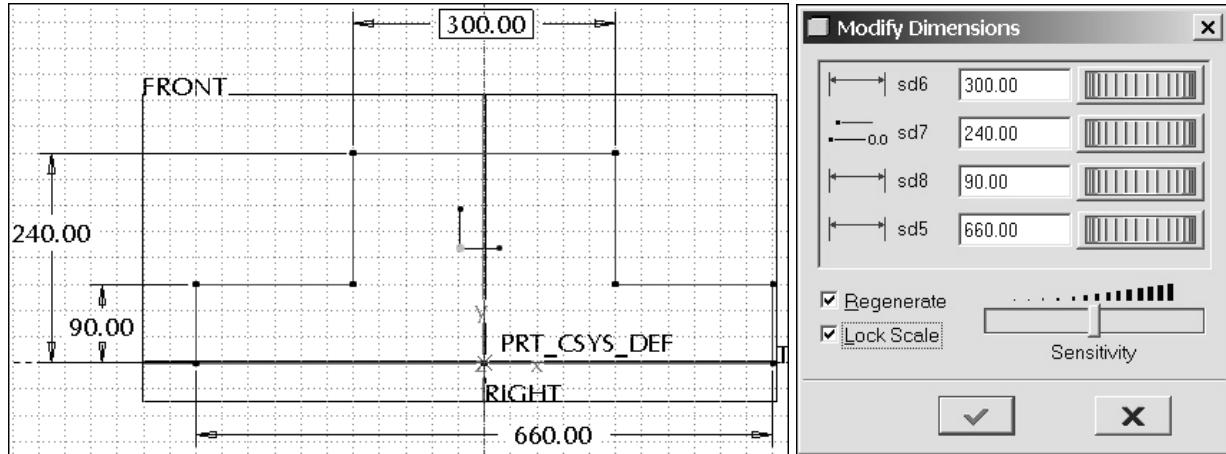


Figure 3.6(n) Modify Dimensions

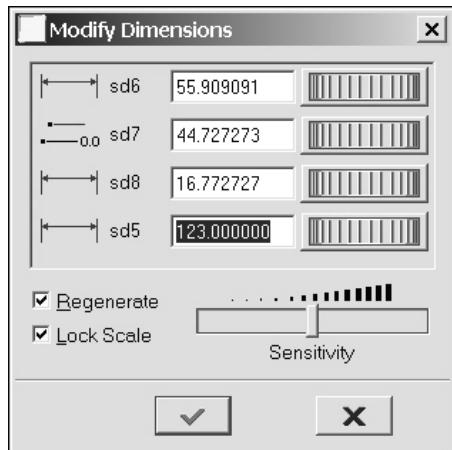


Figure 3.6(o) Modify the Dimension

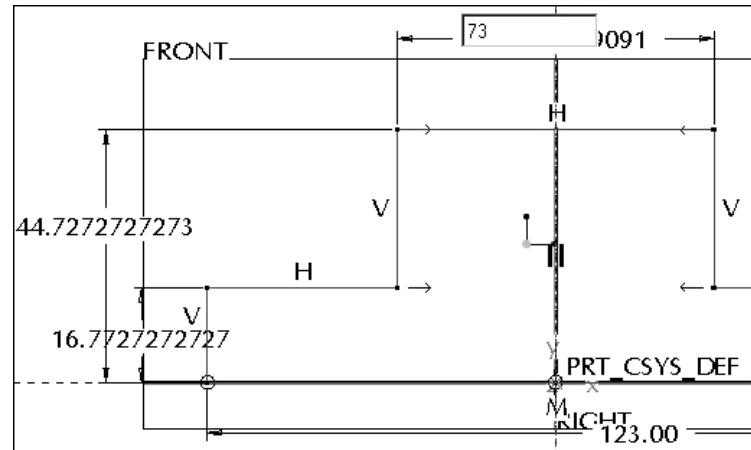


Figure 3.6(p) Modify each Dimension Individually

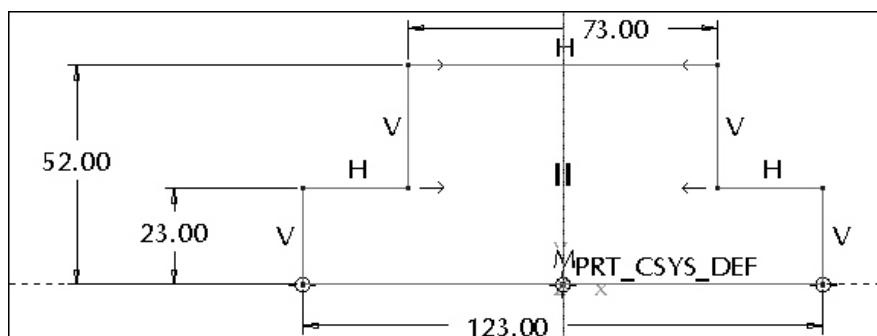


Figure 3.6(q) Modified Sketch showing the Design Values

Click: **Standard Orientation** [Fig. 3.6(r)] **on** **Continue with the current section** [Fig. 3.6(s)] **Zoom Out** as needed to see the whole object **MMB**

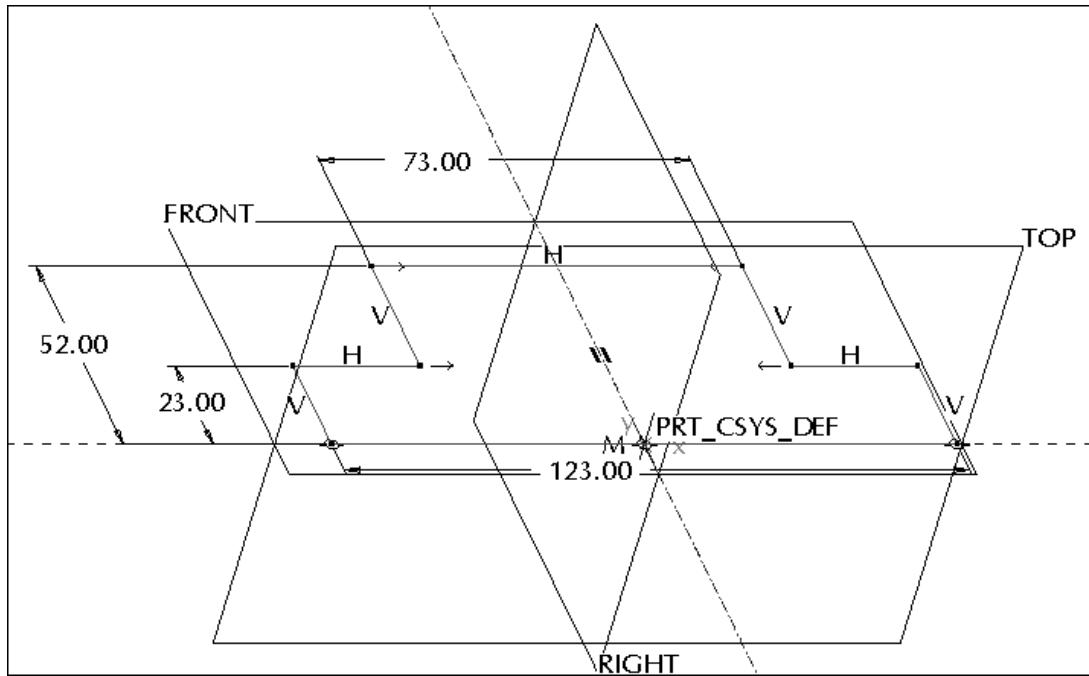


Figure 3.6(r) Regenerated Dimensions

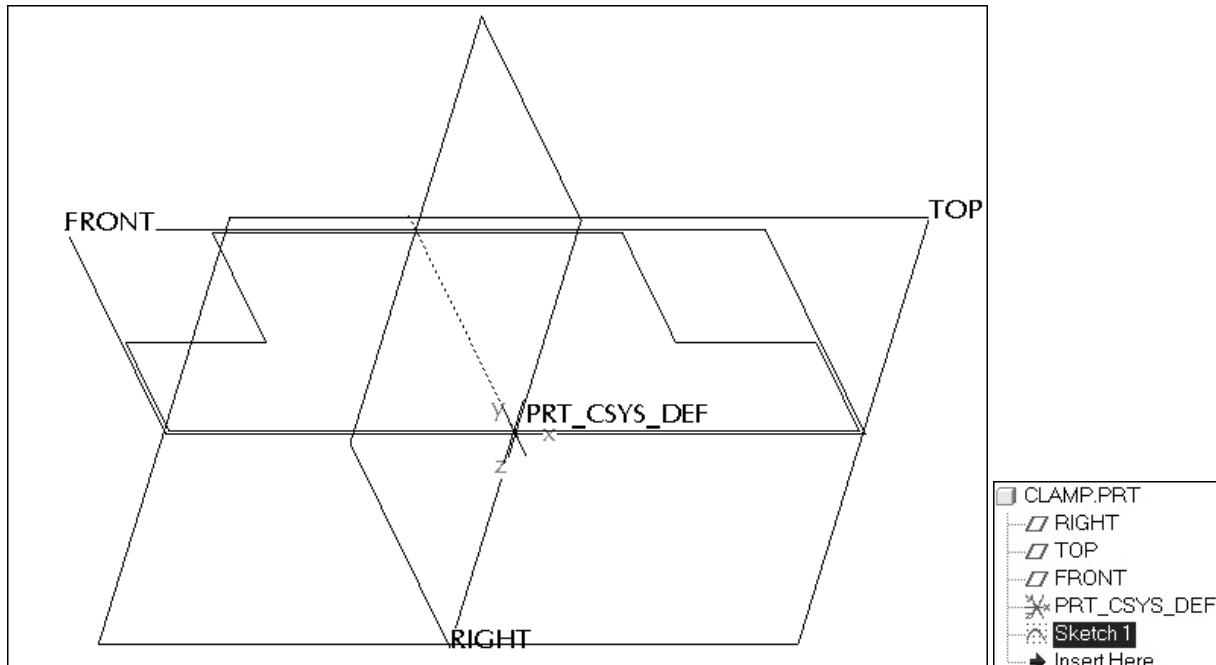


Figure 3.6(s) Completed Sketched Curve (Datum Curve)

The datum curve (Sketch1) will remain red, active and therefore selected.

Click:  **Extrude Tool** [Fig. 3.7(a)] \Rightarrow double-click on the depth value on the model \Rightarrow type **70** [Fig. 3.7(b)] \Rightarrow **Enter** \Rightarrow place your pointer over the square drag handle  (it will turn black) \Rightarrow **RMB** \Rightarrow **Symmetric** [Fig. 3.7(c)] \Rightarrow **MMB** [Fig. 3.7(d)] \Rightarrow  \Rightarrow **MMB**

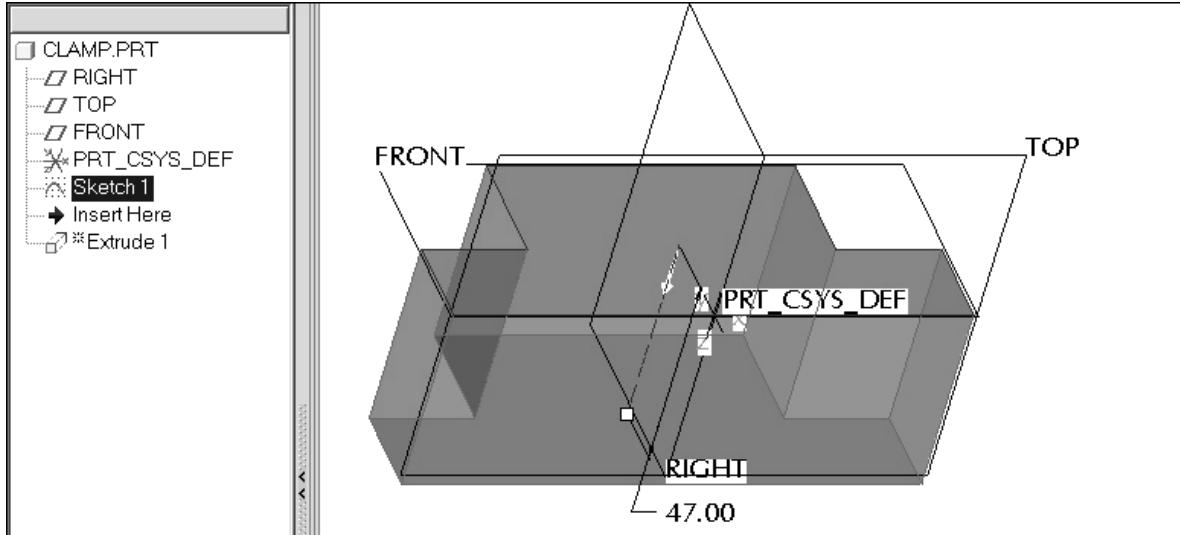


Figure 3.7(a) Depth of Extrusion Previewed

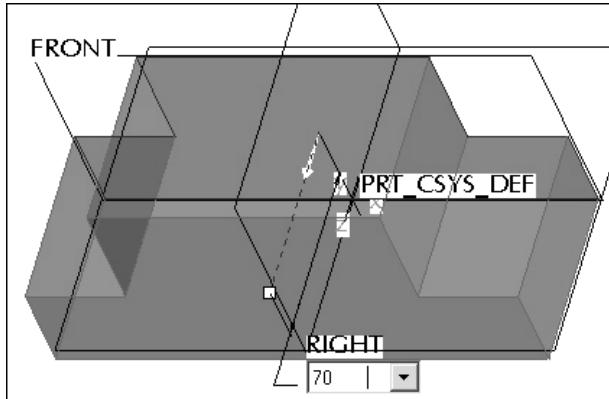


Figure 3.7(b) Modify the Depth Value

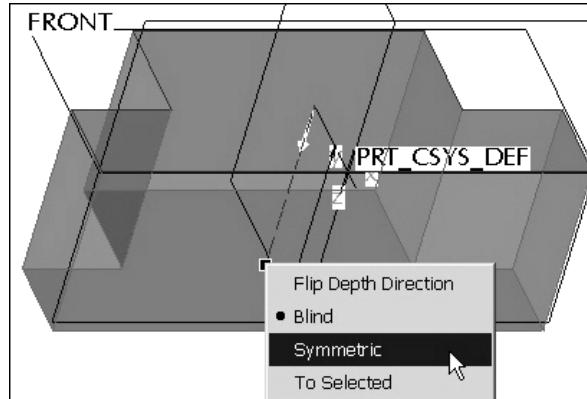


Figure 3.7(c) Symmetric

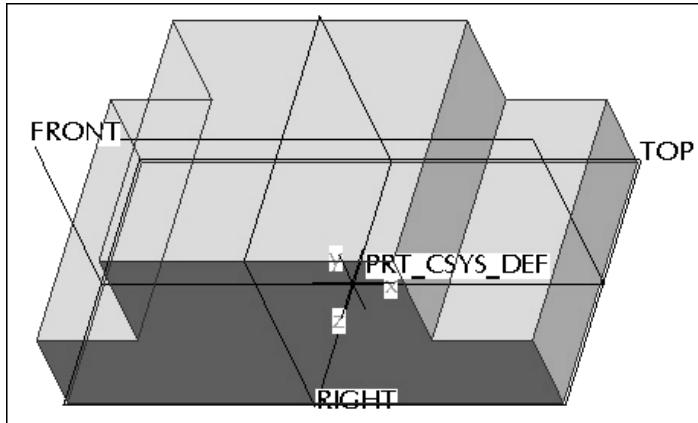
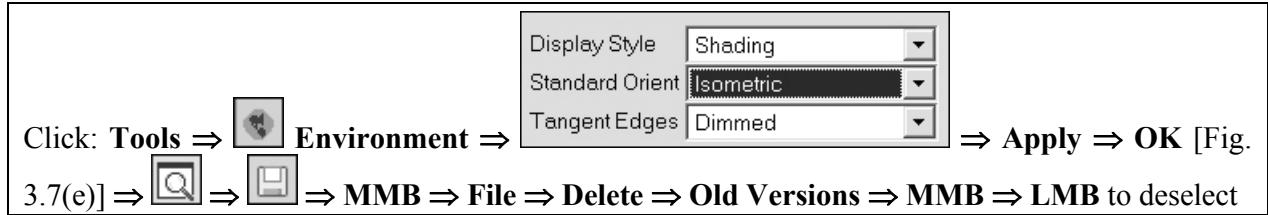


Figure 3.7(d) Completed Extrusion



Storing an object on the disk does not overwrite an existing object file. To preserve earlier versions, Pro/E saves the object to a new file with the same object name but with an updated version number. Every time you store an object using Save, you create a new version of the object in memory, and write the previous version to disk. Pro/E numbers each version of an object storage file consecutively (for example, box.sec.1, box.sec.2, box.sec.3). If you save 25 times, you have 25 versions of the object, all at different stages of completion. You can use *File* ⇒ *Delete* ⇒ *Old Versions* after the *Save* command to eliminate previous versions of the object that may have been stored.

When opening an existing object file, you can open any version that is saved. Although Pro/E automatically retrieves the latest saved version of an object, you can retrieve any previous version by entering the full file name with extension and version number (for example, **partname.prt.5**). If you do not know the specific version number, you can enter a number relative to the latest version. For example, to retrieve a part from two versions ago, enter **partname.prt.3** (or **partname.prt.-2**).

You use *File* ⇒ *Erase* to remove the object and its associated objects from memory. If you close a window before erasing it, the object is still in memory. In this case, you use *File* ⇒ *Erase* ⇒ *Not Displayed* to remove the object and its associated objects from memory. This does not delete the object. It just removes it from active memory. *File* ⇒ *Delete* ⇒ *All Versions* removes the file from memory and from disk completely. You are prompted with a Delete All Confirm dialog box when choosing this command. Be careful not to delete needed files.

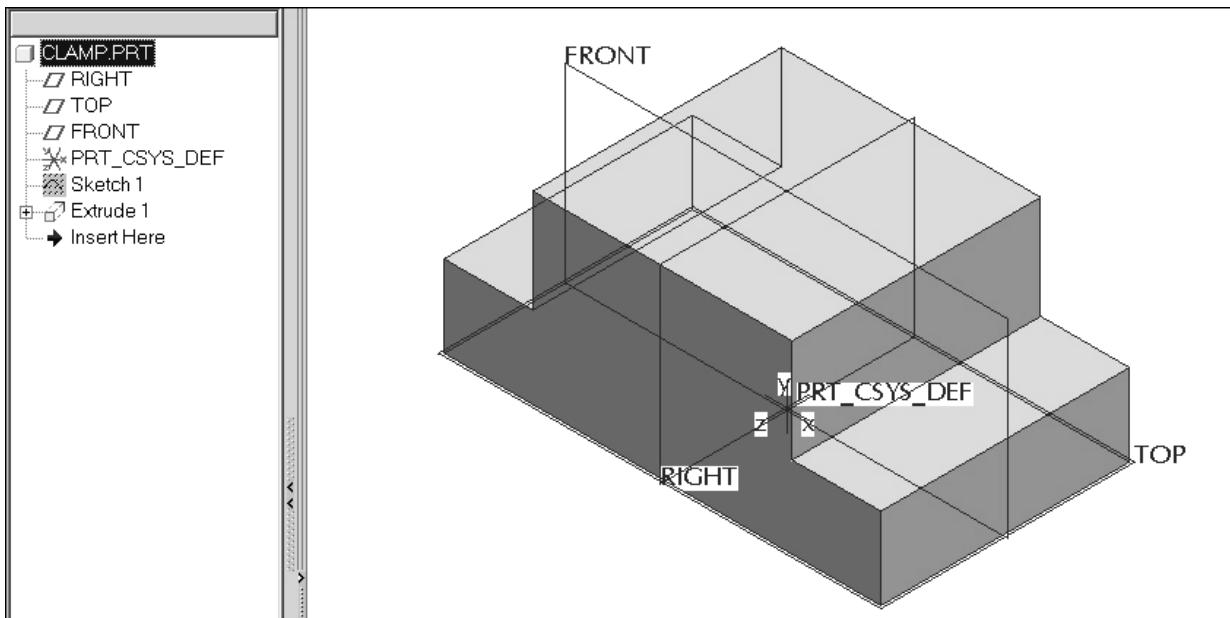


Figure 3.7(e) Isometric Orientation

Next, the cut through the middle of the part will be modeled.

Click: Extrude Tool \Rightarrow RMB \Rightarrow Remove Material \Rightarrow RMB \Rightarrow Define Internal Sketch [Fig. 3.8(a)] \Rightarrow Use Previous from the Sketch dialog box [Fig. 3.8(b)] \Rightarrow Sketch \Rightarrow RMB \Rightarrow Centerline [Fig. 3.8(c)] \Rightarrow create a vertical centerline \Rightarrow RMB \Rightarrow Line \Rightarrow sketch the seven lines of the open outline [Fig. 3.8(d)] \Rightarrow MMB \Rightarrow **Impose sketcher constraints on the section** \Rightarrow [Fig. 3.6(k)] \Rightarrow pick the centerline and then pick two vertices to be symmetric \Rightarrow repeat the process and make the sketch symmetrical \Rightarrow If you attempt to create too many constraints, Pro/E will open the Resolve Sketch dialog box [Fig. 3.8(e)]. **Delete** the extra symmetric constraint if this happens. \Rightarrow Close

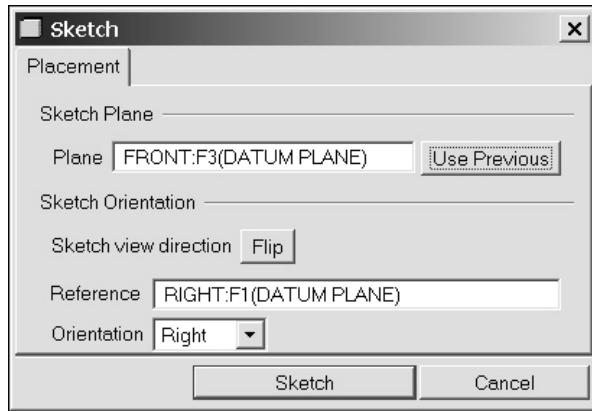
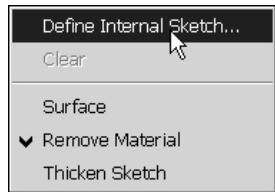


Figure 3.8(a) RMB Options

Figure 3.8(b) Sketch Dialog Box

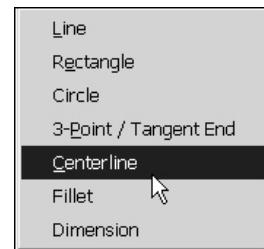


Figure 3.8(c) Centerline

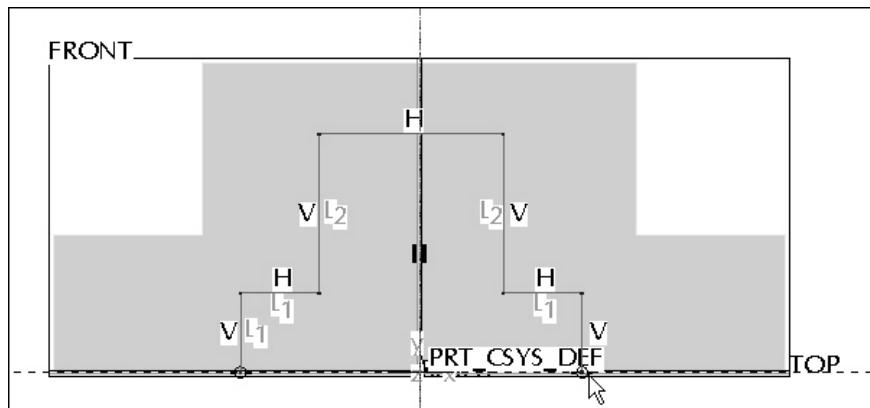


Figure 3.8(d) Sketch

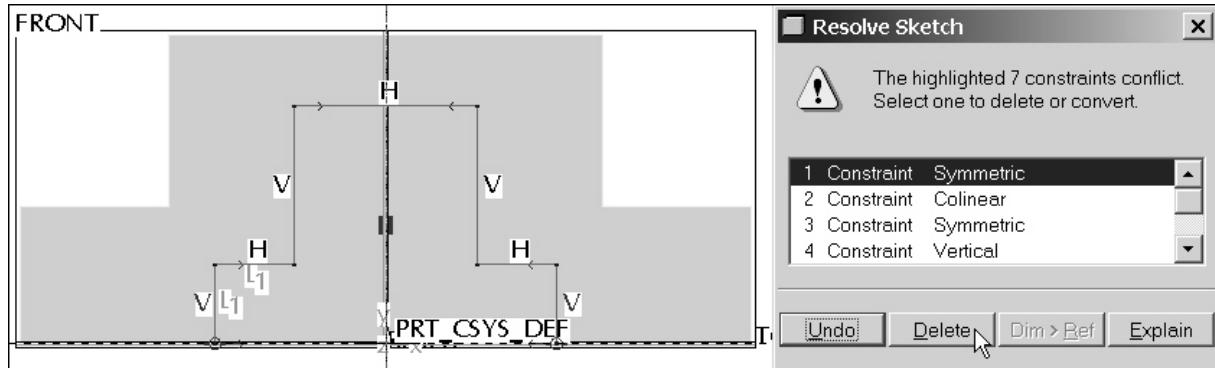


Figure 3.8(e) Resolve Sketch Dialog Box

Click: **RMB** \Rightarrow **Dimension** \Rightarrow add and reposition dimensions, your values will be different \Rightarrow Hidden Line [Fig. 3.8(f)] \Rightarrow **MMB** to deselect dimension tool and activate  **Select items**

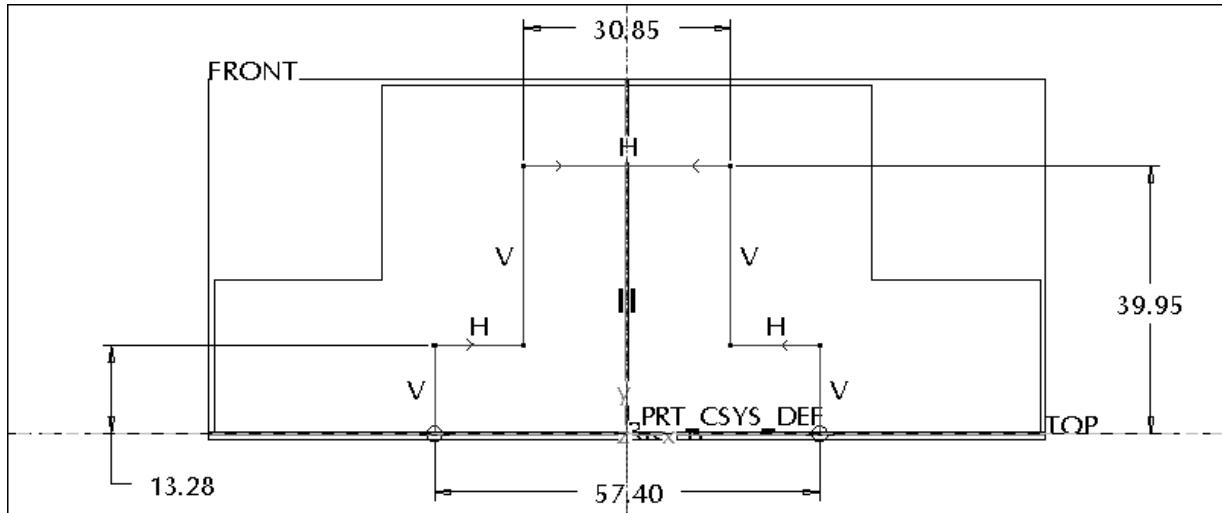


Figure 3.8(f) Dimensioned Sketch

Window-in the sketch to capture all four dimensions. They will turn red. \Rightarrow **RMB** \Rightarrow **Modify** \Rightarrow
 Regenerate
 Lock Scale \Rightarrow modify the values [Fig. 3.8(g)] \Rightarrow  [Fig. 3.8(h)]

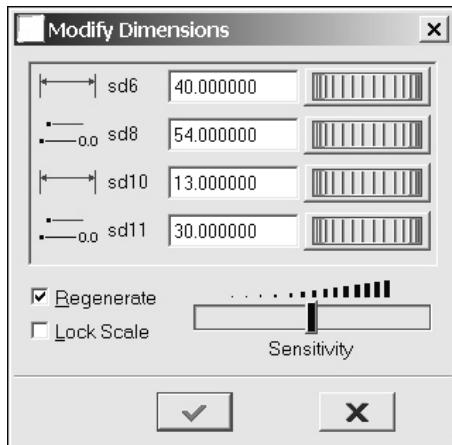


Figure 3.8(g) Modified Dimensions Dialog Box

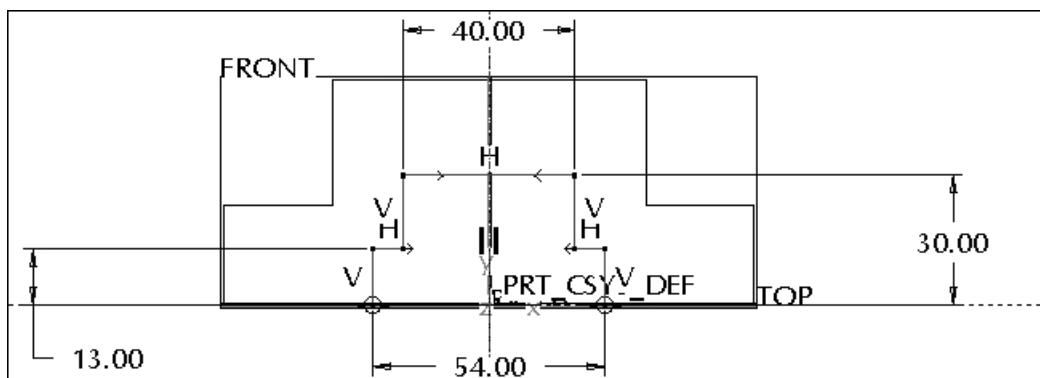


Figure 3.8(h) Modified Sketch Dimensions

Click: Shading \Rightarrow Standard Orientation \Rightarrow note the yellow direction arrow
 \Rightarrow Zoom Out \Rightarrow Options from the Dashboard \Rightarrow Side 1 \Rightarrow Through All [Fig. 3.8(i)] \Rightarrow
Side 2 \Rightarrow Through All [Fig. 3.8(j)] \Rightarrow MMB [Fig. 3.8(k)] \Rightarrow \Rightarrow \Rightarrow MMB \Rightarrow LMB

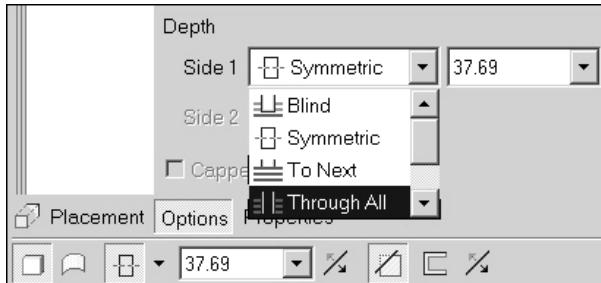


Figure 3.8(i) Options Side 1

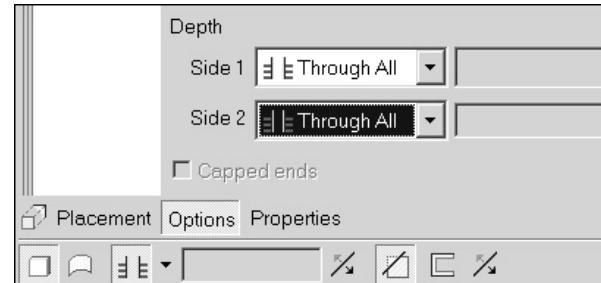


Figure 3.8(j) Options Side 2

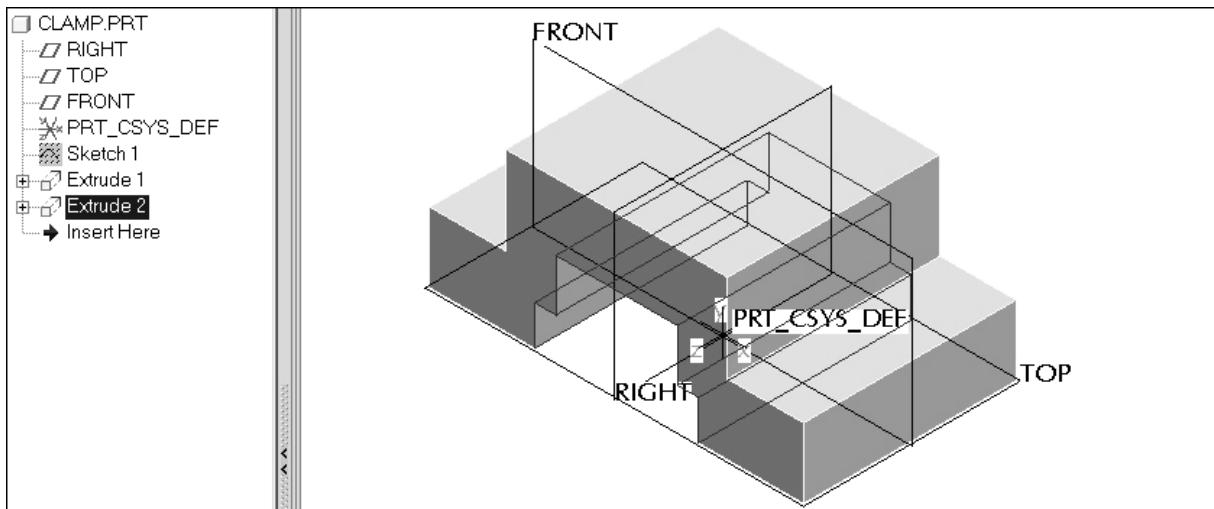


Figure 3.8(k) Completed Cut

The next feature will be a **20 × 20** centered cut (Fig. 3.9). Because the cut feature is identical on both sides of the part, you can mirror and copy the cut after it has been created.

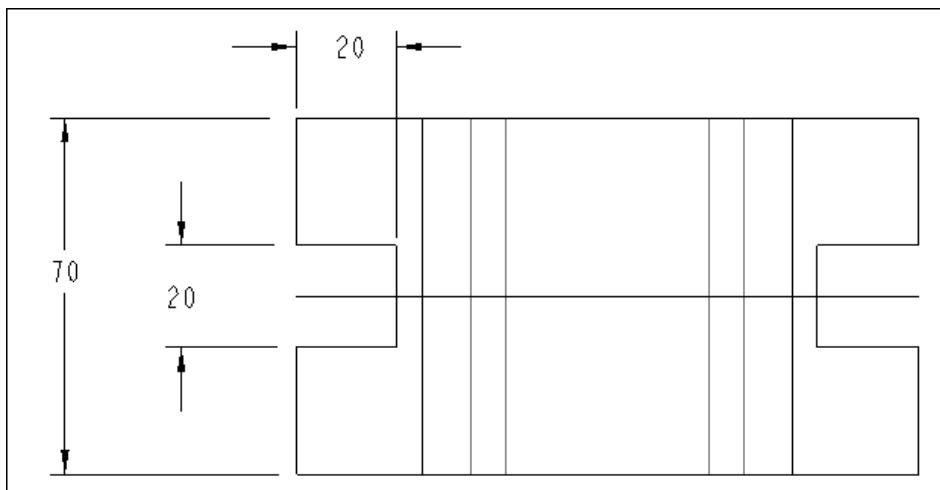


Figure 3.9 Top View of Drawing

Create the cut, click: **Extrude Tool** from the Dashboard Sketch dialog box opens Sketch Plane--- Plane: select **TOP** datum from the model as the sketch plane [Fig. 3.10(a)] pick the left edge of the part to add it to the References dialog box [Fig. 3.10(b)] click **Close** to accept the References **OK**

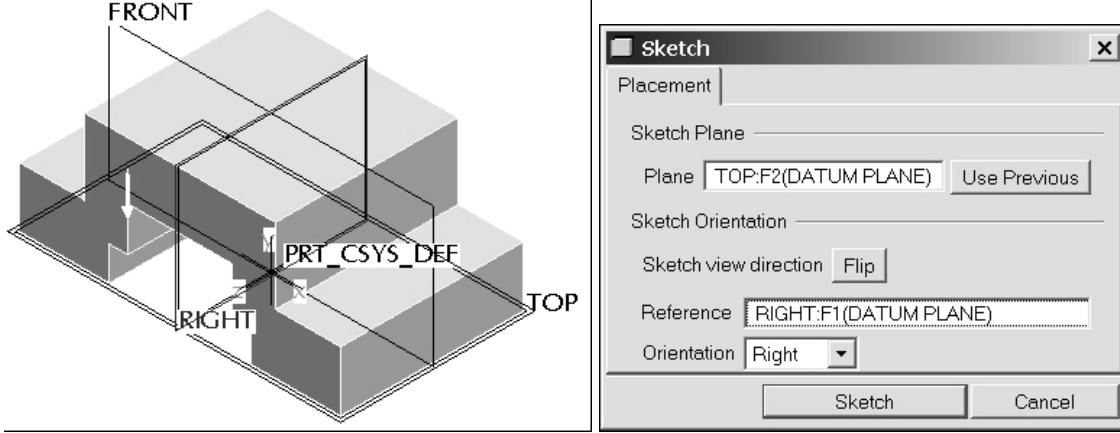


Figure 3.10(a) Sketch Plane Selection and Sketch Dialog Box

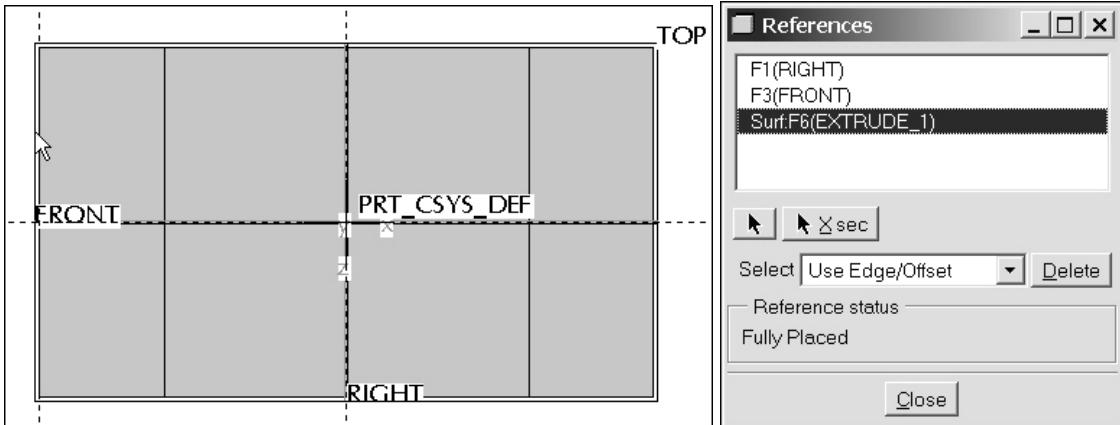


Figure 3.10(b) Add the left edge/surface of the part to the References Dialog Box

Click: Hidden Line **RMB** Centerline create a horizontal centerline through the center of the part **RMB** Line place the mouse on the left edge and create three lines [Fig. 3.10(c)] MMB to end the line sequence [Fig. 3.10(d)] **Impose sketcher constraints on the section** pick the centerline and then pick two vertices to be symmetric add the required dimensions move and modify the values for the two dimensions (20 X 20) [Fig. 3.10(e)] Standard Orientation from the dashboard Remove Material Options tab Through All (Extrude in first direction to interest with all surfaces) [Fig. 3.10(f)] Shading rotate your model using MMB to see the cut clearly [Fig. 3.10(g)] MMB

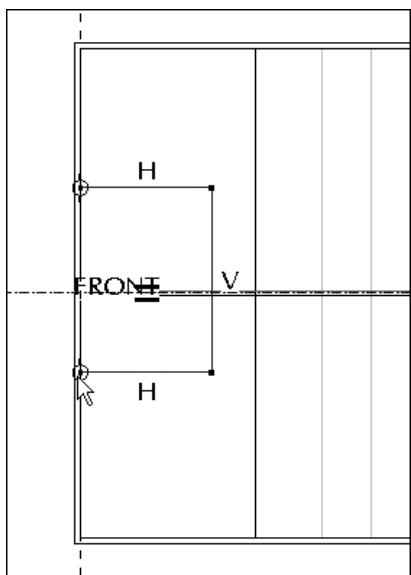


Figure 3.10(c) Three Line Sketch

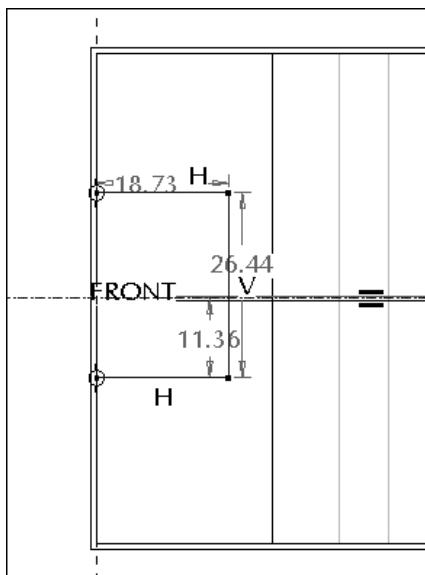


Figure 3.10(d) Default Dimensions

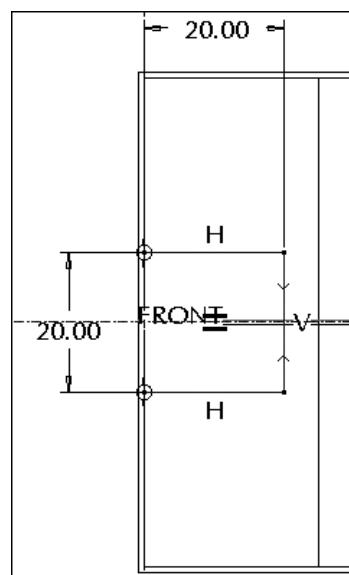


Figure 3.10(e) Modified Values

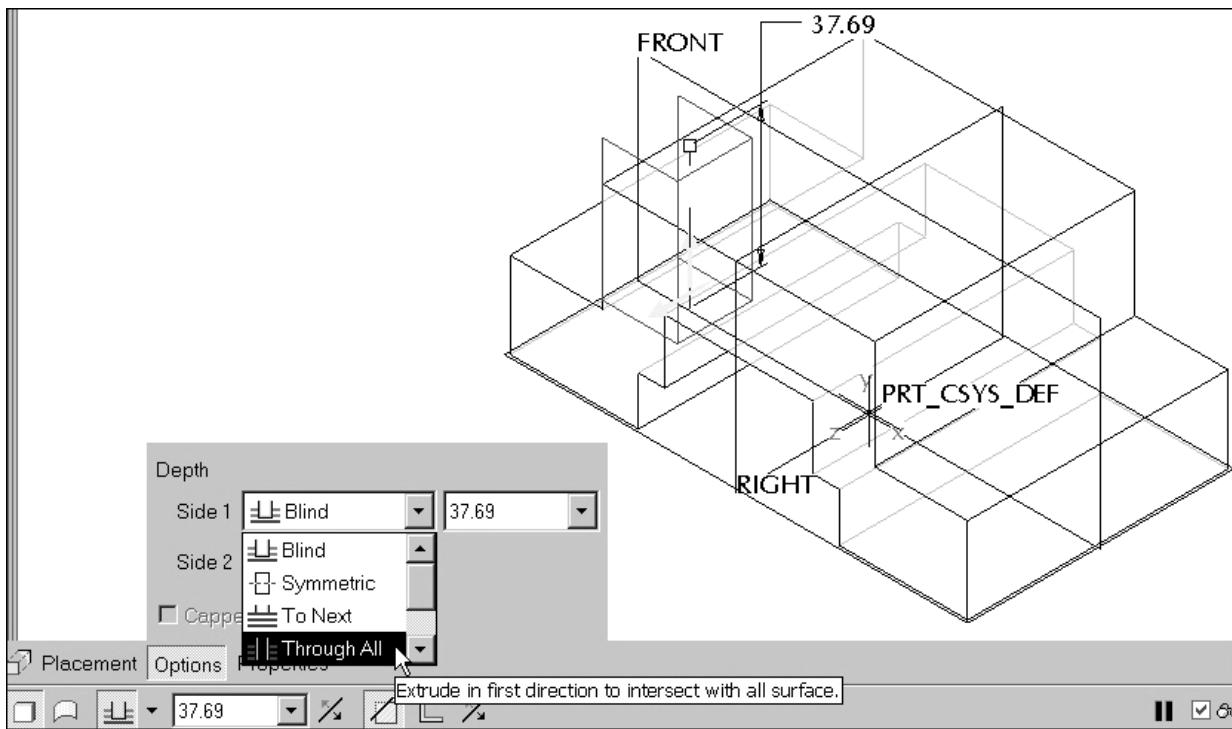


Figure 3.10(f) Options Through All

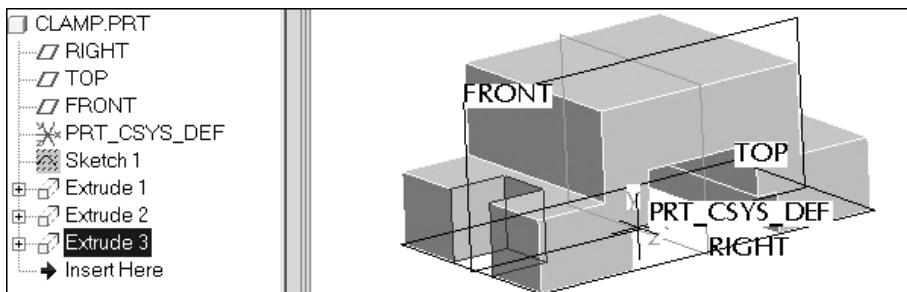


Figure 3.10(g) Completed Second Cut

Click: **Standard Orientation** \Rightarrow with the new cut still highlighted [Fig. 3.11(a)], click: **Mirror Tool** \Rightarrow select the **RIGHT** datum plane from the Model Tree [Fig. 3.11(b)] \Rightarrow **MMB** [Fig. 3.11(c)] \Rightarrow **File** \Rightarrow **Save** \Rightarrow **MMB** \Rightarrow **MMB** rotate part [Fig. 3.11(d)] \Rightarrow **File** \Rightarrow **Close Window**

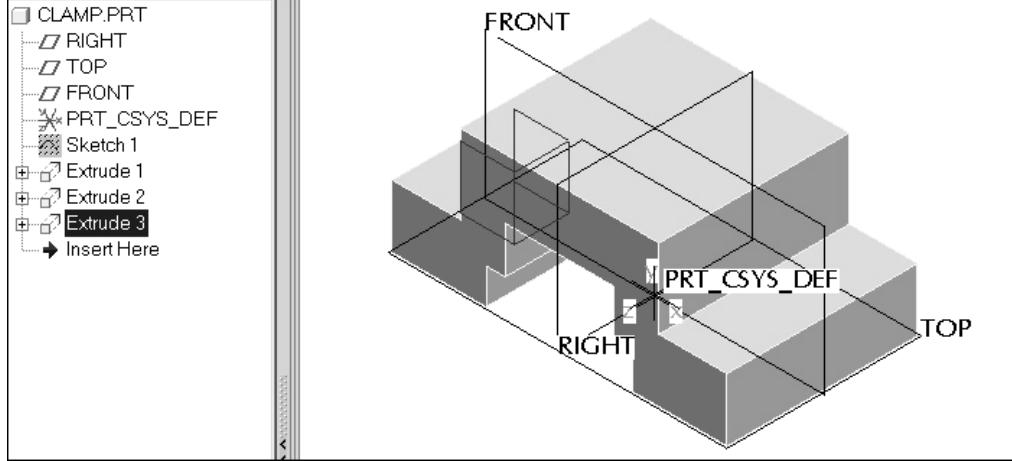


Figure 3.11(a) Extruded Cut is Highlighted (Selected)

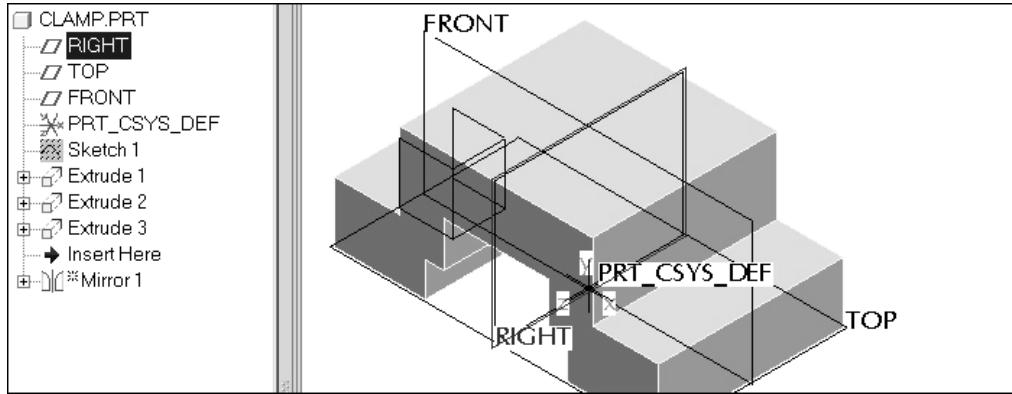


Figure 3.11(b) Select the **RIGHT** Datum Plane as the Mirroring Plane

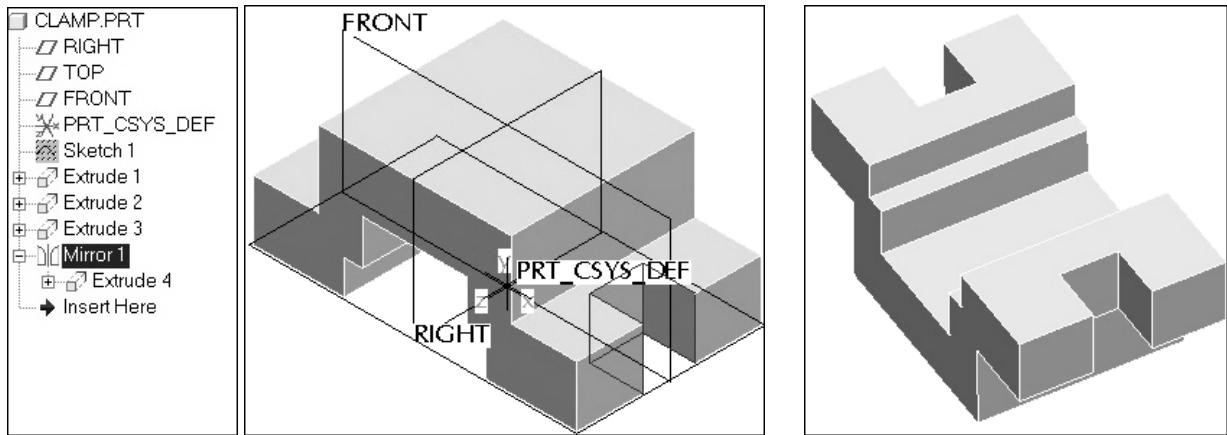


Figure 3.11(c) Mirror Cut

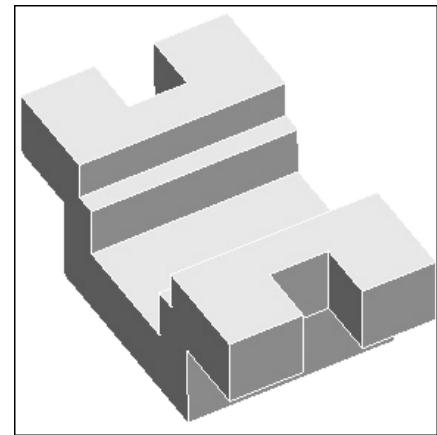


Figure 3.11(d) Completed Part

Lesson 3 is now complete. If you wish to model a project without instructions, a complete set of projects and illustrations are available at www.cad-resources.com \Rightarrow **Downloads**.

NOTES: