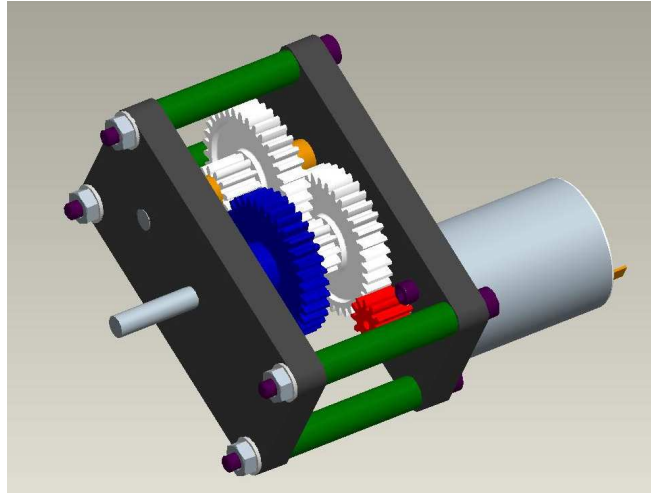# Part II

Parts - Worked Examples

4 Startup

Figure 4: Complete Gearbox

This section is a guided tutorial to produce models of various parts of the gearbox shown in figure 4 and then assemble them. The tutorial is laid out in subsections corresponding to the construction of individual features within the model.

Detailed descriptions are given of the construction of the initial features, as the guide progresses information is limited to only the new commands being employed.

As in the physical production of an item there is often a more than one way of construction, with no one 'correct' method. The construction techniques used here have been chosen to show a range of features and their of method of usage rather than a definitive solution.

4.1 Start ProEngineer

If not already running initiate the system as described in the getting started section, e.g. by selecting START→**3D Modelling** →**Pro/E Wildfire2**.

N.B. By default the CD version starts by Pro/Engineer in a directory on the ram disc containing some predefined parts associated with this tutorial. Any files saved in this directory will be LOST when the system is shuts down. If you wish to retain any file please use the 'X File Explorer' to sae them to a permentant storage location, e.g. floppy, remote account, usb key or hard disc

To change to the directory used to save files select FILE → **Set Working Directory** (or use the hotkey option <u>*cd*</u>) and then select the required directory.

4.2 Key

To aid in the use of this handout a number of conventions/fonts/abbreviations have been used to indicate the difference between keyboard entry, menu titles, menu items etc. These are listed below:

Font Used	Item	Abbreviation	Action
Bold/Times:	**menu item**	LMB	Left Mouse Button
CAPITALS:	WINDOW/MENU TITLE	MMB	Middle Mouse Button
Italic:	_Keyboard inputs_	RMB	Right Mouse Button
Bold/Helvetica:	**Hint**		

Menu items followed by a → indicate that the user should follow the input to a cascaded or flyout menu. N.B. In many cases the choices required are the default and therefore do not need to be individually selected.

Central to the use of the package is the ability to obtain the best view of the object you are constructing, e.g. the orientation and the display mode. Access to commands affecting the display are concentrated under the View pull down menu and the top icon bar. In addition the orientation can be manipulated with a combination of the Ctrl/Shift keys and the mouse.
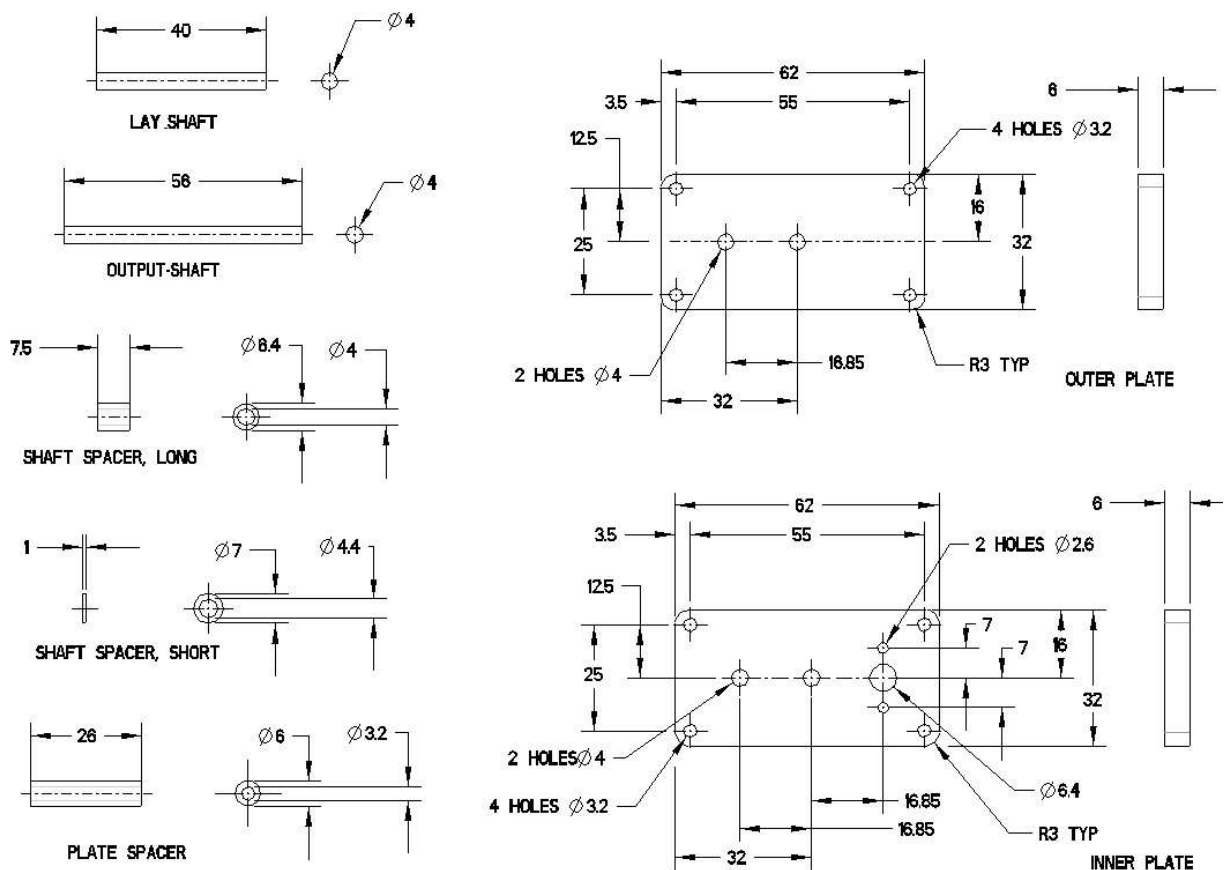

Figure 5: Details of the components that are constructed in this tutorial

5   'Extruded' Parts

5.1   New Part - shaft_spacer_long

The creation of a new part involves the generation of a file together with default coordinate system, datum planes and material properties etc. While this can be achieved using individual commands (see Appendix A), it is highly recommmended that the MDP default 'start part' is used.

Use the LMB to select NEW PART icon (New Part) on the top menu bar, this will start an automatic procedure to generate a new part using the default MDP settings.

After a few seconds a menu will appear prompting for the name of a part, e.g. *shaft_spacer_long*, and then press ENTER or click (LMB) on OK.

This will initiate a procedure to set up the default coordinate system and datum planes (Front, Mid, Top) which will be appear in the main window.

N.B. The brown Datum icons ( ) on the top icon bar can be used to toggle the display of datum features

In addition to the default datums a number of other part paramenters have been set by using the CUED/MDP start-part. If you need to change any paramenter/ use the additional functionality please refer to the online help or the manuals.
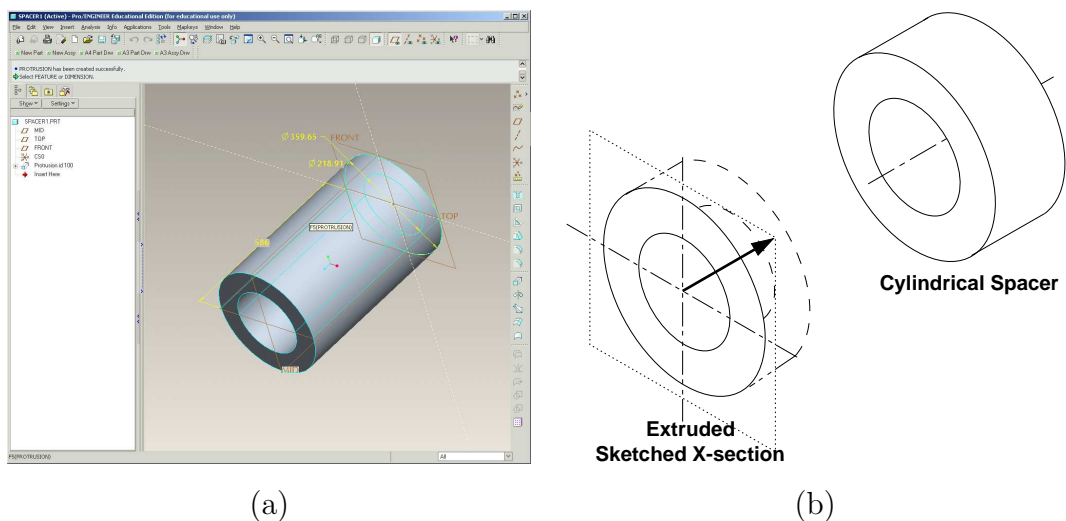

| (a) | (b) |

Figure 6: (a) Example Cylindrical Spacer (b) 'Extrude Tool' used to generate Spacer

The "Extrude" function, often used to generate an initial part (c.f. the concept of stock material in a workshop), is used to produce the basis of a cylindrical spacer, as shown in figures 6 (a) & (b).

5.2   Extrude feature

Create a base protrusion (stock material) by selecting the Extrude Tool icon ( ) from the right hand icon-bar, this will initiate the 'Extrude Dashboard'

( ) just below the message window.

It is possible to use an exisiting sketch as the basis of a sketched feature, here however use the internal sketch option by RMB→Define Internal Sketch.
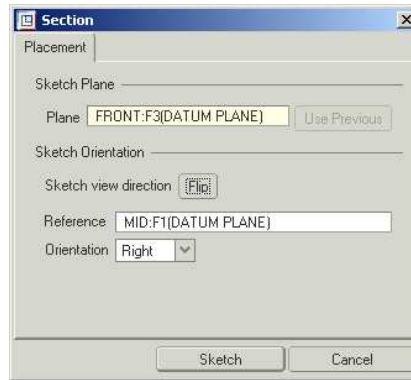
Section Menu

1. **Sketch Plane**

   This will open a SECTION window and a prompt in the message window to "Select a plane or surface to define sketch plane". Moving the mouse over the main window will highlight (in light blue) each of the possible sketch planes.

   Using the LMB, select the plane FRONT when highlighted or the Model Tree window.

   Immediately the menu will be populated with default values and also indicated on the main window.

   An arrow indicates the direction of view of the sketch plane

   The direction can be changed using the **Flip** menu item, in this example any direction will be suitable.

2. **Sketch Orientation** The sketchplane can be presented on the screen in any angle and a preferred orientation can be selected by using the **reference** and **orientation** menu items. In most cases ProEngineer makes an intelligent guess at the required orientation and therefore the default can be accepted.

   In this case choose the defaults, by selecting **Sketch**.

   N.B. Sketch is highlighted on a raised button in the menu and can thus be selected as the default option in the menu by pressing the MMB (with the pointer in the main window), see above.

   N.B. Standard extrusions can be initiated more simply by using the Ext 1 Side**,** Ext Revolve **and** Ext 2 Side**, (Extrude 1 side, Extrude revolve and Extrude 2 sides), from the PART menu. N.B. This requires the CUED** New Part **option to have been used.**

3. **Dimension References**

   To automatically dimension any entity drawn the system needs a local dimensioning reference. When the sketcher mode is entered a REFERENCES

Reference Menu

window is displayed which lists the entities that have been choosen as default references and indicated in the sketching window by brown dash-dot-dot lines.

In this example the default references should be 'F1(MID)' and 'F2(TOP)' thus select (LMB) **Close** . Any further drawing on this plane will be referenced to this temporary coordinate system.
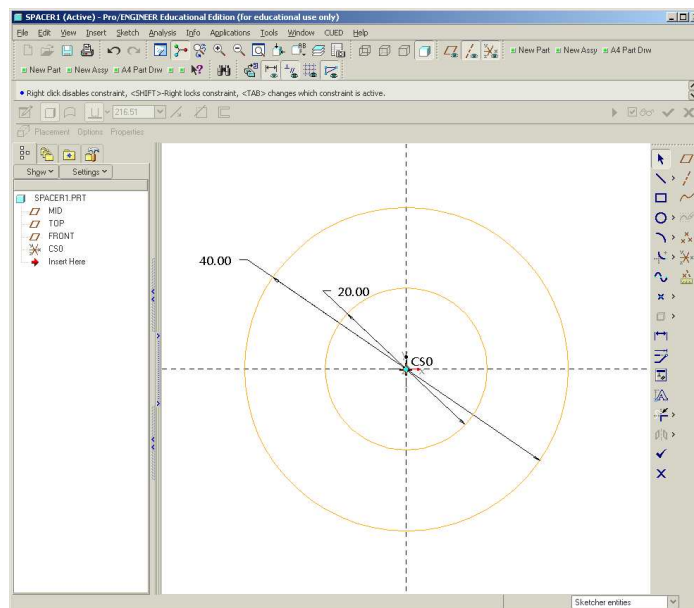 **(N.B. It can be useful at this stage to Deselect the Datum Plane icon to simplify display)**

4. **Sketching a Section**

Figure 7: Initial sketcher mode layout.

Using the sketch circle function, accessed via either the circle icon (3rd icon down) or by selecting the **Circle** entry on the floating menu (Opened by using the RMB while the pointer is over the main drawing window), draw two concentric circles, of arbitary size, centered on the intersection of the reference lines, see Figure 7.

To draw a circle move the pointer, (modified to a cross once the circle function has been selected) over the intersection, click/select (LMB) to set the centre

then drag the pointer and click/select to define the circle radius. Click LMB once to place circle, click MMB once to end the operation and to return to standard pointer.

To delete unwanted items, select the entity (line will turn red) and then select delete by holding down RMB → Delete. Multiple entities can be selected by dragging a boundary using the LMB

(Should artifacts be left on the window use VIEW → Repaint, **the Repaint icon on the top menu bar (** **) or the hot-key sequences (CTRL) + R or vr) to refresh the screen.**

Note :
(a) The pointer has 'intellegence' and snaps as it approaches an intersection/circle centre/line etc
(b)Pressing MMB once while drawing an entity will abort the operation and return to the standard select option.
Once drawn the system will allocate default dimensions to the entities, which are displayed in grey. The values are based on the screen resolution/setup and any previous components of the part.

(It is worthwhile spending some time familiarising yourself with the sketcher mode - add lines, rectangles etc and then delete them.)

Although the dimensions for our spacer are not correct at this stage the sketch is complete and the sketcher mode can be exited be selecting the 'Tick' at the bottom of the side icon bar or from the pull-down menu SKETCH → **Done**. This changes the colour of the extrusion to a trasparent yellow and reenables the protrusion dashboard.

5. **Solid/Surface (** **)**

    Leave the default setting of the second and third dashboard icons, e.g. the "extrude as solid" option (secon icon) selected.

6. **Depth (** **)**

    The left depth icon allows the user to select details of the extent of the extrusion, accept the default ("Extrude from sketch plane by a specified depth value.")

    Adjust the depth to *7.5* by chosing one of the following techniques:

    (a) Enter value in the field on dashboard

    (b) Selecting and dragging the square handle (small white square, on axis) on the extrusion in the main window. Note it can be difficult to acccurately choose a value using this technique.

    (c) Double clicking, with LMB, on the dimension in the main window and entering the required value.

7. **Verify/Preview (** **)**

    The defined feature can now be temporarily previewed by verify icon ( ), from the right hand end of the dashboard. If an error exists individual elements of the protrusion can be modified by re-selecting the appropriate icon.

8. **Exit** (✔ ✖)

   Once completed the protrusion can be accepted by selecting the tick (✔).
   N.B. Selecting the cross will cancel the generation of the feature.

5.3 Review/Display

The Protrusion can now be rotated and/shaded to improve the view of the feature.

1. Press the MMB to rotate the model. Use SHIFT and CTRL to zoom and pan.

2. Select **Shading**, **Wireframe** etc icons (▱ ▱ ▱ ▱)

3. You can also reset the view and repaint it. These options are available by
   choosing **VIEW** pull down menu or from the icon bar (🔍) (🖼) .

5.4 Edit Profile Dimensions

There are several ways of modifying a features dimensions

1. Select the feature (Protrusion idxxx) in the Model Tree or drawing screen,
   (Highlight protrusion feature in light blue and select with LMB) and the using
   the RMB select **Edit**. The defining dimensions are displayed in the main
   window, Select the value to be modified (double click LMB) and enter the
   value at the prompt.

2. Again select the required feature and in the flyout menu (RMB) select **Edit
   Definition** which will return to the protrusion dashboard.

Modify the inner diameter to _4.0_, the outer diameter to _6.4_ and ensure the length is
set to _7.5_. N.B. Dimension values switch from yellow to green when they have been
modified.

To action the modifications select **Regenerate** from the EDIT menu, the image will
animate to the new dimensions.

 **(N.B. The regenerate function can also be actioned by (a) Using the hotkeys
rg (b) Selecting the Regenerate icon** 📑 **)**

5.5 Save Part

Use the FILE → **Save** function or save icon (💾) to save the part in its current
form.

5.6 Exiting ProEngineer Wildfire

If you need to exit ProEngineer, select FILE→**Exit**. The system will the prompt the user to save each of the components, constructed to date, in turn.

(To speed the exit process type either $\underline{a}$ or $\underline{q}$ (save all items or discard all items, respectively) at the prompt)
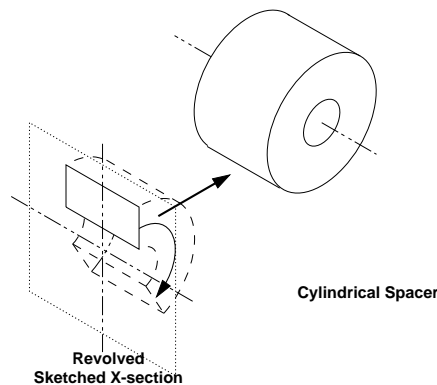
6 'Revolved' Parts

Figure 8: Revolve Tool used to generate Spacer

This example demonstrates an alternative method of producing a spacer, using a revolved cross-section to define the solid material. The procedure is very similar to the previous example and therefore less explanation has been included.

6.1 New part - shaft_spacer_short

Select the NEW PART icon again and name the new part *shaft_spacer_short*. Selecting **OK** will set up the default datum planes as in the last section

6.2 Revolved feature

Select the Revolve Tool icon (⚙) and then systematically work from left → right along the dashboard

- **Section**

  Use the internal sketch option (RMB→Define Internal Sketch) and choose FRONT as the sketch plane and accept the defaults for the direction, reference plane and orientation.

  Accept the default drawing references

  To complete the section it is neccessary to define both the axis of rotation and the revolved section

  1. To define the axis of rotation select RMB → **Centreline** to choose the centreline tool. To place select (LMB) the intersection of the reference line and drag and reselect vertically above to produce a vertical centreline aligned to the reference origin.

  2. Use either the **Rectangle** or **line** functions to sketch one side of the spacer parallel to the axis of rotation, see Figure 9
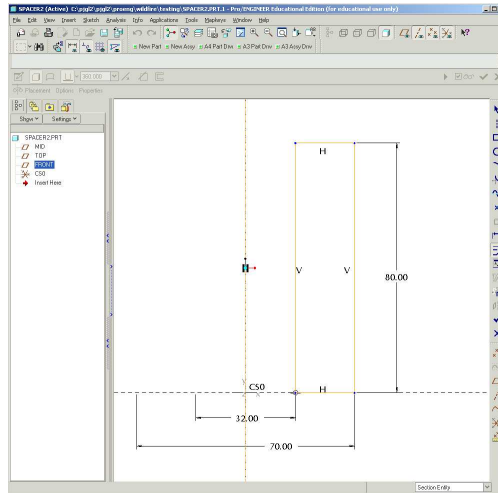
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Figure 9: Axis of rotation and cross-section for the revolved section.

> **By default the soft dimensions chosen by the system are shown as one radius + a thickness. To redefine dimensions, select the RMB → Dimension tool and then select the two ends of the dimension required and then MMB to place the dimension.**

3. Exit the sketcher mode using the 'Tick' at the bottom of the side icon bar

- **Edit** Experiment using the drag handle to adjust the angle of rotation and then ensure that the dimension field is 360°.
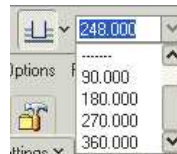
> **The dimension pulldown menu has standard dimensions and recently used dimensions, useful when switching between trial values.**

- **Preview** ( ) Use the Verify/Preview icon to check the completed revolved entity

- **Exit** ( ) Exit the revolve tool

6.3   Edit

. Using LMB to select and the RMB → **Edit** command change the inner radius to _4.4_, the outer radius to _7_ and the length to _7.5_. Do not forget to regenerate (select , **Regenerate** from the EDIT menu) or hotkey 'rg' to update the values.

6.4   Save Part

Use the FILE → **Save** function or save icon ( ) to the save the part in its current form.

7 'Reuse' of parts

A major benefit of using a CAD system is that components can be reused. Reload either the long or short shaft spacer using the open icon (📂) or FILE → OPEN.

7.1 plate_spacer

Using the edit funtions change the dimensions to that of the plate spacer, ID _3.2_mm, OD _6_mm and length _26_mm. (See figure 5 for more details)

Then use the FILE→**Save A Copy** option to save the changed object as _plate_spacer_, at the **New Name** prompt.
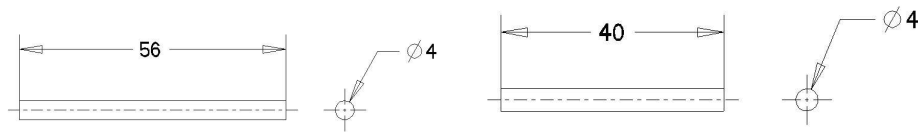

Figure 10: Dimensions of the output and lay shafts

8 Further examples - (Optional)

8.1 shaft_output, shaft_lay

Again using techniques developed above model and save the two shafts used in the gearbox, _shaft_output_ and _shaft_lay_ (see Figure 7.1.)
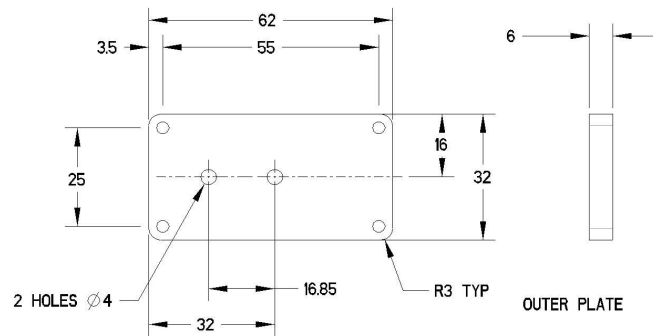
9 Multi-feature Part

Figure 11: Multi-feature construction (Extrusion - Cut - Drill - Round) used to generate the outer plate

This Task demonstrates the use of multiple sketched and pick/place features to define a end plate

As with manufacturing a physical item it is often better to build a CAD model up in a number of stages. This eases problems with editing and allows for simpler reuse of the parts. In this case it is suggested to draw the plate in four stages

1. Basic rectangular plate (stock material)

2. 2 circular cuts, for the shafts

3. 4 outer holes,

   - single hole
   - pattern

4. Rounded courners

To start select the NEW PART icon again and name the new part *endplate1*→**OK**.

9.1 Base extrusion - outer_plate

Use the standard extrude feature to pruduce the base rectangular shape centred on the dimension references, width *62* height *32*.

9.2 Cuts

Again using the extrude feature draw the two holes Φ4 mm centered on the centerline of the plate, but this time use the remove material function ( ). to generate the holes.

(☑👓) Use the Verify/Preview icon to check the cut is correct.

If required, use the material direction function (✄) to swap the side of the section that is cut.

9.3   Holes

The four mounting holes around the edge of the plate could be cut as the bearing holes, but here use the standard hole feature.

Using the (𝖨𝖴) hole feature generate a 3.2 mm diameter circular hole through the plate in one of the corners. Figure 12 shows the hole dashboard. Note if you wish to draw a shaped/threaded hole there are a number of predefined forms available from the menus, see also Figure 12 (b).

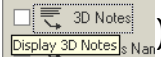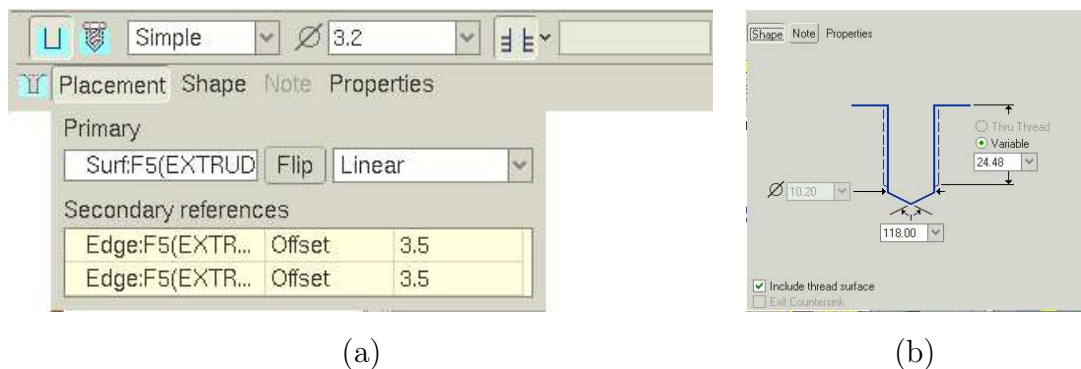(a)   (b)

Figure 12: Hole dashboard. (a) Placement options & (b) Shape Options

1. **Hole Placement** requires three references the initial placement plane and two entities to dimension the hole in x/y space.

   - Select a side of the plate as the placement plane.
   - Select the edges of the plate as the dimension references and align the hole and set both offsets **Distances** to 3.5 mm.
     Use <**ctrl**> LMB to select the second edge, alternatively drag placement handles (small white squares) until they latch to an edge.

2. Select **Hole Type → Straight Hole**

3. Set **Hole Dimension → Diameter** 3.2 mm and **Depth One** Thru All

4. Preview/(Define)/Tick before inspecting the resultant component.

 N.B. A note is added to give information about the hole. To switch this off select TOOLS → Environment **and then deselect** 3D Notes([▭ ☰ 3D Notes / Display 3D Notes s Nan] **) or use the hotkey '3d'.**

9.4 Pattern

It would be possible to additional holes individually, however it is often very useful to pattern such features.

- Select the one mounting hole, this will enable the pattern icon ( ). Select the icon or use RMB→Pattern. This will initate the pattern dashboard that gives access to a number of petterning options, here a linear pattern in 2 directions is used.
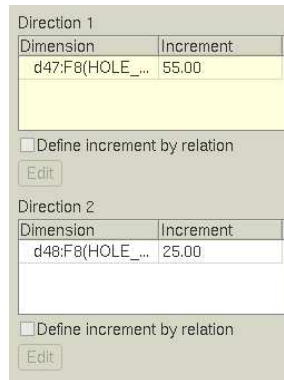

Figure 13: Pattern dimensions/increment sub menu

- Select the horizontal 3.5 mm hole offset dimension as the first pattern driving dimesion and in the Dimension/Direction 1 menu set the increment to *55* mm.

- Select to add a second direction and use the second hole offset with an increment set to *25* mm.

- Set the number of duplicates to 2 in both directions on the top level of the dashboard.

- Exit the tool -NB there is no preview option for this tool

9.5 Rounds

The **Rounds** & **Chamfer** features are normally added as the last constructional details on an object, as in the physical construction process. This eliminates unwanted problems resulting from subsequent features removing the reference edge/surface and also to reduce the computational burden of continuously displaying complex surfaces.

Selecting the round tool icon ( ) or (INSERT→Round ....) starts the 'round' dashboard and a message prompt 'Select and edge or chain of edges, or a surface to create a round set'.
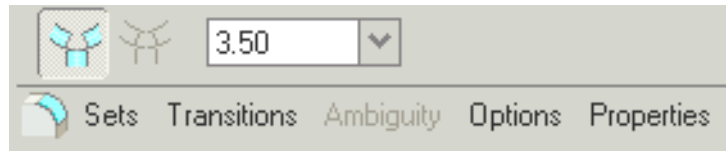
- **Radius** Set the round radius to *3* mm.

- **Edges** Although there are a large number of options, accept the defaults and using the highlight/select function select the four corner edges of the plate. N.B. the most recently accepted edge is highlighted in red and is dynamically adjustable.

- **Review** Check, preview and Exit round tool

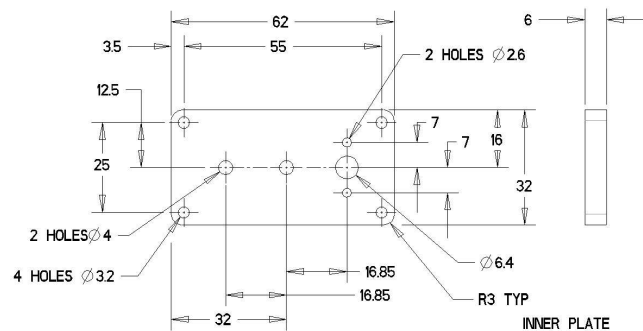# 10 Reuse of plate - plate_inner

Figure 14: Multi-feature construction (Extrusion - Cut - Drill - Round) used to generate the outer plate

Using the option to reuse a part further develop the model for the plate, adding mounting holes etc for the motor and save as *plate_inner*

11 2D Drawings

Even though the direct link from CAD to CAM (Computer aided manufacture) is increasingly common there is still a need to produce 2-D drawings, in ProEngineer this is a relatively painless procedure as the hard work has already been completed in the model generation.

Note that the drawing views are associative. If a dimension is changed in one view and regenerated, all of the views update and so will the entire model.

11.1 Start a Drawing

(By defalt the system attempts to draw the currently active solid model, so to draw the outer plate, ensure that this is selected. If required reselect the window, use the WINDOW menu, or reload/open the model)

To open a new drawing select FILE→**New**→**drawing**. Enter the name of the drawing, _plate_outer_ and select **OK**

The NEW DRAWING menu should now appear, ensure that the default model field is _plate_outer.prt_, ensure the **Empty with format** option is selected and the format field is _a4_part.frm_. Selecting OK will initiate the drawing window and open the DRAWING and DETAIL menus.

11.2 First View

To generate and locate the first view

- **Insert View** select RMB → **Insert General View** or INSERT →**Drawing View** → **general**.
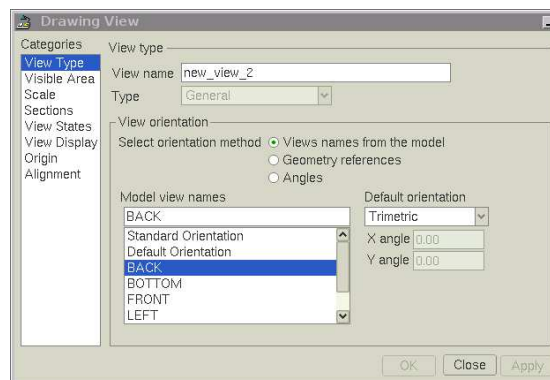

Figure 15: Setting initial view orientation

- **Placement** The user is now requested to choose the location for the master view by a prompt in the message window 'select CENTER POINT for drawing view. Selecting (LMB) a position near the bottom right of the drawing surface will result in a default (isometric) view being placed followed

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by the opening of the DRAWING VIEW window, see Figure 11.2, set in the VIEW TYPES mode.

- **View Orientation** The Orientation functions can be used to obtain the direction of a view. Using the MDP start part a number of predefined named views are available and normally enable the initial view to be placed simply. See the scrollable list shown when the **views names in model** option is selected.

  A suitable master view for this drawing is probably FRONT or BACK, select the view e.g. **Front** and then **Apply**. Review and correct if required. Select **OK** to exit back to the drawing.

- **Scale, Display type etc**

  The DRAWING VIEW menu also allows many other display formats to be set, including scale and individual views line types. This menu can be selected via RMB→ **properties** when a view is selected.

11.3 Additional Views

Additional views can now be added, with the view orientated automatically. Using the default menu choices add two more views to your drawing.

- **Add View** Use RMB → **Insert Projection View**, ( ) or INSERT →**Drawing View** →**Projetion** to initiate the generation of new views.

   N.B. Once the initial view has been placed the system assumes that subsequent views are projections.

- **Placement** Use the LMB over the drawing area to select the position of the two views, the system will ensure that the correct projection is shown.

- **Relocate views** Using LMB to select a view (highlighted in with a red box) and then holding LMB down drag the view to the required location. Select with LMB in a clear area of the drawing to deselect a view. (If the view does not move deselect the **Lock View Movement** option from the RMB context sensitive menu)

- **Set Display mode** Select the **Hidden Line** display option ( ) on the icon bar and deselect the datum icons to display a clean drawing. N.B. **No Hidden** (and **Wireframe**) can be used if required.

- **Setting the Drawing Scale** To change the overall scale of a drawing double click LMB on the scale information at the bottom left of the main window. At the prompt line enter the value you require, _0.4_ is a suitable scale for this drawing.

   N.B. Normally drawings scales are limited to multiples of standard scales, e.g. 1:1 1:2 1:2.5 1:4 1:5 (1, 0.5, 0.4, 0.25, 0.2)
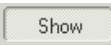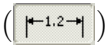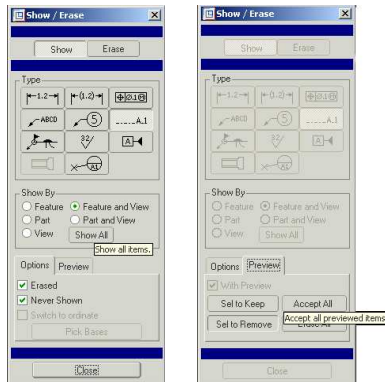
Figure 16: (a) Show-Erase Menu (b) Accept All

11.4 Display Centerlines

Select VIEW → **Show/Erase** or ( ) to open the SHOW/ERASE window.

Select the axis icon and **Show All**, see Figure 16(a), confirm 'Are you sure that you want to show all?' and then **Accept All**

11.5 Drawing Dimensions

To dimension the drawing use the same 'Show and Erase'function ( ) as above (section˜refsec:clines). Select **show** (Show) and **dimension** (⊢1.2⊣) → **ShowAll** → **Accept All**.

11.6 Tidy Drawing

The position of symbols and text can be moved using the LMB to select a item which can then be dragged to the required position. MMB to stop, LMB to exit move.

A large number functions are available via the RMB context sensitive menus, see Figure 17, and from the INSERT and FORMAT menus. (See FORMAT → **Decimal Places**.. to change default number display and therefore implied accuracy)

Use the functions, details about some are given below, to tidy the dimensioning etc. (Also see Figure 18)

- Cleanup Dimensions: Multiple selection → RMB automatically aligns dimensions on a user definable spacing.

- Move Item to View: Single or Multiple selection allows dimension etc to be switched between views.

- Flip Arrows: Single or Multiple selection allows dimension arrows to be realigned
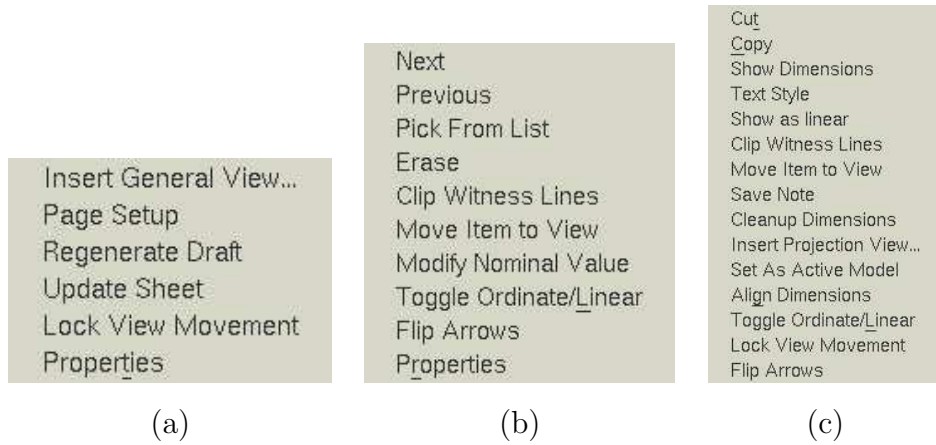
| (a) | (b) | (c) |

Figure 17: Examples of context sensitive Pull out Menus, (a) On background (b) On selection of 1 dimension (c) Multiple dimension selection

- Text Style: Change text in multiple dimensions etc.

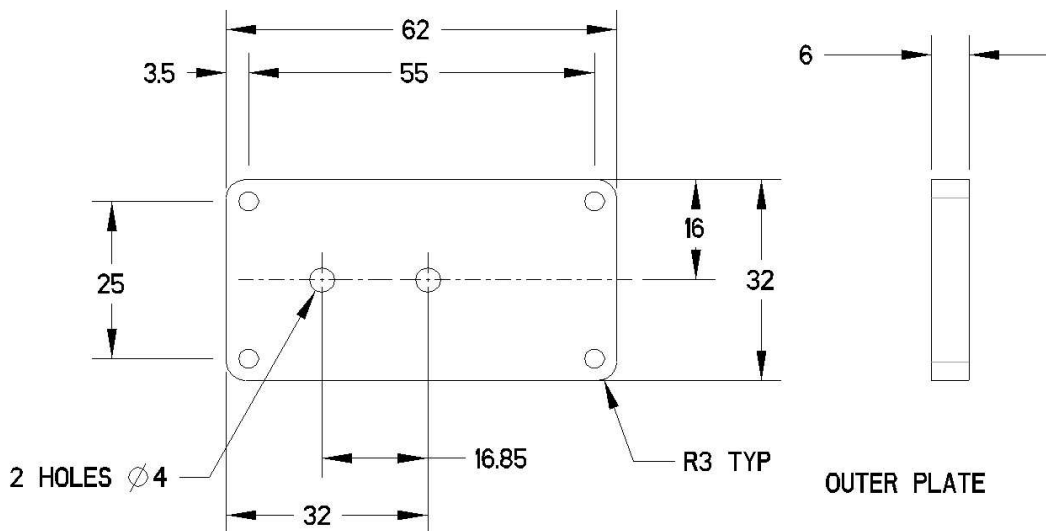- Properties : Gives access to the DRAWING VIEW menu

Figure 18: Example of a dimensioned drawing of the Outer_Plate

11.7   Isometric View

The advantage of CAD is that it is relatively easy to add additional views. It is often useful to add an Isometric view to aid visualisation. To add a general view select RMB → **Insert General View** or (⊞)

Set specific view properties from the DRAWINg VIEW menu, e.g. **Scale**, Figure 19.
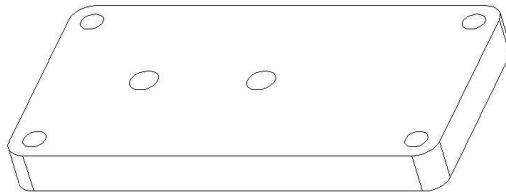
SCALE   1.000

Figure 19: Example of a view specific scaled general view the Outer_Plate

11.8   Printing a Drawing

N.B. By default the system prints the drawing that is visible on the screen, To print the whole drawing either select VIEW→ ORIENTATION → View Refit or use LMB + CTRL to zoom to the required view.

- Select the printer icon or **Print** from the **File** pull down menu.

- Select **Generic Postscript**

- Enter printer command *lp -dxxxx*, where xxxx is the printer name, e.g. ljmr1 for the printer at CUED.