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Modelling with AutoCAD 2002

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Preface

This book is intended for the AutoCAD 2002 user who wants to learn about modelling. My aim is to demonstrate how the user can create 3D wire-frame models, surface models and solid models with practical exercises backed up by user activities. The concept of how multiple viewports can be used to enhance drawing productivity will also be discussed in detail. The user will also be introduced to rendering.

The book will provide an invaluable aid to a wide variety of users, ranging from the capable to the competent. The book will assist students on any national course which requires 3D draughting and solid modelling, e.g. City and Guilds, BTEC and SQA as well as students at higher institutions. Users in industry will find the book useful as a reference and an 'inspiration'. The book will also prove useful to the Design/Technology departments in schools who are now becoming more involved in computer aided design.

Reader requirements

The following are the requirements I consider important for using the book:

- a) the ability to draw with AutoCAD 2002
- b) the ability to use icons and toolbars
- c) an understanding of how to use dialogue boxes
- d) the ability to open and save drawings to a named folder
- e) a knowledge of model/paper space would be an advantage, although this is not essential

Using the book

The book is essentially a self-teaching package with the reader working interactively through exercises using information supplied. The various prompts and responses will be listed in order and icons and dialogue boxes will be included where appropriate.

The following points are important:

- a) All drawing work should be saved to a named folder. The folder name is at your discretion but I will refer to it as **MODR2002**, e.g. open drawing MODR2002\MODEL1 or similar
- b) Icons will be displayed the first time is used
- c) Menu bar selection will be in bold type, e.g. **Draw-Surfaces-3D Face**
- d) Keyboard entry will also be in bold type, e.g. **VPOINT, UCS** etc
- e) Prompts will be in typewrite type, e.g. `First corner`
- f) The symbol **<R>** will require the user to press the return/enter key.

Note

All the exercises and activities have been completed using AutoCAD 2002. I have tried to correct any errors in the drawings and text, but if any error should occur, I apologise for them and hope they do not spoil your learning experience. Modelling is an intriguing topic and should give you satisfaction and enjoyment.

Any comments you have about how to improve the material in the book would be greatly appreciated.

To CIARA, our beautiful
grand-daughter

The 3D standard sheet

To assist us with the models which will be created, a standard sheet (prototype drawing) will be made with layers, a text style, dimension styles etc. This standard sheet will be saved as both a drawing file and a template file. It will be modified/added to as the chapters progress.

1 Start AutoCAD 2002 and:

prompt Startup dialogue box
respond **pick Use a Wizard**
prompt Startup (Use a Wizard) dialogue box
respond **pick Advanced Setup then OK**
prompt Advanced Setup dialogue box
respond select the following to the various steps:
a) Step 1 Units: Decimal; Precision 0.00; Next>
b) Step 2 Angle: Decimal Degrees; Precision 0.0; Next>
c) Step 3 Angle Measure: East(0); Next>
d) Step 4 Angle Direction: Counter-Clockwise(+); Next>
e) Step 5 Area: Width 420 and Length 297 (i.e. A3)
then pick Finish
and a blank screen will be displayed

2 *Layers*

Menu bar with **Format-Layer** and make the following new layers:

<i>name</i>	<i>colour</i>	<i>linetype</i>
MODEL	RED	continuous
TEXT	GREEN	continuous
DIM	MAGENTA	continuous
OBJECTS	BLUE	continuous
SECT	number: 96	continuous
0	white	continuous

NB: other layers will be added as required.

3 *Text style*

Menu bar with **Format-Text Style** and make a new text style:

Name: ST1

Font: romans.shx

Height: 0; Width factor: 1; Oblique angle: 0

Apply then Close then dialogue box

4 *Units*

Menu bar with **Format-Units** and:

Units: Decimal with Precision: 0.00

Angle: Decimal Degrees with Precision: 0.0

Drawing units for DesignCenter blocks: Millimeters

5 Limits

Menu bar with **Format-Drawing Limits** and:

prompt Specify lower left corner and enter: **0,0 <R>**

prompt Specify upper right corner and enter: **420,297 <R>**

6 Drafting Settings

Menu bar with **Tools-Drafting Settings** and use the tabs to set:

a) Snap: 5 and grid: 10 – not generally used in 3D

b) Polar Tracking: off

c) Object Snap: off and all modes: clear

Object Snap Tracking: off

7 Dimension style

Menu bar with **Dimension-Style** and:

prompt Dimension Style Manager dialogue box

respond **pick New**

prompt Create New Dimension Style dialogue box

respond

1. New Style Name: **3DSTD**
2. Start With: ISO-25 (or similar)
3. Use for: All dimensions
4. pick Continue

prompt New Dimension Style: 3DSTD dialogue box

respond

pick Lines and Arrows tab and alter:

1. Dimension Lines
 - a) Baseline spacing: 10
2. Extension Lines
 - a) Extend beyond dim lines: 2.5
 - b) Offset from origin: 2.5
3. Arrowheads
 - a) both Closed Filled
 - b) Leader: Closed Filled
 - c) Arrow size: 4
 - d) Center Mark for Circles: None

then

pick Text tab and alter:

1. Text Appearance
 - a) Text Style: ST1
 - b) Text Height: 5
2. Text Placement
 - a) Vertical: Above
 - b) Horizontal: Centred
 - c) Offset from dim line: 1.5
3. Text Alignment
 - a) ISO Standard

then

pick Fit tab and alter:

1. Fit Options
 - a) Either the text or the arrows active (black dot)
2. Text Placement
 - a) Beside the dimension line active
3. Scale for Dimension Features
 - a) Use overall scale of: 1
4. Fine tuning: both inactive, i.e. blank

then **pick Primary Units tab and alter:**

1. Linear Dimensions
 - a) Unit Format: Decimal
 - b) Precision: 0.00
 - c) Decimal separator: '.' Period
 - d) Round off: 0
2. Measurement Scale
 - a) Scale factor: 1
3. Zero Suppression
 - a) Trailing: active, i.e. tick
4. Angular Dimensions
 - a) Units Format: Decimal Degrees
 - b) Precision: 0.0
 - c) Zero Suppression: Trailing active

then **pick Alternate Units tab and:**

1. Display alternate units: not active

then **pick Tolerances tab and:**

1. Tolerance Format
 - 1 Method: None

then **pick OK from New Dimension Style dialogue box**

prompt Dimension Style Manager dialogue box

- with*
1. 3DSTD added to styles list
 2. preview of 3DSTD style displayed
 3. description of 3DSTD given

- respond*
1. pick 3DSTD and it becomes highlighted
 2. pick Set Current
 3. AutoCAD alert perhaps – just pick OK
 4. pick Close

8 Make layer 0 current and menu bar with **Draw-Rectangle** and:

prompt Specify first corner point and enter: **0,0 <R>**

prompt Specify other corner point and enter: **420,290 <R>**

9 This rectangle will save as a 'reference base' for our models

10 Menu bar with **View-Zoom-All** and pan to suit

11 Make layer MODEL current

12 Set variables to your own requirements, e.g. GRIPS, PICKFIRST, etc. While I generally work with these off, there will be occasions when they will be toggled on

13 Menu bar with **File-Save As** and:

prompt Save Drawing As dialogue box

- respond*
1. scroll and pick named folder (MODR2002)
 2. enter File name: **3DSTDA3**
 3. file type: **AutoCAD 2000 Drawing (*.dwg)**
 4. pick Save

14 Menu bar with **File-Save As** and:

prompt **Save Drawing As** dialogue box

respond

1. scroll at Files of type
2. pick **AutoCAD Drawing Template File (*.dwt)**
3. scroll and pick named folder
4. enter File name as: **3DSTDA3**
5. pick Save

prompt Template Description dialogue box

respond

1. Enter: This is my 3D standard sheet
2. pick OK

15 The created standard sheet has been saved as a drawing file and a template file, both with the name 3DSTDA3. Both have been saved to the MODR2002 named folder – or the name you have given the folder to save all modelling work.

16 *Note*

- a) we could have saved the template file to the AutoCAD Template file – you still can if you want
- b) saving the standard sheet as a template will stop the user ‘inadvertently’ over-writing the basic 3DSTD standard drawing sheet
- c) all models will be created from the 3DSTDA3 template file
- d) all completed models will be saved as drawings to your named folder
- e) the standard sheet has been saved as a drawing file as backup

We are now ready to proceed with creating 3D and solid models.

Extruded 3D models

An extruded model is created by extruding a 'shape' upwards or downwards from a horizontal plane – called the ELEVATION plane. The actual extruded height (or depth) is called the THICKNESS and can be positive or negative relative to the set elevation plane. This extruded thickness is **always perpendicular** to the elevation plane. The extrusion is in the Z direction of the UCS icon – more on the UCS later. The basic extruded terminology is displayed in Fig. 2.1.

Note: Extruded models were one of the first ever 3D displays with a CAD system. The term 3D model is not quite correct, a more accurate description being 2½D model.

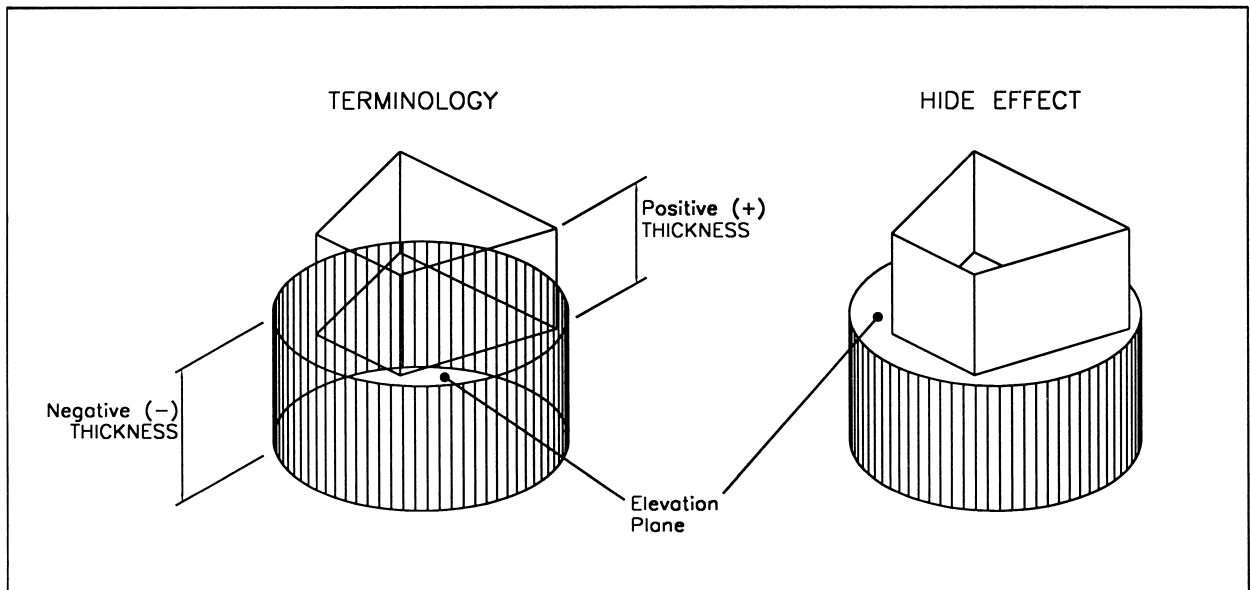


Figure 2.1 Basic extruded terminology.

Example 1

The example is given as a series of user entered steps, these steps also being displayed in Fig. 2.2. The exercise will introduce the user to some of the basic 3D commands and concepts.

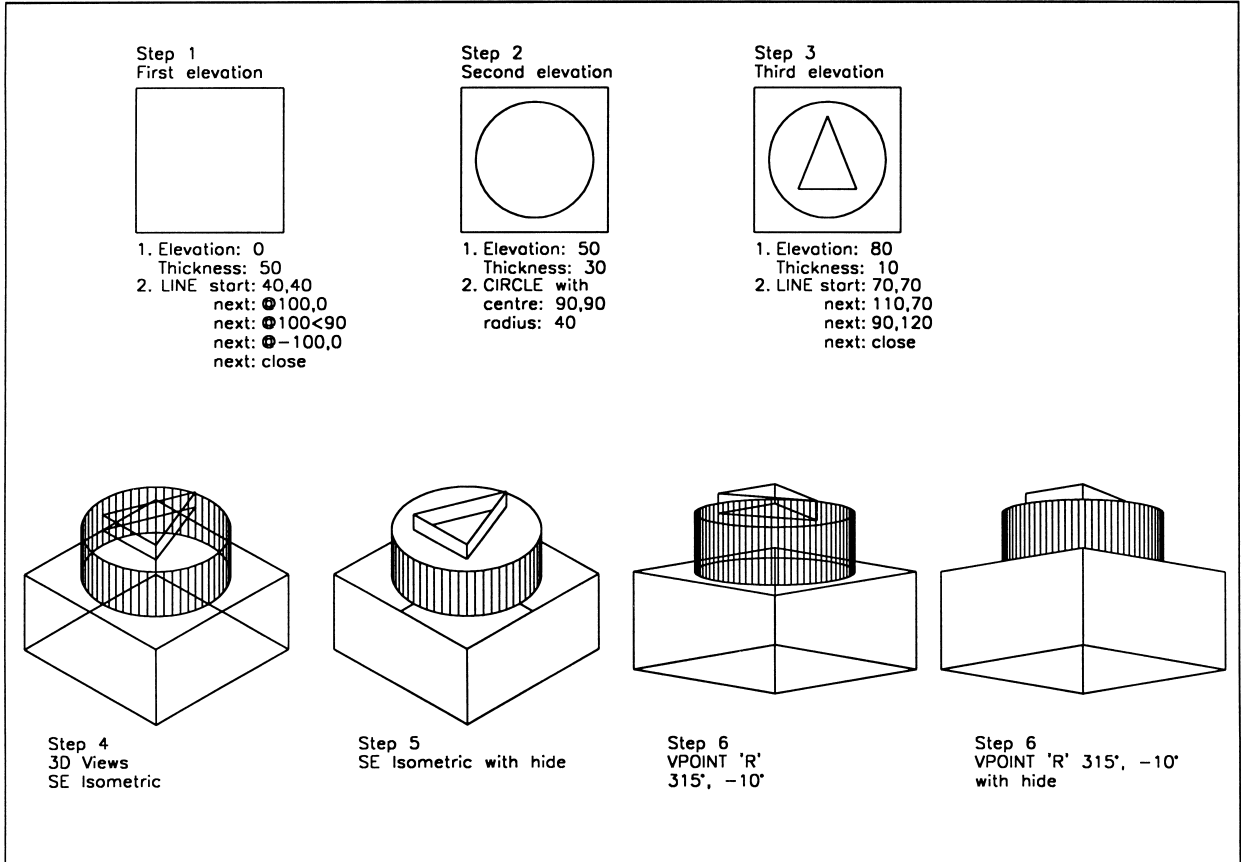


Figure 2.2 Extruded example 1.

To get started:

- 1 Open your 3DSTDA3 template file and display toolbars to suit e.g. Draw, Modify and Object Snap.
- 2 Layer MODEL should be current.

Step 1: the first elevation

1. At the command line enter **ELEV <R>** and:
prompt Specify new default elevation<0.00> and enter: **0 <R>**
prompt Specify new default thickness<0.00> and enter: **50 <R>**
2. Nothing appears to have happened?
3. Select the LINE icon and draw:
Start point: **40,40 <R>**
Next point: **@100,0 <R>**
Next point: **@100<90 <R>**
Next point: **@-100,0 <R>**
Next point: **C <R>** – the close option
4. A red 'square' will be displayed.

Step 2: the second elevation

1. At the command line enter **ELEV <R>** and:
 - prompt* Specify new default elevation<0.00> and enter: **50 <R>**
 - prompt* Specify new default thickness<50.00> and enter: **30 <R>**
2. Select the CIRCLE icon and:
 - a) centre point: enter **90,90 <R>**
 - b) radius: enter **40 <R>**
3. At the command line enter **CHANGE <R>** and:
 - prompt* Select objects
 - respond* **pick the circle then right-click**
 - prompt* Specify change point or [Properties]
 - enter* **P <R>** – the properties option
 - prompt* Enter property to change [Color/Elev/Layer/Ltype etc
 - enter* **C <R>** – the color option
 - prompt* Enter new color
 - enter* **green <R>**
 - prompt* Enter property to change
 - respond* **right-click and pick Enter**
4. The added circle will be displayed with a green colour

Step 3: the third elevation

1. With the ELEV command:
 - a) set the default elevation to 80
 - b) set the default thickness to 10
2. With the LINE icon, draw:
 - Start point:* **70,70 <R>**
 - Next point:* **110,70 <R>**
 - Next point:* **90,120 <R>**
 - Next point:* **C <R>**
3. With the CHANGE command, change the colour of the three lines to blue, using the same procedure as was used previously.
4. We now have a blue triangle inside a green circle inside a red square, and appear to have a traditional 2D plan type drawing.
5. Each of the three shapes has been created on a different default elevation plane:
 - a) square: elevation 0
 - b) circle: elevation 50
 - c) triangle: elevation 80

Step 4: viewing the model in 3D

To 'see' the model in 3D the 3D Viewpoint command is required, so:

1. From the menu bar select **View-3D Views-SE Isometric**
2. The model will be displayed in 3D. The black 'drawing border' is also displayed in 3D and acts as a 'base' for the model.
3. The orientation of the model is such that it is difficult to know if you are looking down on it, or looking up at it. This is common with 3D modelling and is called **AMBIGUITY**. Another command is required to 'remove' this ambiguity.
4. At this stage save your model with **File-Save As** and ensure:
 - a) File type is: AutoCAD 2000 Drawing (*.dwg)
 - b) Save in: MODR2002 – your named folder
 - c) File name: **EXT-1** – the drawing name
5. This saves the drawing as **C:\MODR2002\EXT-1.dwg** – the path name

Step 5: the hide command

1. From the menu bar select **View-Hide** and the model will be displayed with hidden line removal. It is now easier to visualise.
2. From the screen display it is obvious that the model is being viewed from above, but it is possible to view from different angles.
3. Menu bar with **View-Regen** to 'restore' the original model

Step 6: another viewpoint

1. At the command line enter **VPOINT <R>** and:
 - prompt* Specify a view point or [Rotate]
 - enter* **R <R>** – the rotate option
 - prompt* Enter angle in XY plane from X-axis and enter: **315 <R>**
 - prompt* Enter angle in XY plane and enter: **-10 <R>**
2. The model will be displayed from a different viewpoint without hidden line removal
3. At the command line enter **HIDE <R>**
4. The model will be displayed with hidden line removal and is being viewed from below
5. At the command line enter **REGEN <R>** to restore the original

Step 7: the shade command

1. Restore the original 3D view with the menu bar sequence **View-3D Views-SE Isometric**
2. Menu bar with **View-Shade-Flat Shaded** and the model will be displayed in colour. This is the result of the change command after the various objects had been drawn.
3. Note the icon – more on this later
4. Menu bar with **View-Shade-Gouraud Shaded** and note the effect on the model. Can you observe any difference between the flat shading and the Gouraud shading? Look at the 'cylinder' part of the model
5. Investigate the other SHADE options available
6. Restore the model to its original display with **View-Shade-2D Wireframe** and note the icon.

Task

- 1 With the ERASE command pick any line of the 'base' and a complete 'side' is erased because it is an extrusion
- 2 Undo the erase effect with **U <R>**
- 3 Using the erase command pick any point on the top 'circle' and the complete 'cylinder' will be erased.
- 4 Undo this erase effect.
- 5 This completes our first extrusion exercise.
- 6 *Note:*
Although Fig. 2.2 displays several different viewpoints of the model on 'one sheet' this concept will not be discussed until a later chapter. At present you will only display a single viewpoint of the model.

Example 2

This example will use a different method of changing the colour of the model objects – the Properties toolbar so:

- 1 Open your 3DSTDA3 template file, layer MODEL current and refer to Fig. 2.3
- 2 At the command line enter **PICKFIRST <R>** and:
prompt Enter new value for PICKFIRST
enter **I <R>**
and pickfirst box ‘attached’ to cursor cross-hairs

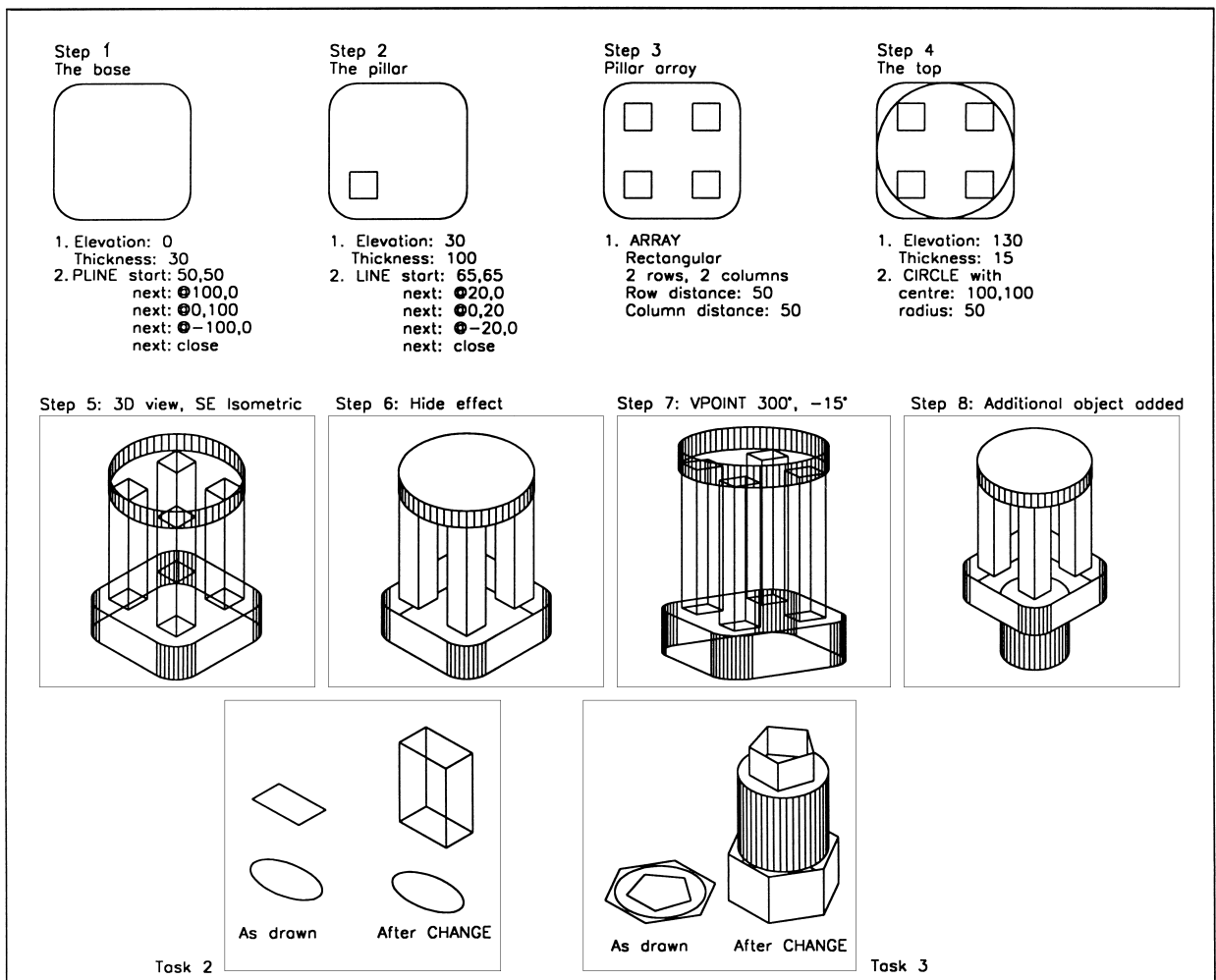


Figure 2.3 Extruded example 2.

Step 1: the base

1. With ELEV at the command line, set the new default elevation to 0 and the new default thickness to 30
2. With the polyline icon from the Draw toolbar, draw a 0 width polyline:
 - Start point **50,50 <R>**
 - Next point **@100,0 <R>**
 - Next point **@0,100 <R>**
 - Next point **@-100,0 <R>**
 - Next point **C <R>**
3. Menu bar with **Modify-Fillet** and:
 - prompt Select first object or [Polyline/Radius/Trim]
 - enter **R <R>** – the radius option
 - prompt Specify fillet radius
 - enter **20 <R>**
 - prompt Select first object [Polyline/Radius/Trim]
 - enter **P <R>** – polyline option
 - prompt Select 2D polyline
 - respond **pick any point on the polyline**
4. The red polyline will be filleted at the four corners

Step 2: the first pillar

1. Set the elevation to 30 and the thickness to 100
2. With the LINE command, draw a 20 unit square the lower left corner being at the point 65,65
3. Using the pickbox on the cursor, pick the four lines of the square then select the Properties icon from the Standard toolbar and:
 - prompt Properties dialogue box
 - respond
 1. pick Categorised tab
 2. pick Color line – highlights
 3. scroll at right of Color line
 4. pick Blue – Fig. 2.4
 5. Close the Properties dialogue box – top right pick
 6. press ESC key
4. The square will be displayed with blue lines

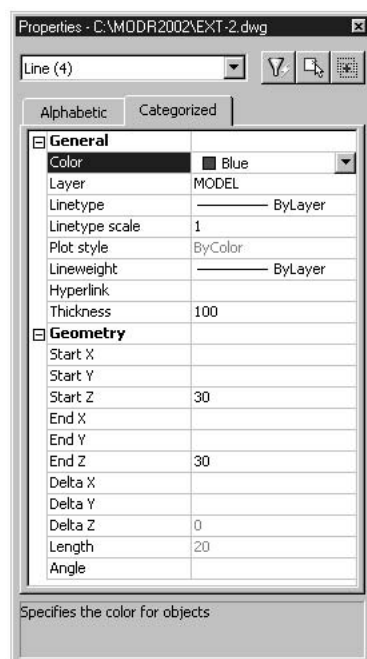


Figure 2.4 The Properties dialogue box for the selected square.

Step 3: arraying the pillar

1. Select the ARRAY icon from the Modify toolbar and:
 - prompt* Array dialogue box
 - respond* 1. Rectangular Array active
 - 2. Rows: 2; Columns: 2
 - 3. Row offset: 50 and Column offset: 50
 - 4. Angle of Array: 0
 - 5. pick Select objects and:
 - prompt* Select objects at the command line
 - respond* window the blue square then right-click
 - prompt* Array dialogue box
 - respond* pick **Preview<**
 - and* blue square arrayed as expected?
 - then* Array message and pick **Accept**
2. The blue square will be arrayed in a 2×2 matrix pattern

Step 4: the top

1. Set the elevation to 130 and the thickness to 15
2. Draw a circle, centred on 100,100 with radius of 50
3. Using the pickbox:
 - a) pick the circle then the Properties icon
 - b) set the colour to green

Step 5: the 3D viewpoint

1. Menu bar with **View-3D Views-SE Isometric**
2. The model is displayed in 3D but appears rather 'cluttered'

Step 6: hiding the model

1. Menu bar with **View-Hide** model displayed with hidden line removal
2. Menu bar with **View-Regen** to restore the original model

Step 7: setting another viewpoint

1. At the command line enter **VPOINT <R>** and:
 - prompt* Specify a new view point or [Rotate]
 - enter* **R <R>** – the rotate option
 - prompt* Enter angle in XY plane from X axis and enter: **300 <R>**
 - prompt* Enter angle from XY plane and enter: **-15 <R>**
2. Menu bar with **View-Hide** to 'see' the model from below
3. Menu bar with **View-Regen** to restore the original model
4. Restore the original 3D view with **View-3D Views-SE Isometric**

Step 8

1. The model should be displayed in 3D at a SE Isometric viewpoint
2. Using the command line, set the elevation to 0 and the thickness to -60
3. Draw a circle with centre at 100,100 and radius 30
4. The circle will be displayed in 3D as a 'cylinder'
5. Change the colour of the added 'cylinder' to magenta
6. As the model is complete, save as **C:\MODR2002\EXT-2**

Task 1

Use the menu bar with the following menu bar sequences:

- a) View-3D Views-SE Isometric
- b) View-Hide and note green circle display
- c) View-Shade-Flat Shaded and note colour effect and icon
- d) View-Shade-3D Wireframe
- e) View-Hide and note the green circle display
- f) View-Shade-Flat Shaded, Edges On
- g) View-Shade-2D Wireframe and note the green circle display
- h) View-Regen to 'restore' the original model

Task 2

- 1 Still with the SE Isometric viewpoint displayed
- 2 Set the elevation to 0 and the thickness to 100
- 3 With **Draw-Rectangle** create a rectangle anywhere on the screen
- 4 With **Draw-Ellipse-Center** create an ellipse anywhere on the screen
- 5 Both the rectangle and the ellipse will be drawn without any thickness, although the thickness was set to 100 in step 2
- 6 At the command line enter **CHANGE <R>** and:
 - prompt* Select objects
 - respond* pick any point on the rectangle then right-click
 - prompt* Specify change point or [Properties]
 - enter* **P <R>** – the Properties option
 - prompt* Enter property to change [Color/Elev/Layer etc]
 - enter* **T <R>** – the thickness option
 - prompt* Specify new thickness <0.00>
 - enter* **100 <R>**
 - prompt* Enter property to change
 - enter* **<R>** – to end command as no other properties to change
- 7 The rectangle will now be displayed in 3D with a thickness
- 8 Using the same sequence and entries as step 6, select the ellipse. No thickness will 'be added'
- 9 With the CHANGE command, alter the elevation of the ellipse to 50.

Task 3

- 1 Display a SE Isometric viewpoint and set the elevation and thickness both to 0. Layer MODEL still current
- 2 Draw the following objects:
 - a) polygon with 6 sides, centred on 0,0 and inscribed in a 50 radius circle
 - b) circle, centre on 0,0 with radius 40
 - c) polygon with 5 sides, centred on 0,0 and inscribed in a 30 radius circle
- 3 Set PICKFIRST to 0 then use the CHANGE command to alter the three objects with the following information:

<i>object</i>	<i>elev</i>	<i>thickness</i>	<i>colour</i>
6 sided polygon	0	50	red
circle	50	80	blue
5 sided polygon	130	30	green
- 4 Investigate the hide and shade commands and other 3D viewpoints
- 5 This exercise is now complete. Do not save these additions.

Summary

- 1 An extruded model is created from an elevation and thickness
- 2 Extruded models are created 'as sides'
- 3 The elevation and thickness values are usually set from the command line
- 4 The elevation and thickness of objects can be altered with:
 - a) command line CHANGE with PICKFIRST 0
 - b) Properties icon with PICKFIRST 1 – dialogue box method
- 5 Extruded models are viewed in 3D with the 3D Views command which will be discussed in detail in a later chapter
- 6 3D models are displayed with AMBIGUITY, i.e. are you looking down from the top or up from the bottom?
- 7 The HIDE command is used to display 3D models with hidden line removal. This removes the AMBIGUITY effect
- 8 The SHADE command gives useful displays with coloured objects.

Assignment

During the assignments you will frequently meet a character called MACFARAMUS. This august gentleman was a great architect in ancient times, but sadly most of his works have not been given the credit they deserve. Your first assignment is to create as a 2½D model, a famous structure of MACFARAMUS which consists of several traditional geometric shapes. (All the activity drawings are at the end of the book, starting on page 323.)

Activity 1: Coloured structure of MACFARAMUS

- 1 Open your 3DSTDA3 template file
- 2 Using the elevation and thickness method, create the 2½D model of the structure in 'plan' view using the following information:

<i>level</i>	<i>shape</i>	<i>elev</i>	<i>thick</i>	<i>colour</i>	<i>no</i>	<i>size</i>
1	square	0	50	red	1	150 sq
2	circle	50	120	blue	4	R 15 arrayed in R55
3	octagon	170	40	green	1	inscribed in R80
4	semi-circle	210	30	magenta	1	size to suit
- 3 All relevant data is given in step 2 and in the Activity 1 drawing, but use your discretion for any sizes not given
- 4 Setting the grid to 10 and snap to 5 may help
- 5 Decide for yourself whether to:
 - a) set the elevation and thickness values then draw the shapes
 - b) draw the shapes with elevation 0 then change the elevation and thickness values
- 6 Decide on whether to use the CHANGE or Properties dialogue box, i.e.
 - a) PICKFIRST 0 – CHANGE at command line
 - b) PICKFIRST 1 – Properties dialogue box
- 7 When the model is complete, view at different 3D viewpoints and then hide and shade.
- 8 Note that at present you will not be able to obtain the two different views on the one screen (unless you have some prior AutoCAD 3D knowledge).
- 9 Remember to save the completed model.

The UCS and 3D coordinates

AutoCAD uses two coordinates systems:

- 1 the world coordinate system (**WCS**)
- 2 the user coordinate system (**UCS**)

The World Coordinate System (WCS)

All readers should be familiar with the basic 2D coordinate concept of a point described as P1 (30,40) – Fig. 3.1. Such a point has 30 units in the positive X-direction and 40 units in the positive Y-direction. These ordinates are relative to an XY axes system with the origin at the point (0,0). This origin is normally positioned at the lower left corner of the screen and is perfectly satisfactory for 2D draughting but not for 3D modelling.

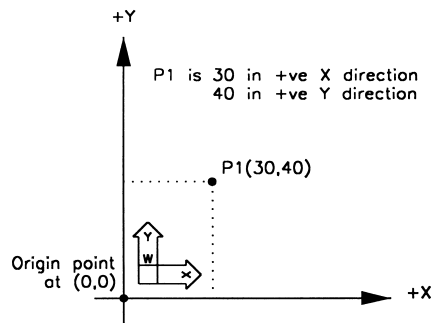


Figure 3.1 2D Coordinate entry with the WCS at the (0,0) origin.

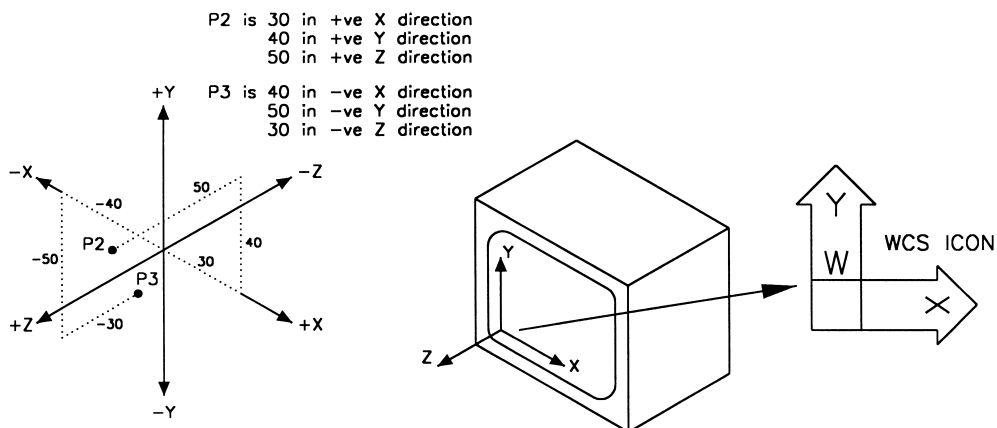


Figure 3.2 3D coordinate input.

Drawing in 3D requires a third axis (the Z axis) to enable three-dimensional coordinates to be used. The screen monitor is a flat surface and it is difficult to display a three-axis coordinate system on it. AutoCAD overcomes this difficulty by using an **ICON** and this icon can be moved to different positions on the screen and can be orientated on existing objects.

Figure 3.2 shows the basic idea of how the icon has been constructed. The X and Y axes are displayed in their correct orientations while the Z axis is pointing outwards towards the user. The **W** on the icon indicates that the user is working with the world coordinate system. The origin is at the point (0,0,0) and is positioned at the lower left corner of the screen – as it is in 2D. The status bar displays the three coordinates of any point on the screen, but these figures can be misleading, especially when viewing in 3D. The origin point can be positioned to suit the model being created – more on this later.

The point P2 (30,40,5) is thus defined as 30 units in the positive X direction, 40 units in the positive Y direction and 50 units in the positive Z direction. Similarly the point P3 (-40,-50,-30) has 40 units in the negative X direction, 50 units in the negative Y direction and 30 units in the negative Z direction.

In the previous chapter, all the extruded models were created with the WCS.

The User Coordinate System (UCS)

The UCS is one of the most important concepts in 3D modelling and all users must be fully conversant with it. The user coordinate system allows the operator to:

- a) set a new UCS origin point
- b) move the origin to any point (or object) on the screen
- c) align the UCS icon with existing objects
- d) align the UCS icon to suit any 'plane' on a model
- e) rotate the icon about the X, Y and Z axes
- f) save UCS 'positions'
- g) recall previously saved UCS settings

Icon display

AutoCAD 2002 allows the user to display the icon as a 2D symbol or as a 3D symbol. The previous discussion has assumed that the user has the traditional AutoCAD 2D icon displayed (as Fig. 3.2) but this may not be the icon displayed on your screen. To investigate the UCS icon display:

- 1 Close all existing drawings
- 2 Menu bar with **File-New** and select Start from Scratch-Metric-OK
- 3 A blank drawing screen will be returned
- 4 Menu bar with **View-Display-UCS Icon** and:
 - a) ensure On active – tick
 - b) ensure Origin active – tick
 - c) pick Properties and:

<i>prompt</i>	UCS Icon dialogue box
<i>respond</i>	1. UCS icon style: pick 2D and note Preview
	2. UCS icon size: set to suit – normally 15–20
	3. UCS icon color: set to Suit – Black is default
	4. Layout tab icon color: set to suit (Black default)
	5. pick OK
- 5 The icon is displayed as Fig. 3.3(a)

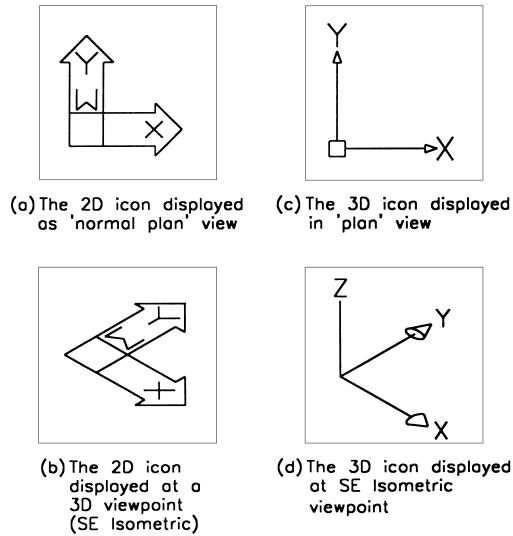


Figure 3.3 The 2D and 3D icon display.

- 6 Menu bar with **View-3D Views-SE Isometric** and the icon will be displayed in 3D as Fig. 3.3(b)
- 7 Enter **U <R>** to restore the original 'plan' icon
- 8 Repeat step 4 and from the UCS Icon dialogue box:
 - a) set UCS icon style: pick 3D and note Preview
 - b) ensure Cone active – tick
 - c) set Line width: 1
 - d) dialogue box as Fig. 3.4
 - e) pick OK
- 9 The icon will be displayed as Fig. 3.4(c) and as Fig. 3.4(d) if a SE Isometric viewpoint is set

As the user, you must now decide on whether to display the 2D or 3D icon. It is your preference.

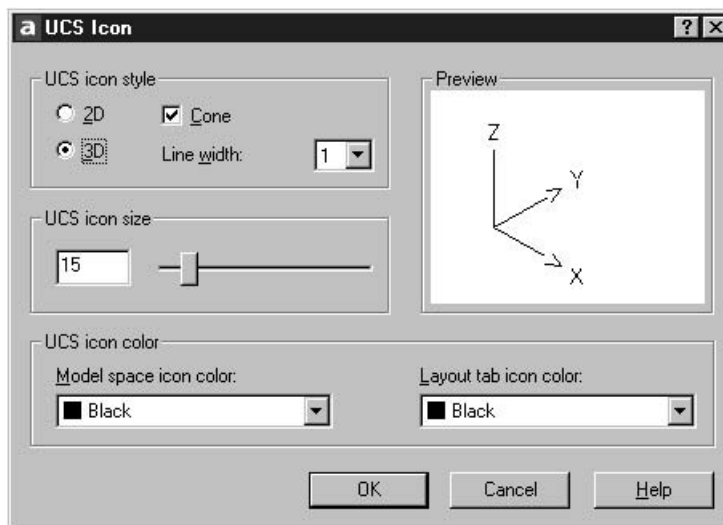


Figure 3.4 The UCS Icon dialogue box with the 3D icon set.

UCS icon exercise

The appearance of the coordinate icon alters depending on:

- a) its orientation, i.e. how it is 'attached' to objects
- b) the viewpoint selected or entered

To investigate the UCS icon display, the following exercise is given as a sequence of operations which the reader should complete. No drawing is involved and it should be noted that several of the commands will be new to some readers, all of which will be explained later. The object of the exercise is to make the reader aware of the 'versatility' of the coordinate icon.

- 1 Close all existing drawings then open your 3DSTDA3 template file. Refer to Fig. 3.5
- 2 Menu bar with **View-Display-UCS icon** and:
 - a) On and Origin both active, i.e. tick
 - b) pick Properties and activate the UCS icon style
- 3 The icon will be displayed at the lower left corner of the screen has a W on it, indicating that it is the WCS icon as fig(a). This is the 'normal' default icon.
- 4 Select the PAN icon from the Standard toolbar or enter PAN <R> at the command line and:
 - a) pan the screen upwards and to the right
 - b) right-click and pick Exit

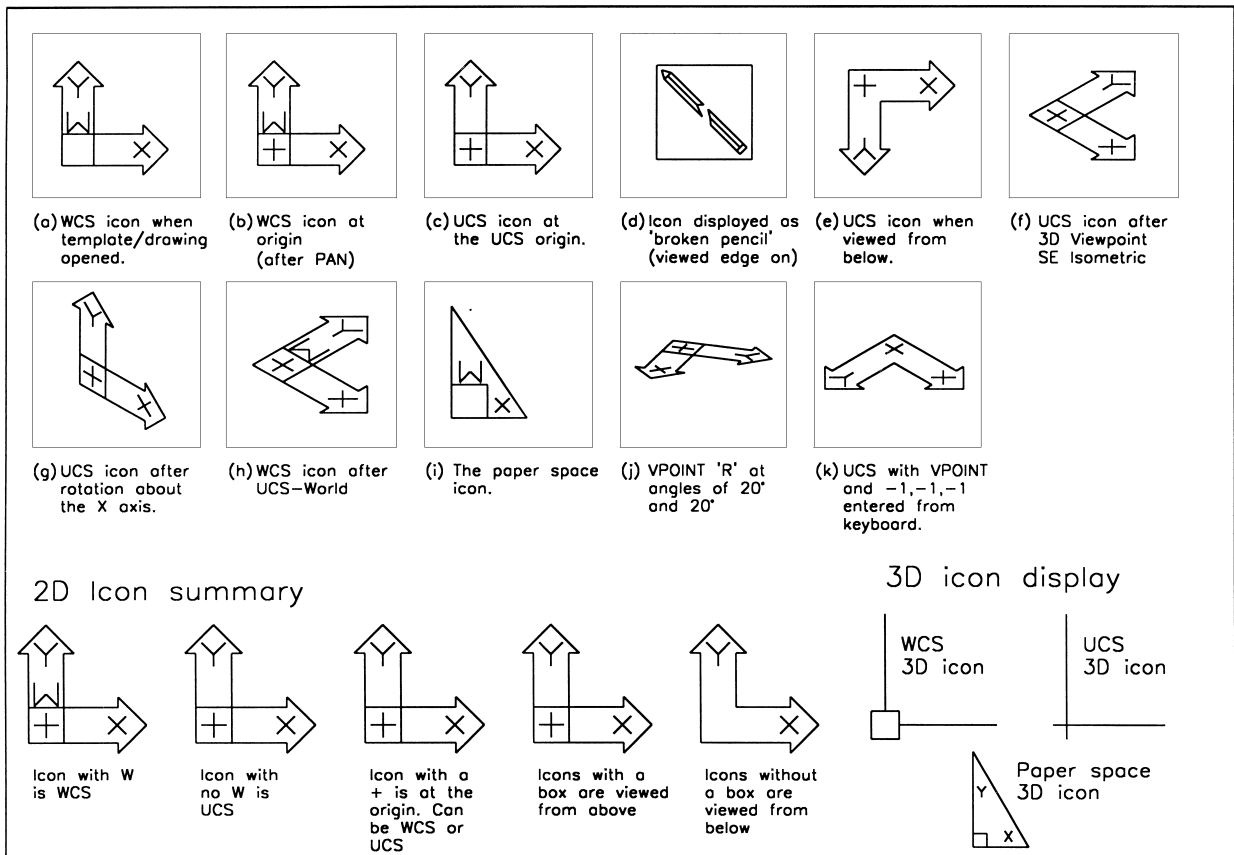


Figure 3.5 Icon exercise.

- 5 The icon will be displayed as fig(b) and be positioned at the lower left corner of the 'drawing sheet'. It has a + sign added at the 'box', indicating that the icon is positioned at the origin
- 6 With snap on, move the cursor onto the icon + and observe the status bar – the coordinates should be 0.00, 0.00, 0.00
- 7 Pick the Undo icon from the Standard toolbar to restore the icon to its original position
- 8 Menu bar with **Tools-New UCS-Origin** and:
 - prompt* Specify new origin point<0,0,0>
 - enter* **100,100 <R>**
 - and* the icon moves to the entered point and is displayed as fig(c). It has no W indicating that it is a UCS icon and has a + indicating it is at the origin
- 9 With snap on, move the cursor onto the + and observe the coordinates in the status bar. They should display 0.00
- 10 Menu bar with **Tools-New UCS-X** and:
 - prompt* Specify rotation angle about X axis<0.0>
 - enter* **90 <R>**
 - and* icon displayed as fig(d). This is the AutoCAD 'broken pencil' icon indicating that we are looking at it 'edge-on'
- 11 At the command line enter **UCS <R>** and:
 - prompt* Enter an option [New/Move/...
 - enter* **N <R>** – the new option
 - prompt* Specify origin of new UCS or [ZAxis/3point/..
 - enter* **X <R>** – the rotate about X axis option
 - prompt* Specify rotation angle about X axis
 - enter* **90 <R>**
 - and* icon displayed as fig(e) and is being viewed from below – there is no 'box'. The + is still displayed indicating the UCS icon is still at the origin
- 12 Menu bar with Tools-New UCS-X and enter 180 as the rotation angle. The icon will again be displayed as fig(c)
- 13 Menu bar with **View-3D Views-SE Isometric** and the icon will be displayed in 3D as fig(f). It still has a + and is therefore still at the origin
- 14 At the command line enter **UCS <R>** and:
 - prompt* Enter an option
 - enter* **N <R>** then **X <R>** – new and X rotate options
 - prompt* Specify rotation angle about X axis
 - enter* **90 <R>**
 - and* icon displayed as fig(g)
- 15 Undo the UCS X rotation with U <R> or pick the Undo icon to display the icon as fig(f) again
- 16 At the command line enter **ZOOM <R>** then **0.75 <R>** to 'reduce' the scale of the drawing sheet
- 17 Menu bar with **Tools-New UCS-World** and the icon will be displayed as fig(h). This is a WCS icon positioned at the original origin point – the lower left corner of the 'drawing sheet'. The icon is still displayed in 3D
- 18 Menu bar with **View-3D Views-Plan View-World UCS** and the icon should be as the original fig(a). The screen should display the drawing sheet 'as opened'

- 19 Left click on the word MODEL in the status bar and:
prompt Page Setup dialogue box
respond **pick Cancel** – more on this later
and the icon will be displayed as fig(i). This is the paper space icon which will be discussed in more detail in a later chapter.
- 20 At present undo this paper space effect with U <R> to restore the icon as fig(a)
- 21 Enter/select the following sequences:
 a) Menu bar with Tools-New UCS-Origin and enter 100,100 to display the icon as fig(c)
 b) enter VPOINT <R> then R <R> with angles of 20 and 20 and the icon will be displayed as fig(j)
 c) enter VPOINT <R> then -1,-1,-1 to give the icon as fig(k)
 d) enter VPOINT <R> then R <R> with angles of 0 and 90 – fig(c)
 e) enter UCS <R> then W <R> – fig(a)
- 22 This completes the first part of the icon exercise.
- 23 *Note:* we could have used the UCS toolbar with icons during this exercise, but at this stage I think that the menu bar and command line selections give the user a ‘better understanding’ of that is actually happening.
 You can investigate the UCS toolbar for yourself.
- 24 *Icon summary*
 Figure 3.5 displays a summary of the various 2D icons which can be displayed on the screen. These are:
 a) icon with a W is a WCS icon
 b) icon with no W is a UCS icon
 c) icon with a + is at the origin
 d) icon with a ‘box’ is viewed from above
 e) icon with no ‘box’ is viewed from below
- 25 *Task*
 a) with the UCS Icon dialogue box, set a 3D style icon
 b) repeat the steps in the previous exercise and observe the orientation of the 3D icon
 c) generally the same ‘type of orientation’ is obtained with the 3D icon as with the 2D icon. The paper space icon with the 3D style is slightly different from the 2D icon
 d) the WCS and UCS icons with a 3D style setting are displayed in Fig. 3.5
 e) it is user-preference whether to use a 2D or 3D icon
- 26 This exercise is now complete

Orientation of the UCS

The completed exercise has demonstrated that the UCS icon can be moved to any point on the screen and rotated about the three axes (we only used the X axis rotation, but the procedure is the same for the Y and Z axes). It is thus important for the user to be able to determine the correct orientation of the icon, i.e. how the X, Y and Z axes are configured in relation to each other.

The axes orientation is determined by the **right-hand rule** and is demonstrated in Fig. 3.6. The knuckle of the right hand is at the origin and the position of the thumb, index finger and second finger determine the direction of the positive X, Y and Z axes respectively.

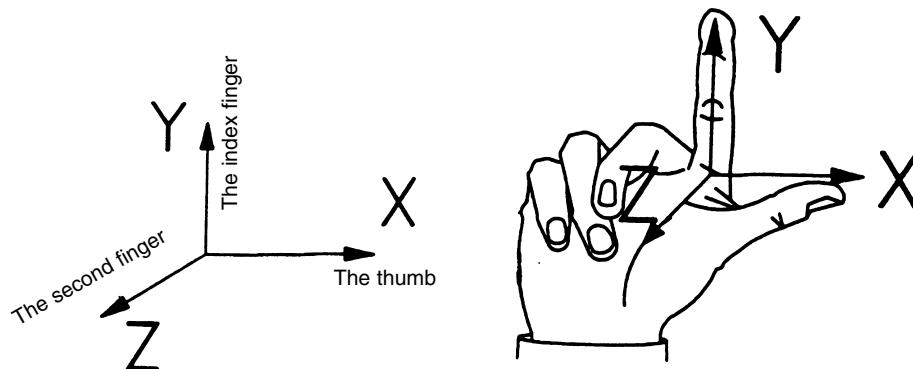


Figure 3.6 The right-hand rule.

Three-dimensional coordinate input

Coordinate input is generally required at some time during the creation of a 3D model. With 3D draughting there are three types of coordinates available, each having both absolute and relative entry modes. The three coordinate types with their formats and examples are:

Type	Format	Absolute	Relative
Cartesian	x dist,y dist,z dist	100,150,120	@300,-100,-50
Cylindrical	dist<angle,Z dist	150<55,120	@75<-15,-120
Spherical	dist<angle 1<angle 2	80<30<50	@120<-10<75

To investigate the different types of coordinate input we will draw some objects on the screen. We will also investigate the effect of the icon position on the coordinate entries.

- 1 Close any existing drawings
- 2 Open your 3DSTDA3 template file and refer to Fig. 3.7
- 3 Menu bar with **View-Display-UCS Icon** and:
 - a) On and Origin – tick
 - b) properties and set a 2D style
 - c) These selections ensure that the icon is displayed in 2D on the screen and is always ‘positioned’ at the origin point.
- 4 Menu bar with **View-3D Views-SE Isometric** to display the screen in 3D
- 5 Menu bar with **View-Zoom-Scale** and:

prompt Enter a scale factor and enter: **0.75 <R>**
- 6 The WCS icon should be positioned at the left vertex of the black border – point A in Fig. 3.7
- 7 Make three new layers – L1, L2, L3 with continuous linetype and colour numbers 30, 72, 240 respectively

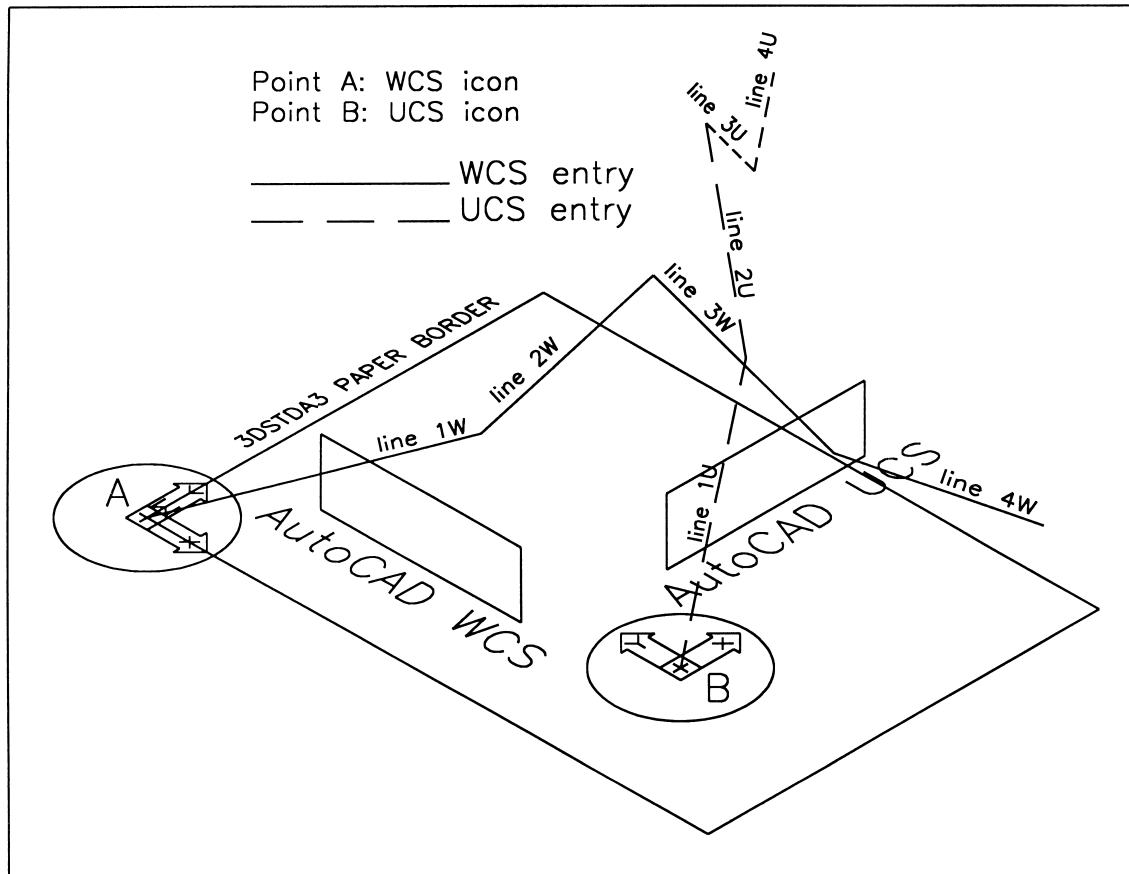


Figure 3.7 Coordinate entry exercise.

A) WCS entry

- 1 With layer L1 current, use the LINE icon and draw:

<i>First point</i>	0,0,0 <R>		
<i>Next point</i>	150,100,80 <R>	absolute	line 1W
<i>Next point</i>	@50,80,90 <R>	relative absolute	line 2W
<i>Next point</i>	@100<30,-100 <R>	relative cylindrical	line 3W
<i>Next point</i>	@120<40<-20 <R>	relative spherical	line 4W
<i>Next point</i>	right-click and pick Enter		

- 2 Draw a circle, centre: 0,0,0 with radius: 50
- 3 Add the following item of text:
 - a) start point: 40,40,0
 - b) height: 20 with 0 rotation
 - c) item: AutoCAD WCS
- 4
 - a) With ELEV at the command line, set the current elevation to 0 and the current thickness to 50
 - b) Draw a line from 60,70 to @150,0,0
- 5 Set the elevation and thickness values back to 0

B) UCS entry

- 1 Menu bar with **Tools-New UCS-Origin** and:
prompt Specify new origin point
enter **300,100,0 <R>**
- 2 Menu bar with **Tools-New UCS-Z** and:
prompt Specify rotation angle about Z axis
enter **90 <R>**
- 3 The icon should be positioned and orientated at point B
- 4 With layer L2 current, use the LINE icon and draw:
First point **0,0,0 <R>**
Next point **150,100,80 <R>** absolute line 1U
Next point **@50,80,90 <R>** relative absolute line 2U
Next point **@100<30,-100 <R>** relative cylindrical line 3U
Next point **@120<40<-10 <R>** relative spherical line 4U
Next point **right-click and Enter**
- 5 Menu bar with **View-Zoom-All** to 'see' the additional lines
- 6 Draw a circle, centred on 0,0,0 with a 50 radius
- 7 Add the text item:
a) start point: 40,40,0
b) height: 20 with 0 rotation
c) item: AutoCAD UCS
- 8 a) With ELEV at the command line, set the current elevation to 0 and the current thickness to 50
b) Draw a line from 60,70 to @150,0,0
- 9 Set the elevation and thickness values back to 0

C) WCS entry with UCS icon

- 1 The UCS should still be at position B
- 2 Set elevation and thickness to 0 and make layer L3 current
- 3 With the LINE icon draw:
First point ***0,0,0 <R>**
Next point ***150,100,80 <R>**
Next point **@*50,80,90 <R>**
Next point **@*100<30,-100 <R>**
Next point **@*120<40<-20 <R>**
next point **right-click and Enter**
- 4 These lines should be identical to those created on layer L1 when the WCS was current

Task

- 1 Save the coordinate exercise if required, but we will not refer to it again
- 2 With **File-Open** recall your 3DSTDA3 template file
- 3 Menu bar with **View-Display-UCS icon** and ensure:
 - a) On and origin active – tick
 - b) Properties and set icon to your preference 2D or 3D
- 4 Menu bar with **View-3D Views-SE Isometric**
- 5 Menu bar with **View-Zoom-Scale** and enter a factor of **0.75**
- 6 The WCS icon should be positioned at left vertex of the border
- 7 Save this layout as:
 - a) the **3DSTDA3.dwt** template file and replace the existing template file. Enter a suitable template description
 - b) the **3DSTDA3.dwg** drawing file, overwriting the existing file
- 8 This will allow the template file to be opened in 3D with the icon always 'set' to the origin position.

Summary

- 1 There are two coordinate systems:
 - a) the world coordinate system WCS
 - b) the user coordinate system UCS
- 2 Each system has its own icon
- 3 The WCS is a fixed system, the origin being at 0,0,0
- 4 The WCS icon is 'standard' and does not alter in appearance. The WCS icon is denoted with the letter W
- 5 The UCS system allows the user to define the origin, either as a point on the screen or referenced to an existing object
- 6 The UCS icon alters in appearance dependent on the viewpoint
- 7 The UCS icon can be rotated about the three axes
- 8 The UCS current position can be saved and recalled
- 9 The user can set a 2D or 3D UCS icon style
- 10 3D coordinate input can be:
 - a) Cartesian, e.g. 10,20,30
 - b) Cylindrical, e.g. 10<20,30
 - c) Spherical, e.g. 10<20<30
- 11 Both absolute and relative modes of input are possible with the three 'types' of coordinates, e.g.
 - a) absolute cylindrical: 100<200,50
 - b) relative cylindrical: @100<200,50
- 12 3D coordinate input can be relative to the current UCS position or to the WCS, e.g.
 - a) 100,200,150 for UCS entry
 - b) *100,200,150 for WCS entry
- 13 It is recommended that **3D coordinate input is relative to the current UCS position.**

Creating a 3D wire-frame model

In this chapter we will create a 3D wire-frame model and use it to:

- investigate how the UCS can be set and saved
- add 'objects' and text to the model 'planes'
- modify the model

Getting started

- Open your 3DSTDA3 template file to display:
 - a 3D viewpoint with the black border
 - the WCS icon at the left vertex of the border
- Ensure layer MODEL is current and refer to Fig. 4.1
- Display the Draw, Modify and Objects Snap toolbars

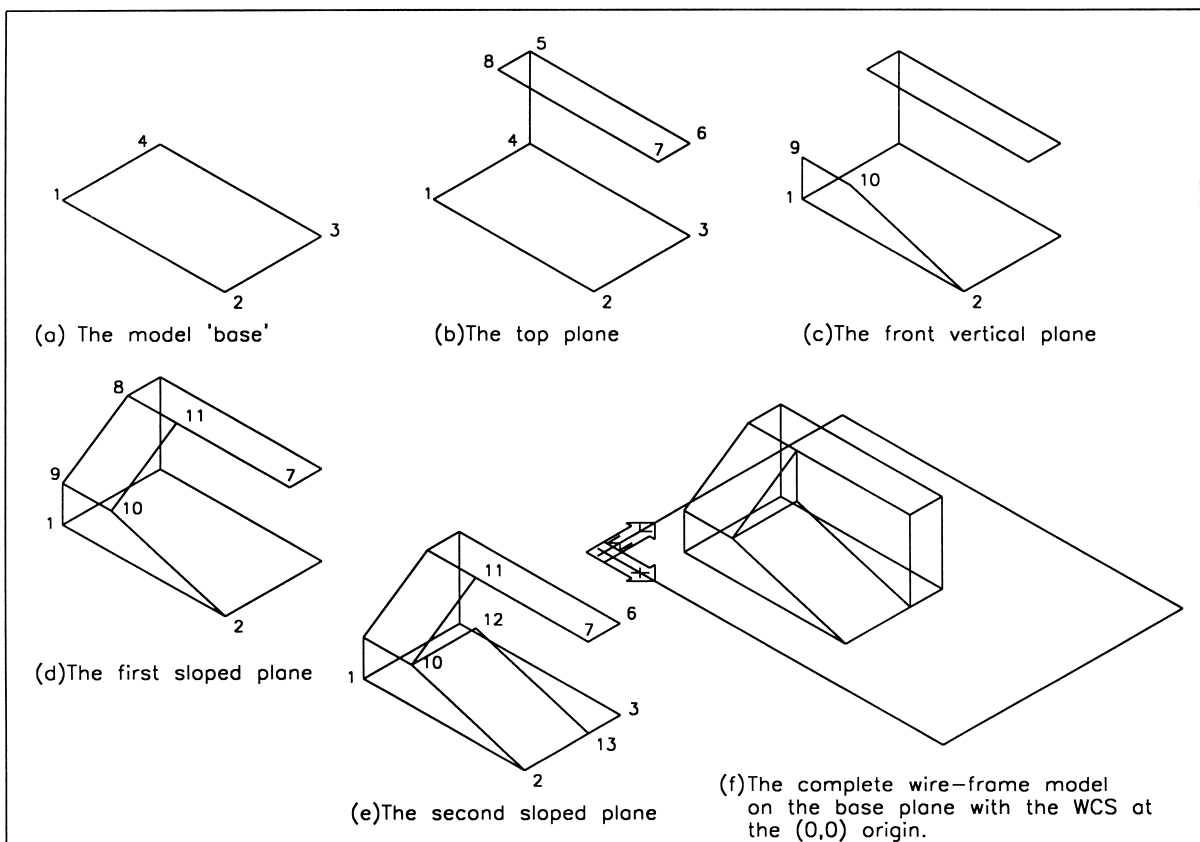


Figure 4.1 Construction of the wire-frame model 3DWFV.

Creating the wire-frame model

- 1 To create the base of the model – fig(a), select the LINE icon and draw:

<i>Start point</i>	50,50 <R>	pt1
<i>Next point</i>	@200,0 <R>	pt2
<i>Next point</i>	@0,120 <R>	pt3
<i>Next point</i>	@-200,0 <R>	pt4
<i>Next point</i>	close	

- 2 The top plane – fig(b) is also created from lines, so with the LINE icon draw:

<i>Start point</i>	Intersection icon of pt4	
<i>Next point</i>	@0,0,100 <R>	pt5
<i>Next point</i>	@200,0,0 <R>	pt6
<i>Next point</i>	@0,-40,0 <R>	pt7
<i>Next point</i>	@-200,0,0 <R>	pt8
<i>Next point</i>	Intersection icon of pt5	pt5
<i>Next point</i>	right-click and pick Enter	

- 3 If you cannot ‘see’ the complete model, then menu bar with **View-Zoom-Scale** and enter a scale factor of 0.9

- 4 To create the front vertical plane – fig(c), select the LINE icon and draw:

<i>Start point</i>	Intersection icon of pt1	
<i>Next point</i>	@0,0,45 <R>	pt9
<i>Next point</i>	@60,0,0 <R>	pt10
<i>Next point</i>	Intersection icon of pt2	
<i>Next point</i>	right-click and Enter	

- 5 With the LINE icon draw:

<i>Start point</i>	Intersection of pt9	
<i>Next point</i>	Intersection of pt8 then right-click/enter	

- 6 LINE icon again:

<i>Start point</i>	Intersection of pt10	
<i>Next point</i>	Perpendicular to line 78	pt11
<i>Next point</i>	right-click and Enter	
<i>and</i>	first sloped plane created – fig(d)	

- 7 To create the second sloped plane – fig(e), select the LINE icon and draw:

<i>Start point</i>	Intersection of pt10	
<i>Next point</i>	@0,80,0 <R>	pt12
<i>Next point</i>	Perpendicular to line 23	pt13
<i>Next point</i>	right-click and Enter – fig(e)	

- 8 To completing the model, three lines require to be added, so with the LINE icon draw:
 - a) from pt3 to pt6
 - b) from pt7 to pt13
 - c) from pt11 to pt12

- 9 The completed model is displayed in fig(f) on ‘its base’, i.e. the black border.

- 10 At this stage save the model as a drawing file with the name **C:\MODR2002\3DWFM**

- 11 *Note*
 The model has been created using 3D coordinate input with the WCS, i.e. no attempt has been made to use the UCS. This is a perfectly valid method of creating wire-frame models, but difficulty can be experienced if objects and text have to be added to the various ‘surfaces’ of the model when the coordinates need to be calculated. Using the UCS usually overcomes this type of problem.

Moving around with the UCS

To obtain a better understanding of the UCS and how it is used with 3D models, we will use the created wire-frame model to add some objects and text. The sequence is quite long but it is important that you persevere and complete the exercise. Both menu bar and keyboard entry methods will be used to activate the UCS command.

- 1 Open the wire-frame model **C:\MODR2000\3DWFM** or continue from the previous exercise. This model has the WCS icon at the black border origin point – the left vertex
- 2 Menu bar with **View-Display-UCS Icon** and:
 - a) On and Origin both active (tick)
 - b) select Properties and set a 2D UCS icon style
- 3 Refer to Fig. 4.2
- 4 PAN the layout until the lower black border vertex is near the lower edge of the screen. This will allow us to 'see' any UCS movements more clearly
- 5 Menu bar with **Tools-New UCS-Origin** and:

prompt Specify new origin point<0,0,0>
respond **Intersection icon and pick pt1**
and a) icon 'moves' to selected point – fig(a)
 b) it is a UCS icon – no W
 c) it is at the origin – the +
note if the icon does not move to the selected point, menu bar with **View-Display-UCS Icon** and pick Origin

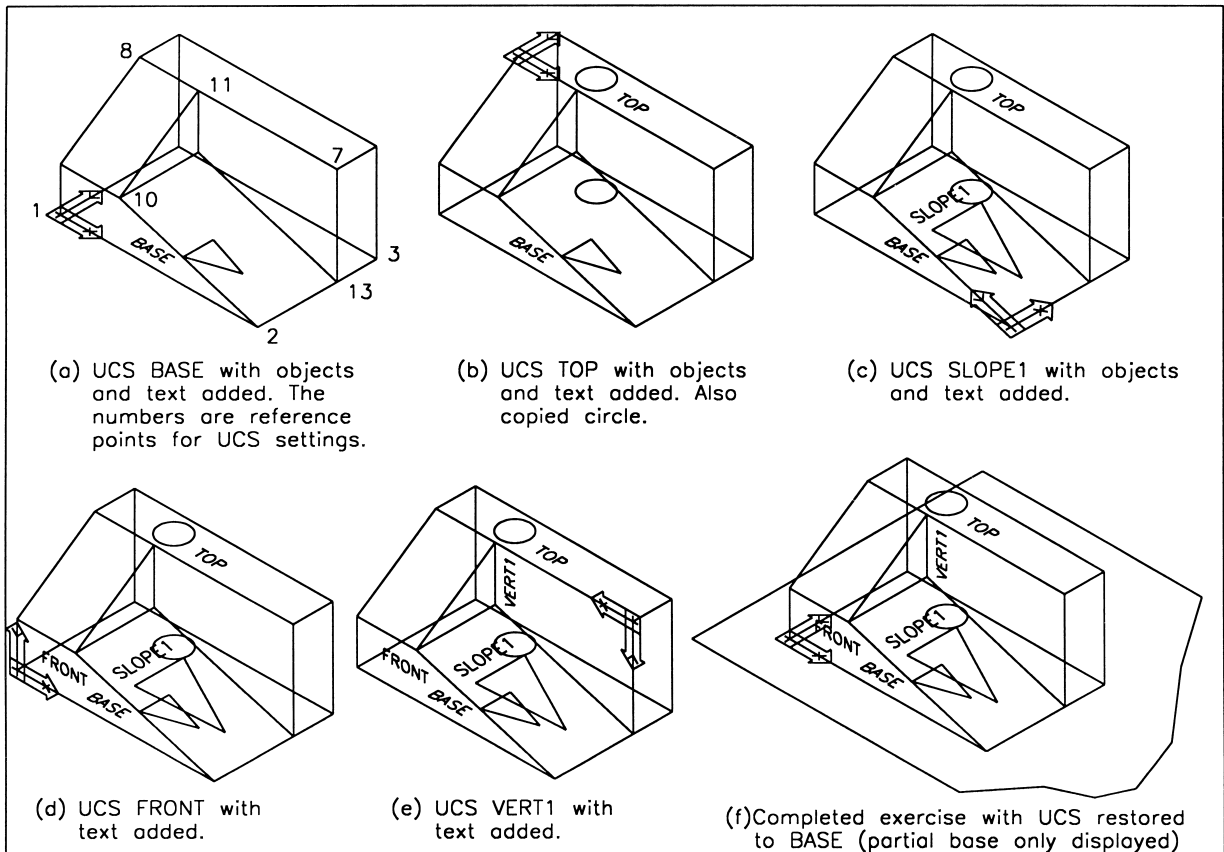


Figure 4.2 Investigating the UCS and adding objects and text to 3DWFV.

- 6 Now that the icon has been repositioned at point 1, we want to save its 'position' for future recall, so at the command line enter **UCS <R>** and:
- ```

prompt Enter an option
enter S <R> – the save option
prompt Enter name to save current UCS
enter BASE <R>

```
- 7 Make layer OBJECTS current and use the LINE icon to draw:
- ```

Start point 100,25,0 <R>
Next point  @0,30,0 <R>
Next point  145,40,0 <R>
Next point  close

```
- 8 Make layer TEXT current and menu bar with **Draw-Text-Single Line Text** and:
- start point: 60,10,0
 - height: 10 and 0 rotation
 - text item: BASE
- 9 The line objects and text item are added as fig(a)
- 10 Menu bar with **Tools-New UCS-Origin** and:
- ```

prompt Specify new origin point<0,0,0>
respond Intersection icon and pick pt8
and icon 'jumps' to the selected point – fig(b)

```
- 11 At the command line enter **UCS <R>** and:
- ```

prompt      Enter an option
enter       S <R> – the save option
prompt      Enter name to save current UCS
enter       TOP <R>

```
- 12 With layer OBJECTS current draw a circle with centre: 60,20 and radius: 15
- 13 With layer TEXT current, add single line text using:
- start point: 85,10
 - height: 10 with 0 rotation
 - text item: TOP
- 14 Using the COPY icon:
- select objects: pick the circle then right-click
 - base point: Center icon and pick the circle
 - second point: enter **@0,0,-100 <R>** – fig(b)
 - question: why these coordinates?
- 15 Menu bar with **Tools-UCS-3Point** and:
- ```

prompt Specify new origin point<0,0,0>
respond Intersection icon and pick pt2
prompt Specify point on positive portion of the X-axis
respond Intersection icon and pick pt3
prompt Specify point on positive-Y portion of the UCS XY plane
respond Intersection icon and pick pt10

```
- 16 The UCS icon will move to point 2 and be 'aligned' on the sloped surface as fig(c)
- 17 *Note*  
The 3 point option of the UCS command is 'asking the user' for three points to define the UCS icon orientation, these being:
- ```

first prompt  the origin point
second prompt the direction of the X axis
third prompt  the direction of the Y axis

```

- 18 Save this UCS position by entering at the command line **UCS <R>** then **S <R>** and:
prompt Enter name to save current UCS
enter **SLOPE1 <R>**
- 19 With layer OBJECTS current use the LINE icon to draw:
Start **15,100,0**
Next **@50,0,0**
Next **40,30,0**
Next **close**
- 20 With layer TEXT current, add a single text item using:
 a) start point: centred on 10,110
 b) height: 10 with 0 rotation
 c) item: SLOPE1 – fig(c)
- 21 At command line enter **UCS <R>** and:
prompt Enter an option
enter **R <R>** – the restore option
prompt Enter name of UCS to restore
enter **BASE <R>**
and icon restored to the base point as fig(a)
 (The restore option is used extensively with UCS's)
- 22 Menu bar with **Tools-New UCS-X** and:
prompt Specify rotation angle about X axis
enter **90 <R>**
and icon displayed as fig(d)
- 23 At command line enter **UCS <R>** then **S <R>** for the save option and **FRONT <R>** as the UCS name to save
- 24 With layer TEXT current add an item of text with:
 a) start point: 25,20
 b) height: 10 with 0 rotation
 c) text: FRONT – fig(d)
- 25 Menu bar with Tools-New UCS-3 Point and:
prompt Specify new origin point
respond **Intersection icon and pick pt7**
prompt Specify point on positive portion of the X-axis
respond **Intersection and pick pt11**
prompt Specify point on positive-Y portion of the UCS XY plane
respond **Intersection icon and pick pt13**
- 26 The UCS icon will be aligned as fig(e)
- 27 Save this UCS position as VERT1 – easy? (UCS-S-VERT1)
- 28 With layer TEXT current add a text item with:
 a) start point: 120,50
 b) height: 10
 c) rotation: –90
 d) text: VERT1 – fig(e)
- 29 Restore UCS BASE and the model will be displayed as fig(f)
- 30 Make layer MODEL current and save the drawing at this stage as **C:\MODR2000\3DWF** updating the original wire-frame model.

Modifying the wire-frame model

To further investigate the UCS we will modify the wire-frame model, so refer to Fig. 4.3 and:

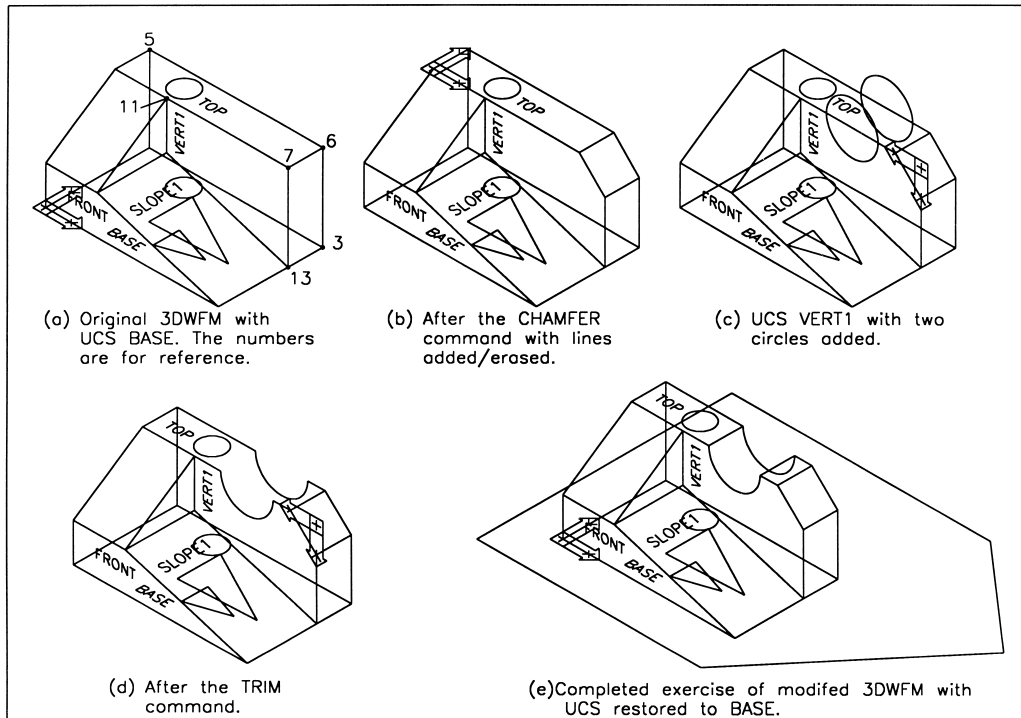


Figure 4.3 Modifying the 3DWF model.

- 1 3DWF model still on the screen? – if not open the drawing file
- 2 Layer MODEL current with UCS BASE – fig(a)
- 3 Select the CHAMFER icon from the Modify toolbar and:
 - a) set both chamfer distances to 30
 - b) chamfer lines 7–11 and 7–13
 - c) chamfer lines 5–6 and 6–3
- 4 Now add two lines to complete the ‘chamfered corner’ and erase the unwanted original corner line – fig(b).
- 5 Restore UCS VERT1 and note its position – fig(c)
- 6 Draw two circles:
 - a) centre at 80,0,0 with radius 30
 - b) centre at 80,0,-40 with radius 30 – fig(c)
- 7 Using the TRIM icon from the Modify toolbar:
 - a) trim the two circles ‘above’ the model
 - b) trim the two lines ‘between’ the circles – fig(d)
- 8 Move the TOP text item from: ENDPOINT of pt5, by: @80,0
- 9 Draw in the two lines on the top plane and restore UCS BASE.
- 10 The modified model is now complete – fig(e)
- 11 Save the model as **C:\MODR2000\3DWF** updating the existing model drawing
- 12 *Note*

The user should realise that the UCS is an important concept with 3D modelling. Indeed I would suggest that 3D modelling would be very difficult (if not impossible) without it.

Task 1

- 1 The wire-frame model has eleven flat planes and one 'curved surface'. We have set and saved UCS positions for five of these planes – BASE, TOP, SLOPE1, FRONT and VERT1.
- 2 You now have to set and save the other six flat positions, i.e. one for each surface and add an appropriate text item to that surface.
- 3 My suggestions for the UCS name and text item are LEFT, RIGHT, REAR, SLOPE2, SLOPE3 and VERT2 but you can use any names that you consider suitable.
- 4 Figure 4.4 displays the complete wire-frame model with text added to every plane (with the exception of the curved surface) using the UCS positions I 'set'. Realise that your additional text may differ in appearance from mine. This is acceptable as your UCS positions may be 'set' different from mine
- 5 When complete, remember to save as **MODR2002\3DWFM** as it will be used in other chapters.

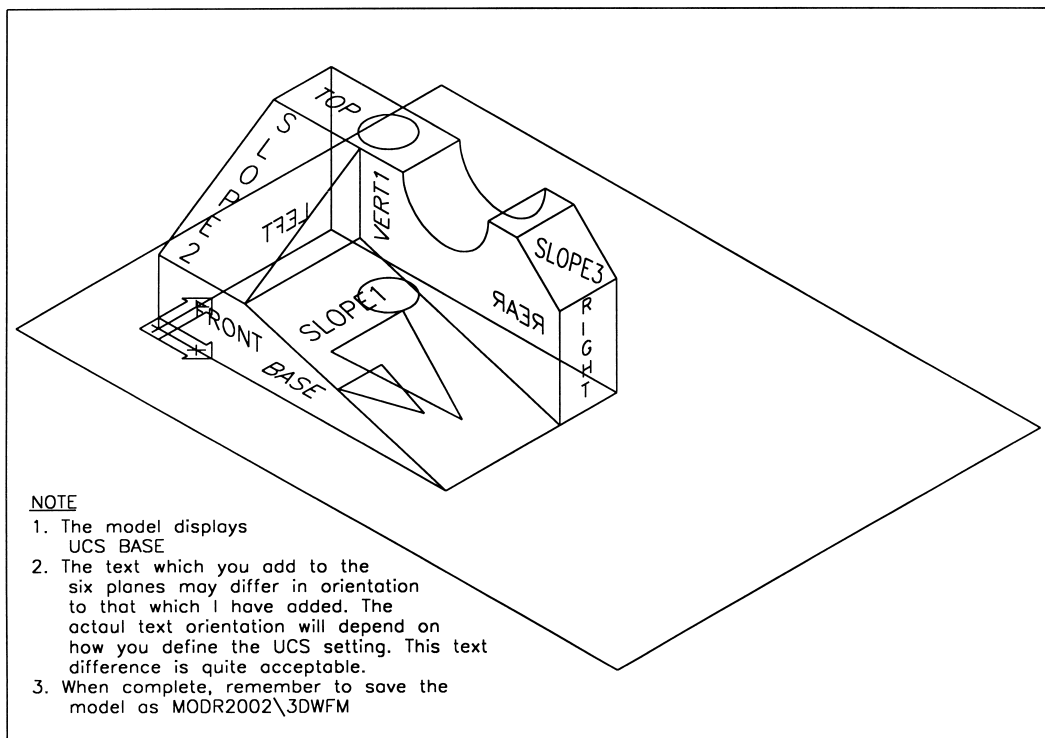


Figure 4.4 The complete 3D wire-frame model (3DWFM) with text added to every plane.

Task 2

- 1 Restore UCS BASE – should be current?
- 2 With the MOVE command:
 - a) window the complete model then right-click
 - b) base point: 0,0
 - c) second point: @100,100
- 3 The complete model moves as expected, but do the set UCS's move with the model? This can be a nuisance when moving models. The UCS is 'not tied' to a specific model, it is **ONLY A POSITION ON THE SCREEN**
- 4 This exercise is now complete. Do not save the changes.

Summary

- 1 Wire-frame models are created by coordinate input and by referencing existing objects
- 2 Both the WCS and UCS entry modes can be used, but I would recommend:
 - a) use the WCS to create the basic model outline
 - b) use the UCS to modify and add items to the model
- 3 It is strongly recommended that a UCS be set and saved for every surface on a wire-frame model.

Assignments

Creating wire-frame models at this stage is important as it allows the user to:

- a) use 3D coordinate entry with the WCS and/or the UCS
- b) set and save different UCS positions
- c) become familiar with the concept of 3D modelling

I have included two 3D wire-frame models which have to be created. The suggested approach is:

- 1 Open your 3DSTDA3 standard file – template or drawing
- 2 Complete the model with layer MODEL current, starting at some convenient point, e.g. 50,50,0. Use WCS entry and add one ‘plane’ at a time
- 3 Save each completed model as a drawing file in your named folder with a suitable name, e.g. C:\MODR2002\ACT2, etc.
- 4 *Note:*
 - a) **do not attempt to add dimensions**
 - b) do not attempt to display the two models on ‘one screen’ – you will soon be able to achieve this for yourself.
 - c) these models will be used for later assignments, so ensure they are saved
 - d) use your discretion for any sizes not given

The activities concern our master builder MACFARAMUS, and you have to create 3D wire-frame models of two of his famous shaped blocks. It is not known how these blocks were used, i.e. in road building, structures, plazas etc, but they allow us to create wire-frame models.

Activity 2: MACFARAMUS’s shaped block 1

A relatively simple wire-frame model to create. I suggest that you construct it in a similar manner to the worked example, i.e. create the base, then the front vertical plane. The ‘back’ vertical plane can be drawn or copied from the front plane. The top and slopes are then easy to complete. When finished, save as **MODR2002\ACT2**

Activity 3: MACFARAMUS’s shaped block 2

This shaped block is slightly more difficult due to the curves. Create the basic shape as two rectangular blocks, then add four circles, using an obvious ‘corner point’ as the circle centre. The circles and lines can then be trimmed ‘to each other’, but the UCS position is important. When complete, save as **MODR2002\ACT3**

The UCS

The UCS is one of the basic 3D draughting 'tools' and it has several commands associated with it. Although it was used in the previous chapter, we will now investigate in more detail:

- a) setting a new UCS position
- b) moving the UCS
- c) the UCS and UCS II toolbars
- d) the UCS dialogue box
- e) Orthographic UCSs
- f) UCS specific commands

Getting started

- 1 Open your MODR2002\3DWFWM model from the previous chapter. This model has several blue objects with several saved UCS positions and is 'positioned' on the black 'sheet border'
- 2 Restore the UCS BASE – probably is current?
- 3 Layer MODEL current and freeze layer TEXT. Refer to Fig. 5.1 which does not display the black sheet border. This is for clarity only.

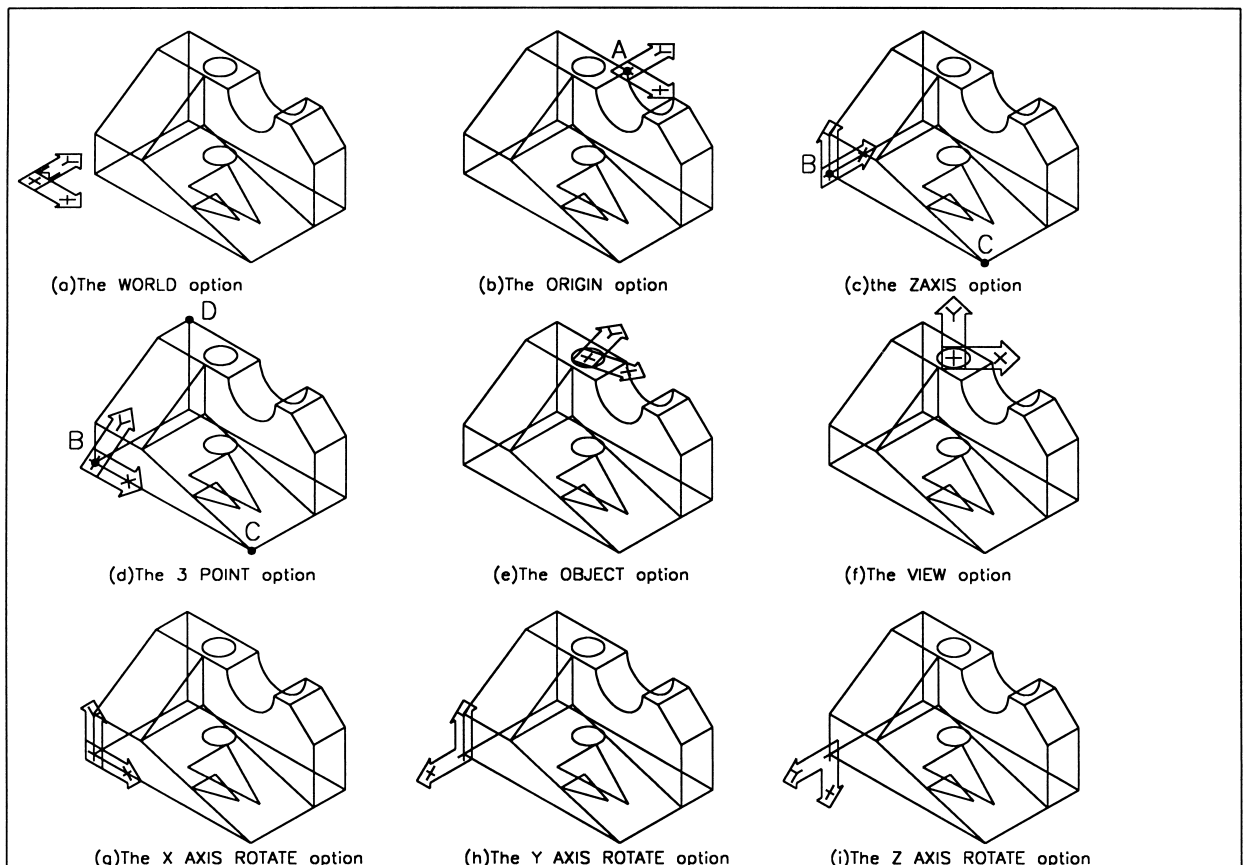


Figure 5.1 The UCS (NEW) options exercise.

Setting a new UCS position

The user can set a new UCS position from the menu bar with **Tools-New UCS** or by entering **UCS <R>** then **N <R>** at the command line. Both methods give the user access to the same options although the selection order differs. The menu bar options are displayed as:

World/Object/Face/View/Origin/ZAxis Vector/3 Point/X/Y/Z

The following is an explanation of these UCS option:

World

- 1 This option restores the WCS setting irrespective of the current UCS position. It is the default AutoCAD setting.
- 2 At the command line enter **UCS <R>** then **W <R>** to display the WCS icon on the sheet border at the left vertex as fig(a)

Origin

- 1 Used to set a new origin point. The user specifies this new origin point by:
 - a) picking any point on the screen
 - b) coordinate entry
 - c) referencing existing objects
- 2 When used, the UCS icon is positioned at the selected point if the UCS Icon display is set to Origin. This option has been used in previous exercises.
- 3 Menu bar with **Tools-New UCS-Origin** and:

prompt Specify new origin point<0,0,0>
respond **Intersection icon and pick ptA**
and icon positioned as fig(b)

Z Axis Vector

- 1 Defines the UCS position relative to the Z axis, the user specifying:
 - a) the origin point
 - b) any point on the Z axis
- 2 Menu bar with **Tools-New UCS-Z Axis Vector** and:

prompt Specify new origin point
respond **Intersection icon and pick ptB**
prompt Specify point on positive portion of Z-axis
respond **Intersection icon and pick ptC** – fig(c)
- 3 The icon will be aligned with:
 - a) the X axis along the shorter base left edge
 - b) the Y axis along the front left vertical edge
 - c) the Z axis along the line BC

3 Point

- 1 Defines the UCS orientation by specifying three points:
 - a) the actual origin point
 - b) a point on the positive X axis
 - c) a point on the positive Y axis
- 2 Menu bar with **Tools-New UCS-3 Point** and:

prompt Specify new origin point

respond **Intersection of ptB**

prompt Specify point on positive portion of the X-axis

respond **Intersection of ptC**

prompt Specify point on positive-Y portion of the UCS XY plane

respond **Intersection of ptD** – icon as fig(d)
- 3 This is a very useful option especially if the icon is to be aligned on sloped surfaces. It is probably my preferred method of setting the UCS.

Object

- 1 Aligns the icon to an object, e.g. a line, circle, polyline, item of text, dimension, block etc.
- 2 Menu bar with **Tools-New UCS-Object** and:

prompt Select object to align UCS

respond **pick any point on circle on top surface**
- 3 The icon is aligned as fig(e) with:
 - a) the origin at the circle centre point
 - b) the positive X axis pointing towards the circumference of the circle at the point 'picked' by the user

View

- 1 Aligns the UCS so that the XY plane is always perpendicular to the view plane.
- 2 Menu bar with **Tools-New UCS-View**
- 3 The UCS icon will be displayed as fig(f) and is similar to the traditional 2D icon?
- 4 This is a useful UCS option as it allows 2D text to be added to a 3D drawing – try it for yourself.

X/Y/Z

- 1 Allows the UCS to be rotated about the entered axis by an amount specified by the user
- 2 Make UCS BASE current
- 3 Menu bar with **Tools-New UCS-X** and:

prompt Specify rotation angle about the X axis

enter **90 <R>** – fig(g)
- 4 Menu bar with **Tools-New UCS-Y** and:

prompt Specify rotation angle about the Y axis

enter **-90 <R>** – fig(h)
- 5 Menu bar with **Tools-New UCS-Z** and:

prompt Specify rotation angle about the Z axis

enter **-90 <R>** – fig(i)

Face

- 1 Aligns the UCS with a selected solid model face. This option *cannot be used with 3D wire-frame models*.
- 2 Restore UCS BASE
- 3 Menu bar with **Tools-New UCS-Face** and:
 - prompt* Select face of solid object
 - respond* **pick any line of the top plane**
 - prompt* A 3D solid must be selected
 - No solids detected

Apply

An option which allows the user to apply the current UCS setting to a specific viewport. We will use this option in later chapters, but not yet.

Moving a UCS

A selection which allows the user to move the UCS to a new origin position, the UCS icon retaining both its orientation and name. Refer to Fig. 5.1 and:

- 1 UCS restored to BASE
- 2 Menu bar with **Tools-Move UCS** and:
 - prompt* Specify new origin point or [Zdepth]
 - respond* **Intersection icon and pick ptA**
 - and* icon moved to point A and retains the name BASE
- 3 Restore UCS TOP
- 4 At the command line enter **UCS <R>** and:
 - prompt* Enter an option [New/Move/..
 - enter* **M <R>** – the move option
 - prompt* Specify new origin point or [Zdepth]
 - respond* **Intersection icon and pick ptC**
 - and* icon moves to point C and retains the name TOP
- 5 Restore UCS FRONT
- 6 Menu bar with **Tools-Move UCS** and:
 - prompt* Specify new origin point or [Zdepth]
 - enter* **Z <R>** – the Z depth option
 - prompt* Specify Zdepth<0>
 - enter* **-120 <R>**
 - and* icon moved to ‘back of model’ and retains name FRONT
- 7 *Note:*
 - a) This UCS command should be used with caution as the user may not want a named UCS to be ‘repositioned’
 - b) I never use this command. If I want to reposition the UCS, I use the origin option
 - c) Do not save the drawing, as you will saved these moved UCS’s
- 8 Task
 - Reset the three moved UCS’s to their original positions, i.e. BASE, TOP and FRONT

Other UCS options

The new UCS options are available from the command line but the menu bar selection Tools-New UCS is the usual method of activating the command. The command line has other UCS options available for selection, these being:

Prev

- 1 Restores the previously 'set' UCS position and can be used to restore the last 10 UCS positions.
- 2 The command is activated from the command line by entering **UCS <R>** then **P <R>** and can be used continually until the command line displays '*no previous coordinate system saved*'.

Restore

- 1 Allows the user to restore a previously saved UCS position but the names of the saved UCS's must be remembered (This will be modified shortly). This option has been used in our examples
- 2 At the command line enter **UCS <R>** then **R <R>** and:
prompt Enter name of UCS to restore or [?]
enter **TOP <R>**
then restore UCS BASE

Save

- 1 Allows the user to save a UCS position for future recall. It should be used every time a new UCS has been defined.
- 2 The option is activated from the command line with **UCS <R>** then **S <R>** and the user can enter any name for the UCS position.

Del

- 1 Entering **UCS <R>** then **D <R>** prompts for the UCS name to be deleted.
- 2 The default is none. Use with care!

?

- 1 The query option which will list all saved UCS positions
- 2 At the command line enter **UCS <R>** then **? <R>** and:
prompt Enter UCS name(s) to list<*>
respond **press the RETURN key**
prompt AutoCAD Text Window with:
Current UCS Name: BASE
Saved Coordinate systems
and Details about all the saved UCS's, their origin point and their X,Y and Z axes orientation
respond **cancel the window**

The UCS toolbar

All the UCS options have so far been activated by keyboard entry with **UCS <R>** or from the menu bar with **Tools**. The only reason for this is that I think it easier for the user to understand what option is being used. The UCS options can also be activated in icon form from the UCS and UCS II toolbars – Fig. 5.2. The toolbars have no icon selection for the orthographic options or for Restore, Save, Delete or for query (?), although these can easily be activated by selecting the actual UCS icon. An additional icon in both the UCS and UCS II toolbars is Display UCS Dialog, while the UCS II toolbar allows saved UCS's to be made current, i.e. restored.

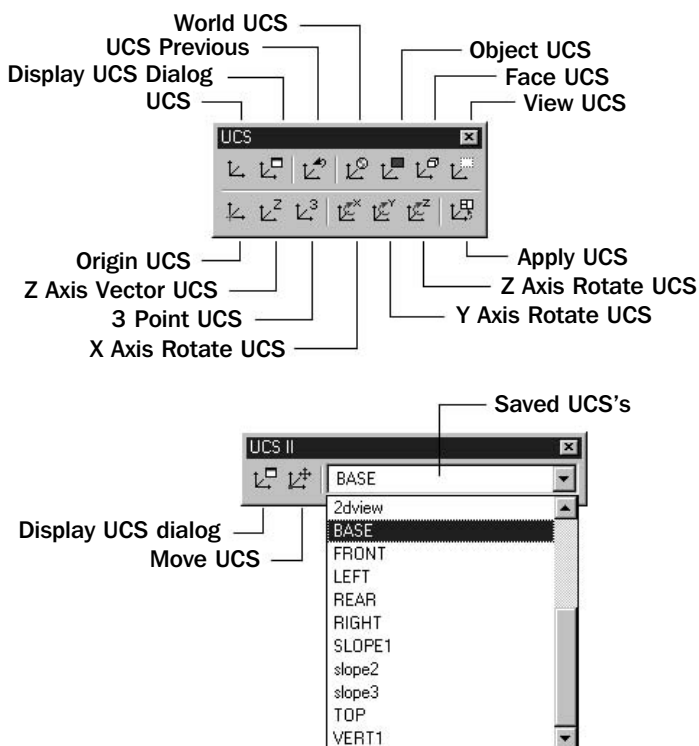


Figure 5.2 The UCS and UCS II toolbars.

The user now has three methods of activating the various UCS options, these being:

- from the menu bar
- by command line entry
- in icon form from the appropriate toolbar

It is user preference as to what method is used.

The UCS dialogue box

The UCS dialogue box can be activated by three different methods:

- a) from the menu bar with **Tools-Named UCS**
- b) by selecting the Display UCS dialog icon from either the UCS or UCS II toolbar
- c) by entering **UCSMAN <R>** at the command line



When activated, the dialogue box allows the user three tab selections, these being:

- a) Named UCSs – the default
- b) Orthographic UCSs
- c) Settings

To demonstrate using the UCS dialogue box:

- 1 Ensure the 3DWFV is displayed with UCS BASE current
- 2 Menu bar with **Tools-Named UCS** and:
 - prompt* UCS dialogue box
 - with* three tab selections and Named UCSs tab active
 - and*
 - a) a list of saved UCS names for the model
 - b) a World and Previous selection option
 - respond*
 1. pick Top
 2. pick Set Current – Fig. 5.3
 3. pick OK
- 2 The model will be displayed with the icon at the TOP setting
- 3 Use the Named UCS tab of the UCS dialogue box to set current some other saved UCS positions
- 4 Set UCS BASE current

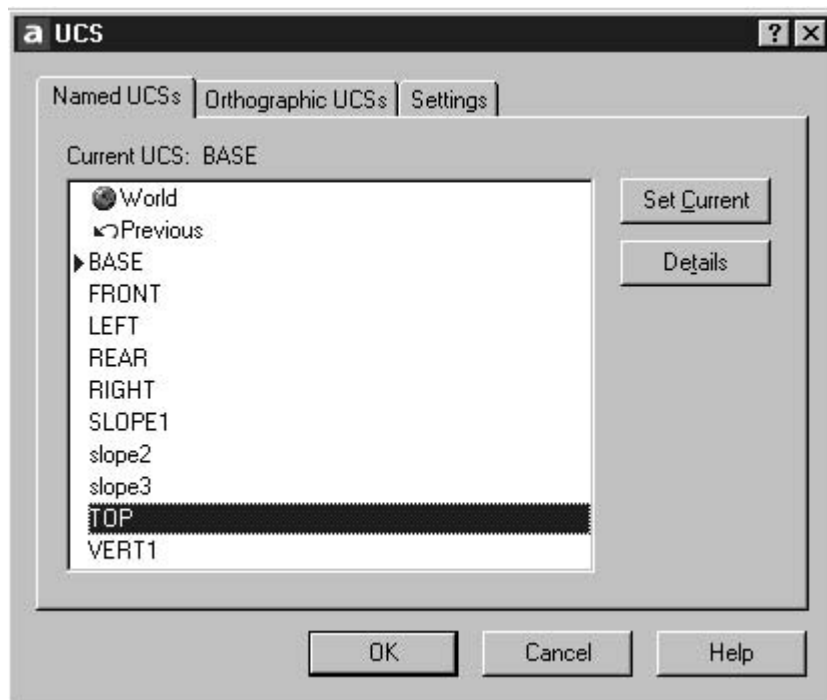


Figure 5.3 The UCS dialogue box – Named UCS tab.

- 5 Activate the UCS dialogue box and:
 - prompt* UCS dialogue box – Named UCS tab active
 - respond*
 1. pick TOP and it becomes highlighted
 2. right-click the mouse
 - prompt* shortcut menu
 - with* selections for: Set Current, Rename, Delete, Details
 - respond*
 1. pick Rename
 2. enter new name: ABOVE <R>
 3. pick Set Current
 4. pick OK
- 6 The UCS will be displayed in the ‘old top position’
- 7 Now:
 - a) rename the ABOVE UCS to TOP again
 - b) make UCS BASE current
- 8 Activate the UCS dialogue box and pick the Settings tab and:
 - prompt* Settings tab – Fig. 5.4
 - with*
 1. UCS icon settings for ON and ORIGIN – both active
 2. UCS settings for viewports and plan
 - respond* note the settings then pick Cancel
- 9 *Note*
 - a) The UCS icon settings from the dialogue box are the same as the menu bar selection of View-Display-UCS Icon-On/Origin
 - b) The other Settings options will be discussed in later chapters

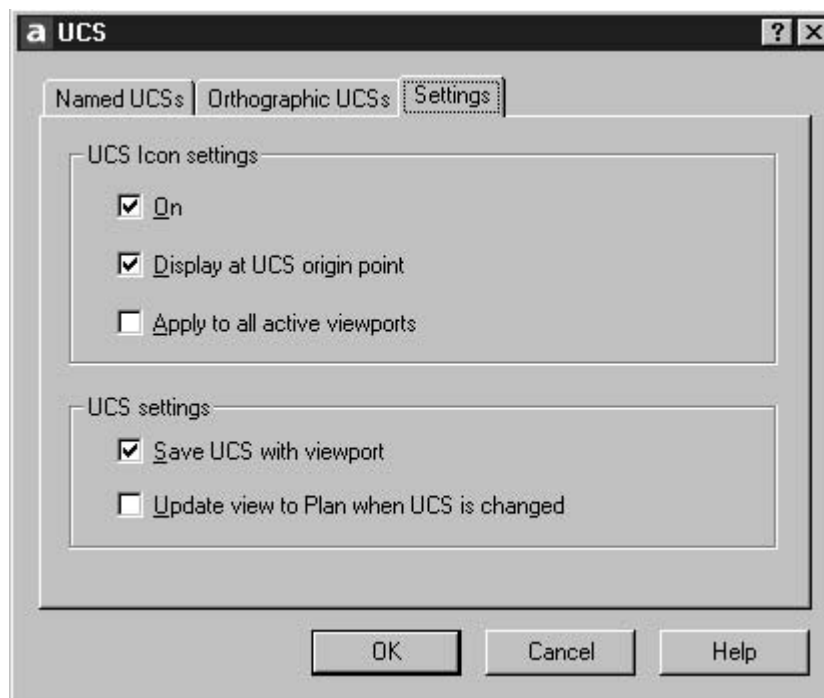


Figure 5.4 The UCS dialogue box – Settings tab.

Setting an orthographic UCS

This allows the user to restore six preset UCS positions, the orientation being set relative to a saved UCS. Refer to Fig. 5.5 and:

- 1 Ensure the 3DWFM is displayed with the saved UCSs
- 2 Restore UCS SLOPE1 current – fig(a)
- 3 Activate the UCS dialogue box with the Orthographic UCS tab active and:
 - prompt* UCS dialogue box – Orthographic tab display
 - respond* 1. scroll at Relative to
 - 2. pick BASE
 - 3. at Current UCS Name, pick TOP
 - 4. pick Set Current – Fig. 5.6
 - 5. pick OK
- and* icon displayed as fig(b)
- 4 With the Orthographic tab of the UCS dialogue box, set Bottom current relative to BASE – fig(c)
- 5 Menu bar with **Tools-Orthographic UCS-Left** to display the icon as fig(d)

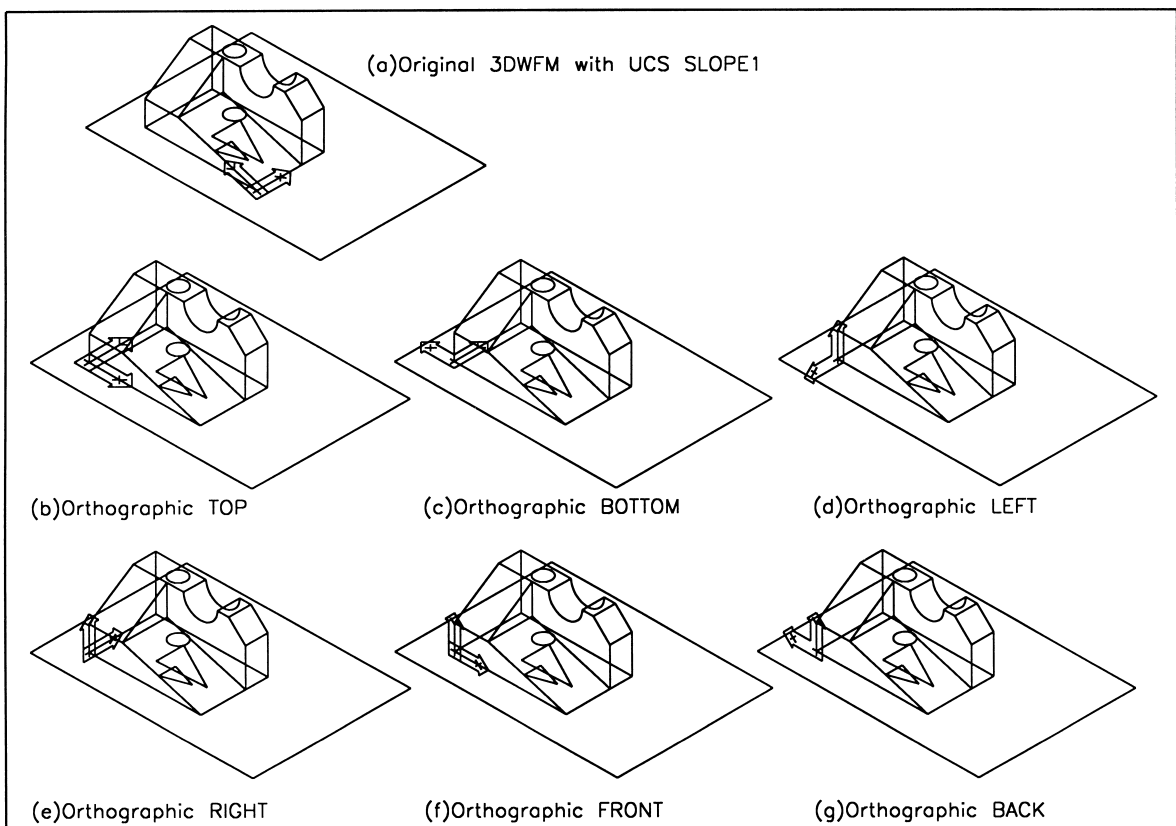


Figure 5.5 The Orthographic UCS options exercise relative to BASE with UCS SLOPE1 current.

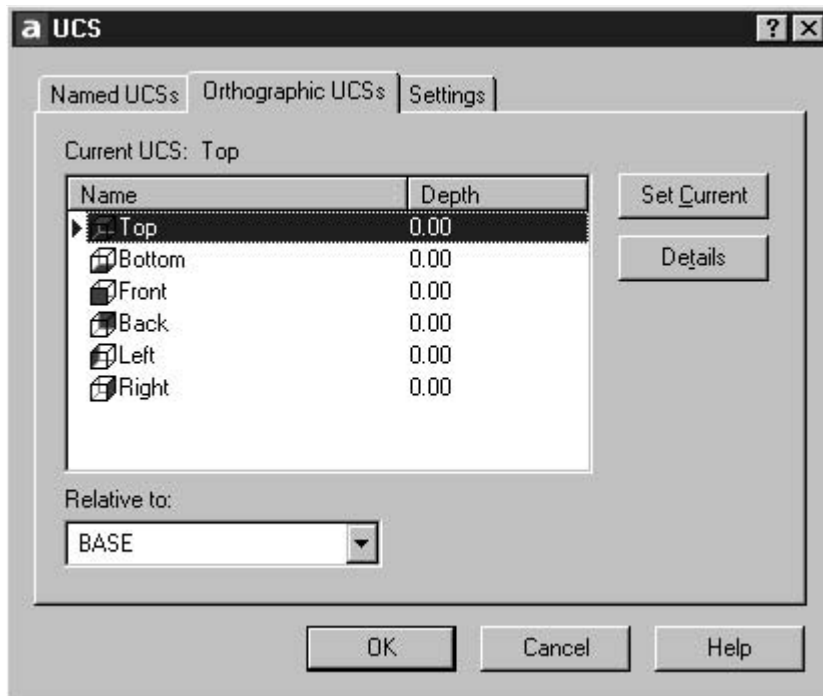


Figure 5.6 The UCS dialogue box – Orthographic tab active.

- 6 At the command line enter **UCS <R>** and:
 - prompt* Enter an option [New/Move/orthoGraphic/..
 - enter* **G <R>** – the orthographic option
 - prompt* Enter an option [Top/Bottom/Front/..
 - enter* **R <R>** – the right orthographic UCS option
 - and* the icon will be displayed as fig(e)
- 7 With the menu bar **Tools-Orthographic UCS** sequence, select relative to BASE:
 - a) FRONT current – fig(f)
 - b) BACK current – fig(g)
- 8 Activate the Orthographic tab of the UCS dialogue box and:
 - a) set relative to SLOPE1
 - b) activate the six orthographic UCS positions
 - c) note the orientation of the UCS with each orthographic name
 - d) restore UCS BASE
- 9 This exercise with the UCS dialogue box is now complete.
- 10 *Note*
The menu bar sequence **Tools-Orthographic UCS-Presets** will display the Orthographic tab of the UCS dialogue box.

UCS specific commands

The UCS has several specific commands associated with it. At this stage, we will only investigate:

- a) the PLAN command
- b) the UCSFOLLOW system variable.

Plan

Plan is a command which displays any model perpendicular to the XY plane of the current UCS position.

- 1 Ensure the 3DWFM is displayed with UCS BASE current
- 2 Refer to Fig. 5.7
- 3 At the command line enter **PLAN <R>** and:
 - prompt* Enter an option [Current ucs/Ucs/World] <Current>
 - enter* <R> i.e. accept the Current UCS default
- 4 The screen will display the model as a plan view – fig(a). This view is perpendicular to the current UCS setting (BASE) and is really a ‘top’ view in orthogonal terms
- 5 Restore UCS FRONT – pencil icon?
- 6 Menu bar with **View-3D Views-Plan View-Current UCS** and the model will be displayed as fig(b). This is a plan view to the current UCS FRONT and is a ‘front’ view in orthogonal terms
- 7 Menu bar with **View-3D Views-Plan View-Named UCS** and:
 - prompt* Enter name of UCS
 - enter* **SLOPE1 <R>**
- 8 The model will be displayed as a plan to the UCS SLOPE1 setting as fig(c)
- 9 At the command line enter **PLAN <R>** and:
 - prompt* Enter an option [Current ucs/Ucs/World]<Current>
 - enter* **U <R>** – the Ucs option
 - prompt* Enter name of UCS
 - enter* **VERT1 <R>**

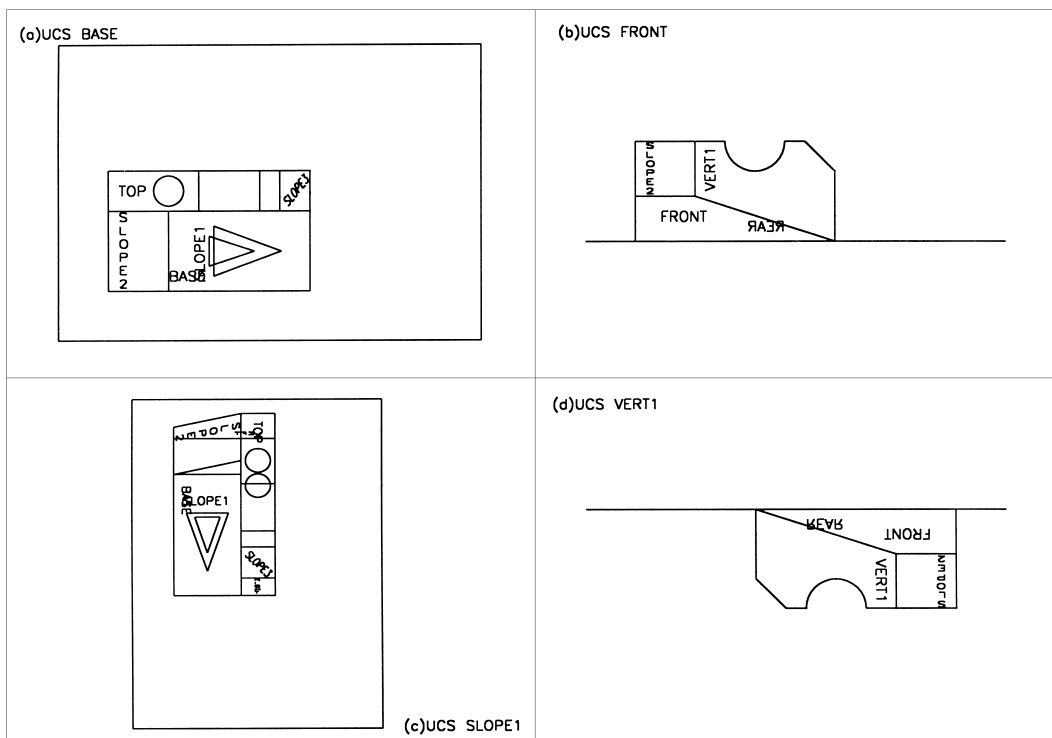


Figure 5.7 The PLAN command with 3DWFM.

- 10 The model display is as fig(d) i.e. a plan view to the UCS setting VERT1. This display should be upside-down – why?
- 11 Finally restore UCS BASE and menu bar with View-3D Views-SE Isometric to return the original model display.

UCSFOLLOW

UCS FOLLOW is a system variable which controls the screen display of a model when the UCS position is altered. The variable can only have the values of 0 (default) or 1 and:

- a) UCSFOLLOW 0: no effect on the display with UCS changes
- b) UCSFOLLOW 1: automatically generates a plan view when the UCS is altered

- 1 Original 3D display with UCS BASE on the screen?
- 2 At the command line enter **UCSFOLLOW <R>** and:
prompt Enter new value for UCSFOLLOW <0>
enter **1 <R>**
- 3 Nothing has changed?
- 4 Restore UCS FRONT – plan view as Fig. 5.7(b)
- 5 Restore UCS SLOPE1 – plan view as Fig. 5.7(c)
- 6 Restore UCS VERT1 – plan view as Fig. 5.7(d)
- 7 Restore UCS BASE – plan view as Fig. 5.7(a)
- 8 Set UCSFOLLOW to 0 and restore the original screen display with View-3D Views-SE Isometric
- 9 This completes the exercises with the UCS

Summary

- 1 The UCS is an essential 3D modelling aid
- 2 The UCS command has several options including:
 - a) New: origin, 3 point, X,Y,Z rotate
 - b) Move: which should be used with caution
 - c) Orthographic: six preset UCS settings
- 3 The orientation of the UCS icon is dependent on the option used
- 4 The UCS toolbars offer fast option selection
- 5 It is **STRONGLY RECOMMENDED** that the UCS icon and the UCS icon origin are ON when working in 3D. These can be activated with:
 - a) the menu bar sequence View-Display-UCS Icon
 - b) the Settings tab of the UCS dialogue box
- 6 The UCS dialogue box allows flexible management of the UCS with three tab selections:
 - a) Named UCSs – set current, rename, delete
 - b) Orthographic UCSs (Presets)
 - c) Settings
- 7 PLAN is a command which displays the model perpendicular to the XY plane of the current UCS
- 8 UCSFOLLOW is a system variable which can be set to give automatic plan views when the UCS is re-positioned. It is recommended that this variable be set to 0, i.e. off.

The modify commands with 3D models

All the modify commands are available for use with 3D models, but the results are dependent on the UCS position. We will investigate how the COPY, ARRAY, ROTATE and MIRROR commands can be used with our 3D wire-frame model so:

- 1 Open your 3DWFM model with UCS BASE and layer MODEL current
- 2 Display the Modify, Object snap and UCS toolbars
- 3 Erase all text but the FRONT text item

The COPY command

- 1 Select the COPY icon from the Modify toolbar and:

prompt Select objects

respond **pick the 4 red lines and the green FRONT text item on the 'front vertical' plane then right-click**

prompt Specify base point or displacement

respond **Intersection icon and pick ptA**

prompt Specify second point of displacement

enter **@0,0,260 <R>** – Fig. 6.1.A(a)

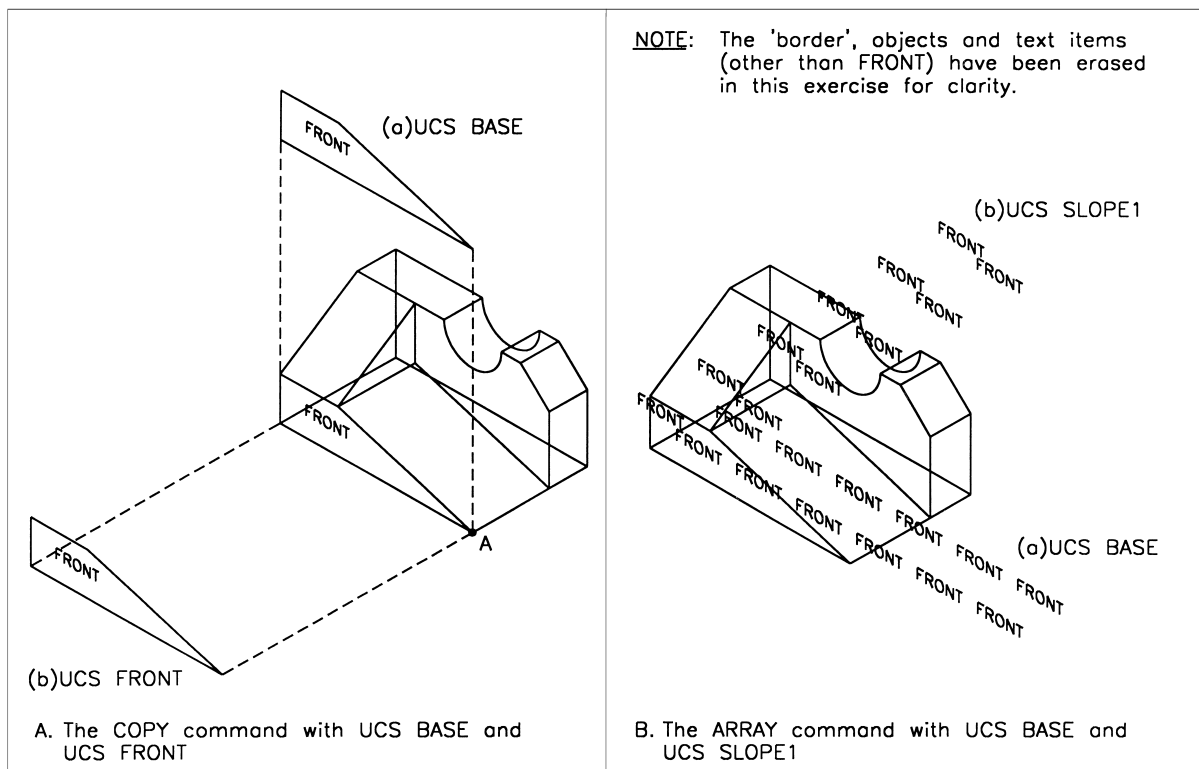


Figure 6.1 The COPY and ARRAY commands with 3DWFM.

- 2 Restore UCS FRONT
- 3 Select the COPY icon and:
 - prompt* Select objects
 - enter* **P <R><R>** – previous selection set option
 - prompt* Specify base point and: **pick Intersection of ptA**
 - prompt* Specify second point and enter: **@0,0,260 <R>** – Fig. 6.1.A(b)
- 4 Menu bar with **View-Zoom-All**
- 5 Undo (or erase) the copied effects to leave the original model

The ARRAY command

- 1 Restore UCS BASE
- 2 At the command line enter **-ARRAY <R>** and:
 - prompt* Select objects
 - respond* **pick the FRONT text item then right-click**
 - prompt* Enter the type of array and enter: **R <R>**
 - prompt* Enter the number of rows and enter: **2 <R>**
 - prompt* Enter the number of columns and enter: **6 <R>**
 - prompt* Enter the row distance and enter: **40 <R>**
 - prompt* Specify the distance between columns and enter: **60 <R>**
- 3 The text item is arrayed in a 2×6 rectangular matrix as Fig. 6.1.B(a)
- 4 Restore UCS SLOPE1
- 5 Rectangular array the original FRONT text item using the same entries as step 2 – Fig. 6.1.B(b)
- 6 Undo (or erase) the arrayed effects

The ROTATE command

- 1 Restore UCS BASE
- 2 Select the ROTATE icon from the Modify toolbar and:
 - prompt* Select objects
 - respond* **pick the 4 red lines and the FRONT text item as before then right-click**
 - prompt* Specify base point
 - respond* **Intersection icon and pick ptA**
 - prompt* Specify rotation angle
 - enter* **90 <R>**
- 3 The selected objects will be rotated as Fig. 6.2.A(a)
- 4 Undo this rotated effect
- 5 Restore UCS FRONT and rotate the same objects with the same entries as step 2. This will give Fig. 6.2.A(b)
- 6 Undo this effect

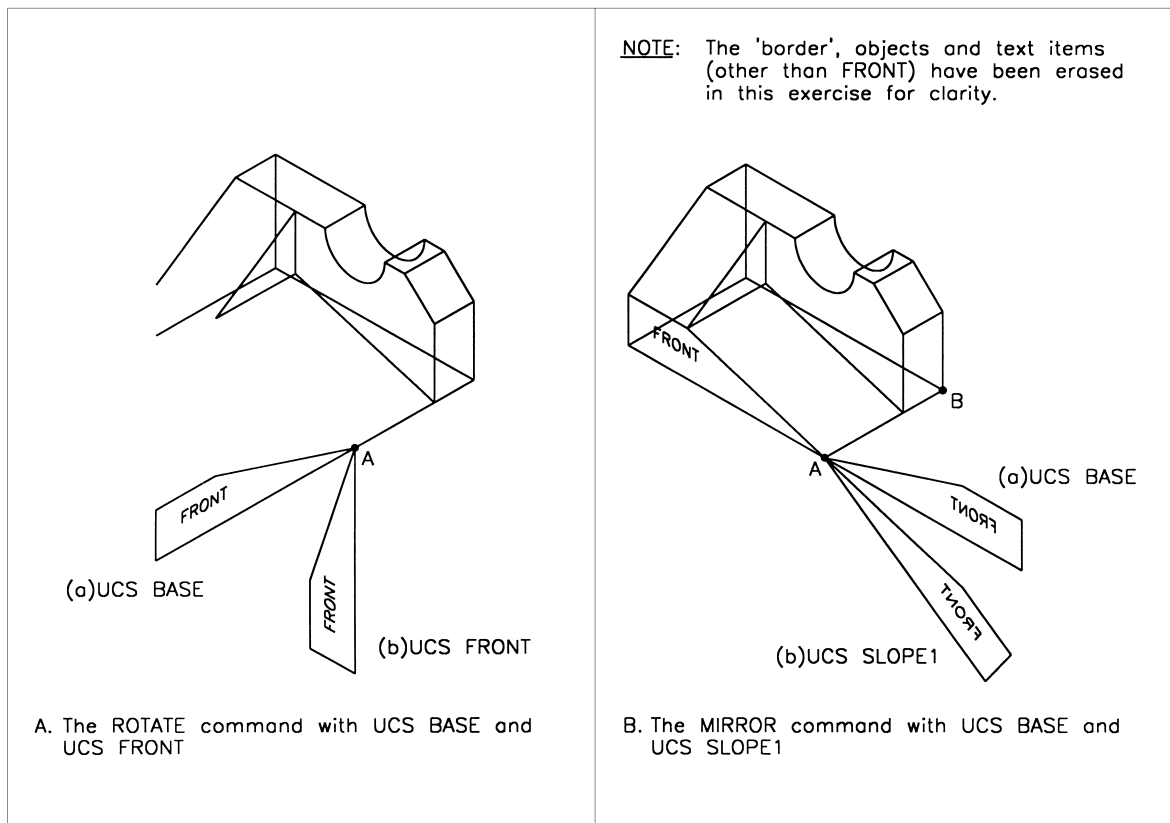


Figure 6.2 The ROTATE and MIRROR commands with 3DWMF.

The MIRROR command

- Restore UCS BASE, activate the Mirror icon and:
 - prompt* Select objects
 - respond* **pick the four lines and text item as before and right-click**
 - prompt* Specify first point of mirror line
 - respond* **Intersection icon and pick ptA**
 - prompt* Specify second point of mirror line
 - respond* **Intersection icon and pick ptB**
 - prompt* Delete source objects
 - enter* **N <R>**
- The selected objects will be mirrored about the line AB as Fig. 6.2.B(a)
- Restore UCS SLOPE1 and repeat the mirror command with the same objects and same entries as step 1. This will give Fig. 6.2.B(b)

Other modify commands

All of the modify commands are available for use with 3D models, but the final result is dependent on the UCS position. The only requirement for the user is to ensure that the icon is positioned on the 'plane' for the modification.

Summary

The AutoCAD modify commands with 3D models have to be used with care. The result is UCS dependent.

Dimensioning in 3D

There are no special commands to add dimensions in 3D. Dimensioning is a 2D concept, the user adding the dimensions to the XY plane of the current UCS setting. This means that the orientation of the complete 'dimension object' will depend on the UCS position. The user should be aware of:

- a) AutoCAD's automatic dimensioning facility
- b) linear dimensioning will be horizontal or vertical, depending on where the dimension line is located in relation to the object being dimensioned.

We will demonstrate how dimensions can be added to 3D models with two examples. The first will be the 3D wire-frame model 3DWFM, and the second will use AutoCAD's stored 3D objects.

Example 1

- 1 Open MODR2002\3DWFM and display the Dimension, Object Snap and other toolbars to suit
- 2 Freeze layer TEXT and make layer DIM current
- 3 The standard sheet created as the template/drawing file had a created dimension style – 3DSTD. You may want to 'alter' the Overall Scale (Fit tab) to a value of 1.5 which will make the added dimensions 'clearer'
- 4 Ensure UCS BASE is current and refer to Fig. 7.1

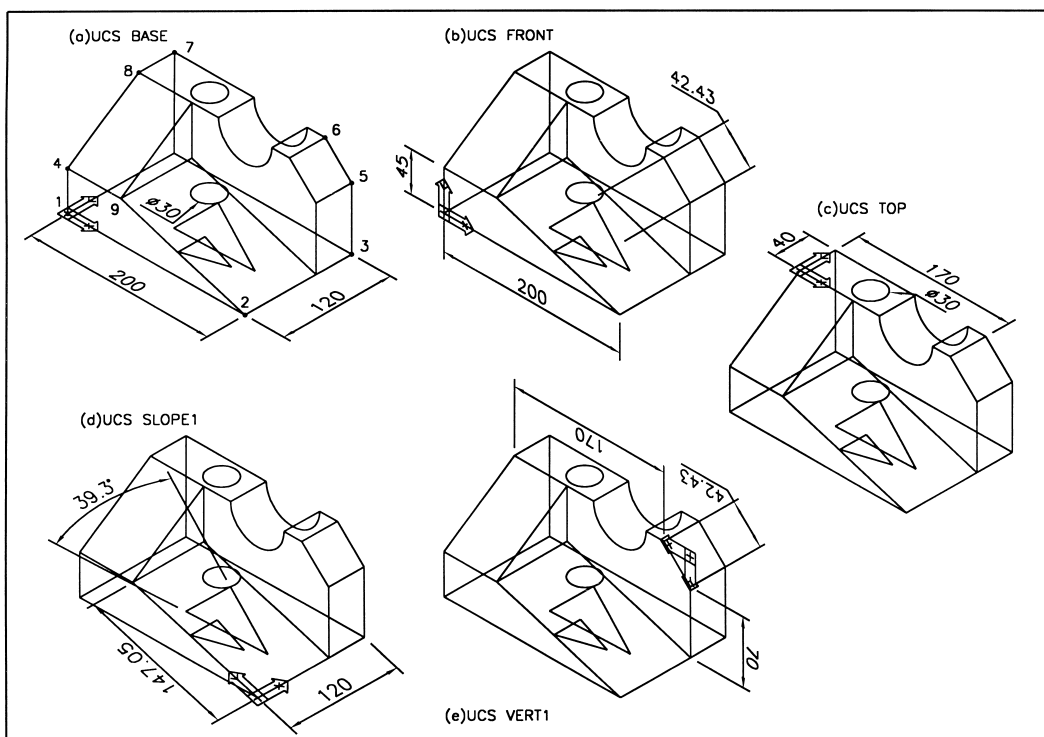



Figure 7.1 Dimension exercise with 3DWFM.

- 5 Select the LINEAR dimension icon and:
 - prompt* Specify first extension line origin
 - respond* **Intersection icon and pick pt1**
 - prompt* Specify second extension line origin
 - respond* **Intersection icon and pick pt2**
 - prompt* Specify dimension line location
 - respond* **pick to suit**
- 6 Repeat the LINEAR dimension selection and dimension line 23, positioning the dimension line to suit
- 7 Select the DIAMETER icon and:
 - prompt* Select circle or arc
 - respond* **pick the circle on the TOP 'plane'**
 - prompt* Specify dimension line location
 - respond* **pick to suit** – interesting result?
- 8 The three added dimensions will be displayed as fig(a)
- 9 Erase the added dimensions and restore UCS FRONT
- 10 Using the dimension icons:
 - a) linear dimension lines 12 and 14
 - b) align dimension line 56
 - c) try and add a diameter dimension to the top circle
 - d) dimensions displayed as fig(b)
- 11 Erase these added dimensions and restore UCS TOP and:
 - a) linear dimension line 67 and line 78
 - b) diameter dimension the circle on the top
 - c) result as fig(c)
- 12 Restore UCS SLOPE1, erase the previous dimensions and:
 - a) linear dimension line 23 and line 29
 - b) angular dimension a vertex of the blue triangle 'on the slope'
 - c) the three dimensions will be displayed as fig(d)
- 13 With UCS VERT1 current, erase the dimensions from SLOPE1 and:
 - a) linear dimension line 67 and line 35
 - b) align dimension line 56
 - c) interesting result as fig(e) – why?
- 14 This exercise should demonstrate to the user that:
 - a) adding dimensions to a 3D model is **VERY UCS DEPENDENT**
 - b) there are no special 3D dimension commands
 - c) the actual orientation of added dimensions depends on the UCS position and orientation
 - d) dimensions are added to the XY plane of the current UCS
- 15 *Task*
 - a) erase any dimensions still displayed
 - b) with layer DIM current refer to Fig. 7.2 and add the given to the model
 - c) some of the existing saved UCS positions will be used
 - d) you may have to set a new UCS position for the continuous 80,40 and the 70 dimensions
 - e) when complete save if required, but not as 3DWFM

- 4 Menu bar with **Draw-Surfaces-3D Surfaces** and:
- prompt* 3D Objects dialogue box
respond **pick Box3d** then OK
prompt Specify corner of box and enter: **50,50,0 <R>**
prompt Specify length of box and enter: **120 <R>**
prompt Specify width of box or [Cube] and enter: **80 <R>**
prompt Specify height of box and enter: **60 <R>**
prompt Specify rotation of box about Z axis and enter: **0 <R>**
- 4 Select the WEDGE icon from the surface toolbar and:
- prompt* Specify corner of wedge and enter: **200,150,0 <R>**
prompt Specify length of box and enter: **100 <R>**
prompt Specify width of box and enter: **75 <R>**
prompt Specify height of box and enter: **65 <R>**
prompt Specify rotation of wedge about Z axis and enter: **0 <R>**
- 5 Using the 3 point UCS option:
- origin at point 1
 - x axis along line 12
 - y axis along line 13
 - save UCS position as POS1
 - add two linear dimensions with this UCS
- 6 Use the 3 point UCS option again with:
- origin at point a
 - x axis along line ab
 - y axis along line ac
 - save as POS2
 - add two linear dimensions with this UCS
- 7 Now add the three other dimensions, i.e. one to the box and two to the wedge. Some UCS manipulation is required but you should manage this without any problems.
- 8 Menu bar with **Tools-New UCS-World** to return to the WCS.
- 9 *Task*
 Another 3D object has to be created and dimensioned so select the pyramid icon from the Surfaces toolbar and: 
- prompt* Specify first corner point for base of pyramid and enter: **40,220 <R>**
prompt Specify second corner point for base of pyramid and enter: **80,250 <R>**
prompt Specify third corner point for base of pyramid and enter: **60,320 <R>**
prompt Specify fourth corner point for base of pyramid or [Tetrahedron]
enter **T <R>** – the tetrahedron option
prompt Specify apex point of tetrahedron
enter **60,320,75 <R>**
- 10 Now set appropriate UCS positions and add the three linear dimensions given
- 11 Save if required, but the drawing will not be used again

Summary

- 1 There are no special 3D dimension commands
- 2 Dimensioning is a 2D concept and dimensioning a 3D model involves adding the dimensions to the XY plane of the required UCS setting
- 3 If the UCS is not positioned correctly, dimensions can have the 'wrong orientation' in relation to the object being dimensioned.

Assignment

Two dimensioning assignments for you to complete, one being an existing wire-frame model, the other being a new wire-frame model. Both involve the work of our great architect MACFARAMUS.

Activity 4: Shaped block of MACFARAMUS

Recall the shaped block from activity 2 and:

- a) set and save appropriate UCS positions
- b) add the given dimensions
- c) save the completed model for later recall

Activity 5: The famous rectangular topped pyramid of MACFARAMUS

The rectangular topped pyramid designed by Macfaramus was never built but his design is still considered unique. You have to:

- a) create a wire-frame model of the pyramid using the sizes given
- b) set and save several UCS positions, these being on the following surfaces:
 1. the base and top – UCS BASE and UCS TOP
 2. the four 'sides' of the pyramid, which I have names as SLOPE1, SLOPE2, SLOPE3 and SLOPE4 (I know that two of these sides are not sloped, I named them in a logical order)
 3. the vertical sides of the top. Use your own names, e.g. V1, etc.
- c) add the given dimensions
- d) save the completed model

MACFARAMUS had to calculate certain angles and distances without the aid of calculators or computers. This took him a great deal of time and involved using his knowledge of trigonometry. The angles and distances he required were:

1. the angle of the two sloped 'planes' in relation to the base
2. the length of the three sloped 'edges'

His calculations gave the following values:

- a) angle of first sloped plane to the base: 50.19
- b) angle of second sloped plane to the base: 56.31
- c) length of the three sloped edges:
 - 1 – 234.31
 - 2 – 263.25
 - 3 – 216.33

Was he correct?

Hatching in 3D

There are no special 3D hatch commands. Hatching (like dimensioning) is a 2D concept, the hatch pattern being added to the XY plane of the current UCS. Two examples will be used to demonstrate adding hatching to 3D models.

Example 1

- 1 Open your 3DSTDA3 template file
- 2 Display the Draw, Modify and Object Snap toolbars
- 3 With layer MODEL current, refer to Fig. 8.1 and draw the four 'mutually perpendicular planes' using the LINE icon with:

	<i>Plane 1234</i>	<i>Plane 1564</i>	<i>Plane 3764</i>	<i>Plane 7896</i>
Start	30,30,0	30,30,0	130,130,0	130,130,100
Next	@100,0,0	@0,0,100	@0,0,100	@0,100,0
Next	@0,100,0	@0,100,0	@-100,0,0	@-100,0,0
Next	@-100,0,0	@0,0,-100	<RETURN>	@0,-100,0
Next	close	<RETURN>		<RETURN>

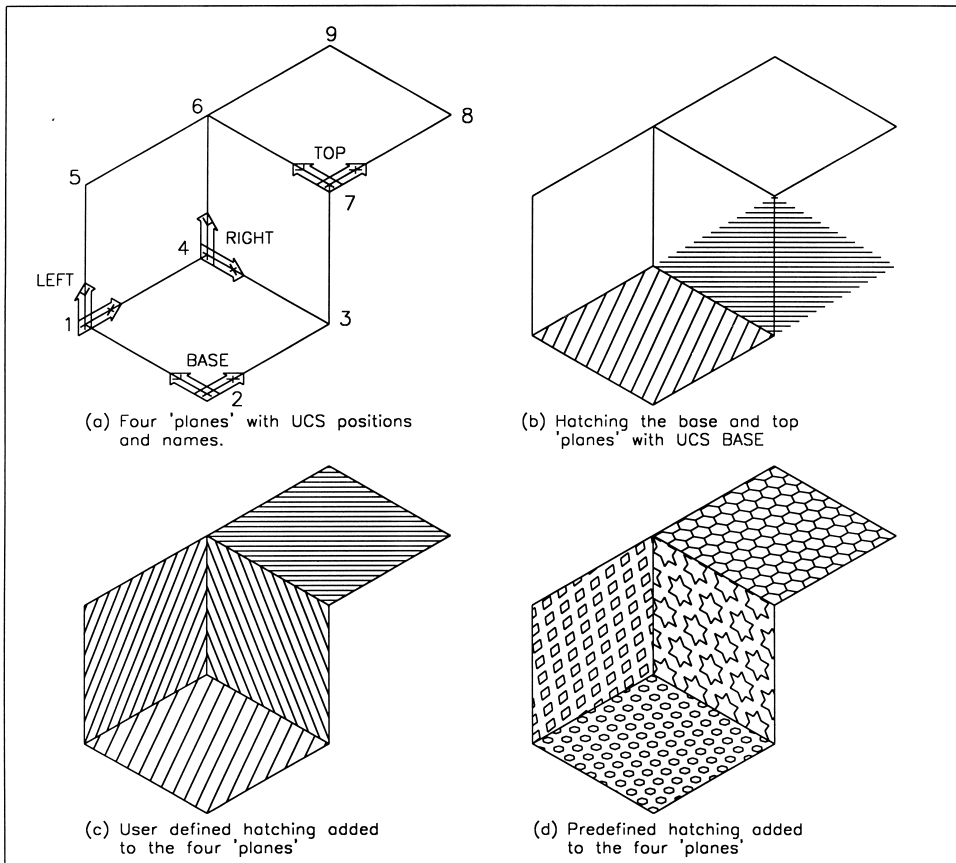


Figure 8.1 3D hatch exercise 1.

- 4 Erase the black border and pan the drawing to suit
- 5 Set and save the four UCS positions as fig(a)
- 6 Restore UCS BASE and make SECT the current layer
- 7 Select the HATCH icon from the Draw toolbar and using the Boundary Hatch dialogue box:
 - a) pick User defined pattern type
 - b) set angle to 30 and spacing to 8
 - c) use the **Pick Points**< option and:
 1. select a point within the 1234 plane then right-click/enter
 2. Preview-right click-OK
- 8 Repeat the HATCH icon selection and:
 - a) using the Pick Points option pick a point within the 6789 plane and:

prompt Boundary Definition Error dialogue box – Fig. 8.2

respond **pick OK then <RETURN>**
 - b) using the Select Objects option pick the four lines of the 6789 plane then right-click/enter and:
 1. set angle to –45 and spacing to 5
 2. Preview-right click-OK



Figure 8.2 The Boundary Definition Error Message box.

- 9 The result of the two hatch operations is displayed in fig(b) with plane 1234 having the correct hatching, but plane 6789 has none, the hatching having been added to the plane of UCS BASE.
- 10 Use the HATCH icon and try and add hatching to the vertical planes 1564 and 3467.
 - a) Not possible?
 - b) Unable to hatch the boundary message at the command line
 - c) Why is this?
- 11 Erase the 'wrong' hatching and restore UCS TOP
- 12 Hatch the top plane (6789) using the HATCH icon with:
 - a) pick points option
 - b) angle –45 and spacing 5
- 13 Add hatching to the two vertical planes remembering to restore UCS LEFT and UCS RIGHT – fig(c). Use your own angle and spacing values.
- 14 *Task*
 - a) Erase the added hatching then add the following predefined hatch patterns using the information given:

UCS	Pattern	Scale	Angle
BASE	HEX	1	–10
TOP	HONEY	2	0
LEFT	SQUARE	2	10
RIGHT	STARS	2	0
 - c) the result is fig(d)
 - d) this completes example 1 which does not have to be saved.

Example 2

- 1 Open the drawing C:\MODR2002\3DWFM and refer to Fig. 8.3
- 2
 - a) erase all text and any dimensions
 - b) erase the smaller triangle and lower circle on the base plane
 - c) make layer SECT current
 - d) restore UCS TOP
- 3 Select the HATCH icon and:
 - a) Predefined pattern type: scroll and pick STARS
 - b) set scale to 1 and angle to 0
 - c) select Pick Points: pick internal points in the **TWO** top planes then right-click/enter
 - d) preview-right click-OK
- 4 Restore UCS FRONT and with the HATCH icon:
 - a) pattern type: ZIGZAG
 - b) scale: 1.5 and angle: 30
 - c) select objects: pick the four lines of front vertical plane then right-click/enter
 - d) preview-right click-OK
- 5 Repeat the HATCH icon selection and add the following hatch patterns:

UCS	Pattern	Scale	Angle	Selection type
SLOPE1	TRIANG	1	25	pick points
VERT1	HOUND	2.5	0	pick points
- 6 Menu bar with **View-Hide** and the model is as before. Hatching a wire-frame model does not produce a hide effect.
- 7 Save your completed hatched model but not as MODR2002/3DWFM.
- 8 *Note*

In the two hatch exercises we used the one layer (SECT) for all hatching. It is sometimes desirable to have a different layer for each current UCS that is to be used for hatching. This is a user decision.

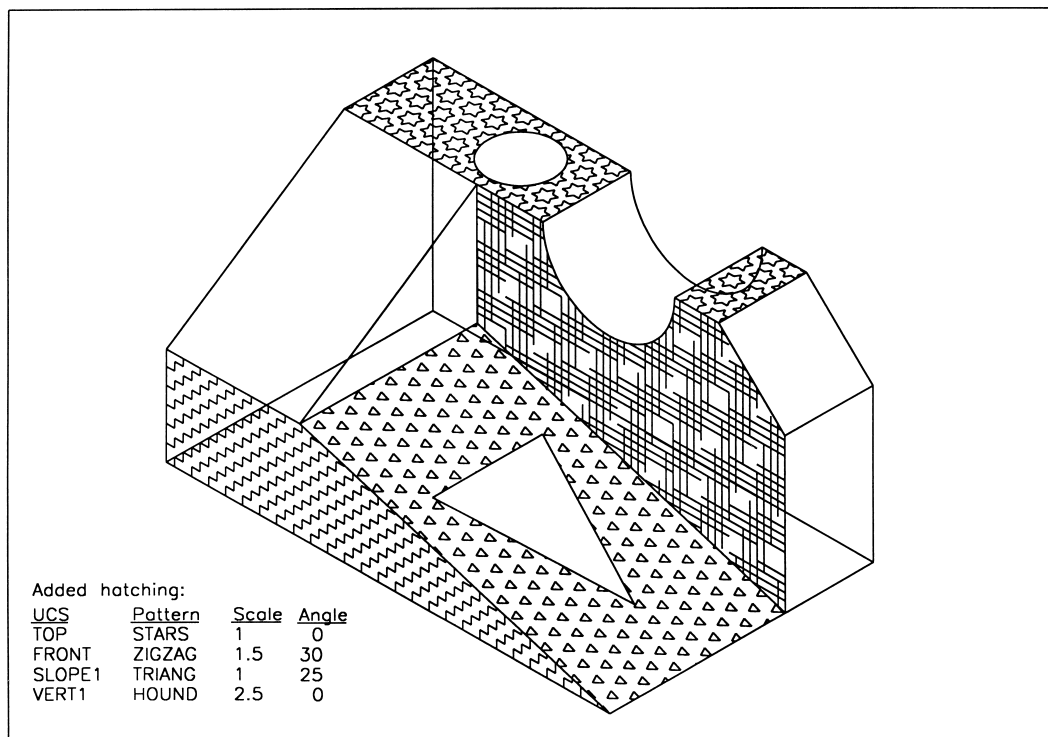


Figure 8.3 Hatch exercise with 3DWFM.

Summary

- 1 Hatching is a 2D concept
- 2 Hatching a 3D model requires the UCS to be set to the 'plane' which is to be hatched
- 3 a) both user-defined and pre-defined hatch patterns can be used
b) both the select objects and pick points methods are permitted
- 4 Hatching a 3D wire-frame model does not produce a hide effect

Assignments

Two hatch activities have been included, both using previously created wire-frame models these being MACFARAMUS's shaped block and rectangular topped pyramid.

The procedure with both activities is to:

- 1 Open the saved drawing
- 2 Freeze layer DIM, or erase the dimensions if necessary
- 3 Add appropriate hatching, making new hatch layers for each UCS position if required – your decision!
- 4 Use the pick points method where possible
- 5 Save the completed model.

Activity 6: The shaped block of MACFARAMUS

- 1 Restore the appropriate UCS and add the following user defined hatching:

<i>UCS</i>	<i>Angle</i>	<i>Spacing</i>
FRONT	45	8
RIGHT	45	8
MID	45	8
SLOPE	45	8

- 2 When complete save the model as **MODR2002\SHBLOCK** for recall.

Activity 7: The rectangular topped pyramid of MACFARAMUS.

- 1 Using the correct UCS, add the following predefined hatch patterns:

<i>UCS</i>	<i>Pattern</i>	<i>Scale</i>	<i>Angle</i>
four 'shaped' planes	BRICK	2	0
four vert 'top' planes	BRSTONE	1	0
the horiz top surface	EARTH	1.5	0

- 2 Save the complete hatched model as **MODR2002\PYRAMID**

- 3 *Note:*

- a) extra lines required for the hatch effect?
- b) the viewpoint has been altered for effect.

Tiled viewports

- 1 Up until now, all the models that have been created have been displayed as a single object on the screen. The graphics screen can however, be divided into a number of separate viewing areas called **viewports** and each viewport can display different viewpoints of a model.
- 2 Viewports are **interactive**, i.e. what is drawn in one viewport is automatically displayed in the others and the user can switch between viewports when creating a model. Viewport layouts (**configurations**) can be saved thus allowing different displays of the same model to be stored for future recall.
- 3 Viewports are essential with 3D modelling as they allow different views of the model to be displayed on the screen simultaneously.
- 4 When used with the VIEWPOINT command (next chapter) the user has a very powerful 3D draughting tool.
- 5 There are two types of viewport (displayed in Fig. 9.1) these being:
 - a) Tiled or fixed
 - b) Untiled or floating
- 6 The type of viewport which is displayed is controlled by the **TILEMODE** system variable and:
 - a) Tilemode 1: tiled viewports – the default setting. These viewports are fixed and cannot be altered by the user
 - b) Tilemode 0: untiled viewports – can be altered by the user.
- 7 In this chapter we will only investigate TILED viewports and leave the untiled viewport discussion to a later chapter when we will investigate model and paper space.
- 8 Making viewports can be activated by keyboard entry or from the menu bar.

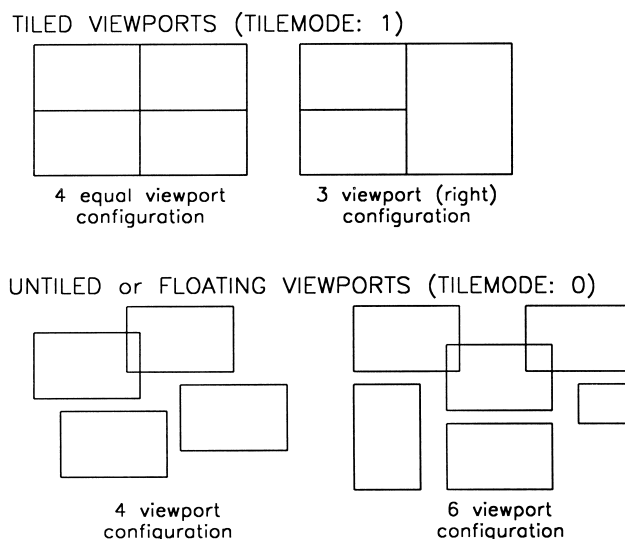


Figure 9.1 Tiled and untiled viewports.

Example 1

This exercise is rather long, but persevere with it.

- 1 Open your 3DWF drawing of the wire-frame model on the black border with layer MODEL current and UCS BASE.
- 2 Deactivate all floating toolbars and display the model without any text, dimensions or hatching. Erase or freeze layers?
- 3 At the command line enter **TILEMODE <R>** and:
prompt Enter new value for TILEMODE<1>
and observe the 1 default then press **ESC**
- 4 The TILEMODE value of 1 indicates that only TILED viewports can be used. The same condition is also evident with:
 - a) Status bar: word MODEL is displayed
 - b) Menu bar: View-Viewports indicate that Polygonal Viewport and Object are not available for selection
- 5 Menu bar with **View-Viewports-3 Viewports** and:
prompt Enter a configuration [Horizontal/Vertical/Above..
enter **R <R>** – the right configuration option
- 6 The drawing screen will:
 - a) be divided into three separate 'areas' – one large at the right and two smaller to the left. The three viewports will 'fill the screen' as Fig. 9.2(a)
 - b) display the same view of the model in the three viewports

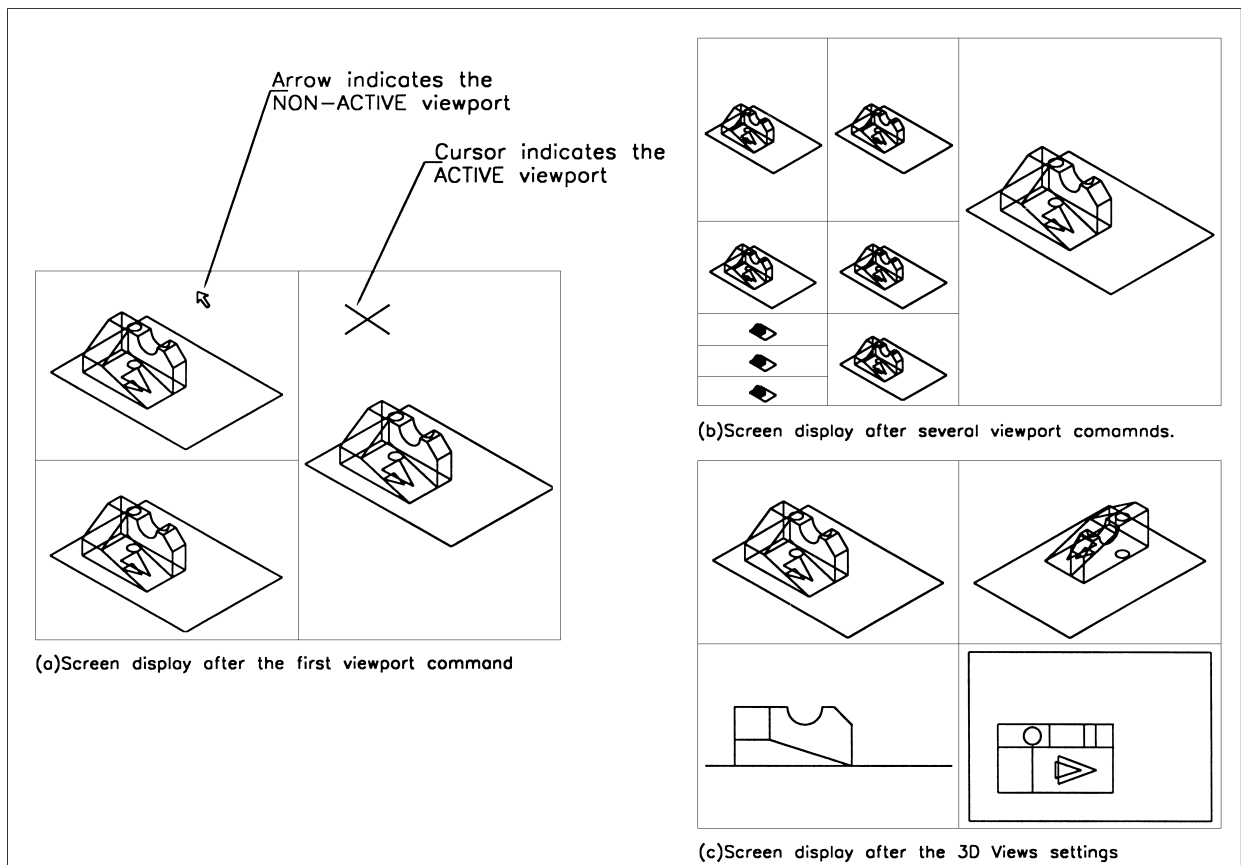


Figure 9.2 Viewport example 1.

- 7 Move the mouse about the screen and:
 - a) the large viewport will display the cursor cross-hairs and is the **ACTIVE** viewport, i.e. it is 'current'
 - b) the other viewports will display an arrow and these viewports are **NON-ACTIVE**
- 8 Any viewport can be made active by:
 - a) moving the mouse into the viewport area
 - b) left-click
 - c) try this for yourself a few times
- 9 At the command line enter **-VPORIS <R>** and:


```
prompt Enter an option [Save/Restore/Delete/..
enter   S <R> – the save option
prompt Enter name for new viewport configuration
enter   CONF1 <R>
```
- 10 Make the upper left viewport active and select the menu bar sequence **View-Viewports-2 Viewports** and:


```
prompt Enter a configuration option [Horizontal/Vertical..
enter   V <R> – the vertical option
and     the top left viewport will be further divided into two equal vertical viewports,
        each displaying the model layout
```
- 11 Make the lower left viewport active and menu bar with **View-Viewports-4 Viewports** to display an additional four viewports of the model
- 12 At the command line enter **-VPORIS <R>** and:


```
prompt Enter an option [Save/Restore/..
enter   S <R> – the save option
prompt Enter name for new viewport configuration
enter   CONF2 <R>
```
- 13 With the lower left viewport active, enter **-VPORIS <R>** at the command line and:


```
prompt Enter an option [Save/Restore..
enter   3 <R> – the 3 viewport option
prompt Enter a configuration option [Horizontal/Vertical/Above..
enter   H <R> – the horizontal option
```
- 14 The lower left viewport will be further divided into another three viewports and at this stage your screen should resemble Fig. 9.2(b)
- 15 Make the lower left viewport active and enter **-VPORIS <R>** then **4 <R>** and the following message will be displayed at the prompt line: *The current viewport is too small to divide*
- 16 Save the screen viewport configuration as CONF3 – easy?
- 17
 - a) make the large right viewport active
 - b) menu bar with View-Viewports-1 Viewport
 - c) original screen displayed?
 - d) zoom-all needed?
- 18 Menu bar with **View-Viewports-4 Viewports** to 'fill the screen' with four viewports of the model
- 19 Using the menu bar **View-3D Views** selection make each viewport current and set different viewpoints using the following:

<i>Viewport</i>	<i>3D View</i>
top left	SE Isometric
top right	NE Isometric
lower right	Plan-Current UCS
lower left	Front

- 20 The screen display should resemble Fig. 9.2(c)
- 21 Save the screen configuration as CONF4
- 22 *Task*
Restore the screen to a single viewport configuration to display the original model layout
- 23 At the command line enter **-VPOR**TS <R> and:
prompt Enter an option [Save/Restore..
enter **R** <R> – the restore option
prompt Enter name of viewport configuration to restore
enter **CONF1** <R>
and screen displays the first saved configuration
- 24 Restore the other three saved viewport configurations using the command line **-VPOR**TS, then restore the display to a single viewport
- 25 *Notes*
 a) The command line entry **-VPOR**TS gives the user the viewport options at the command line. This was deliberate for this first example.
 b) Generally the viewports command is activated from the menu bar in dialogue box form.
- 26 Menu bar with **View-Viewports-Named Viewports** and:
prompt Viewports dialogue box
with Named Viewports tab active
and four saved viewport configurations
respond **pick CONF3 then OK** – Fig. 9.3
- 27 The screen will display the named viewport configuration
- 28 Using the Named viewport dialogue box, display the other named named viewports then restore the model in the original single viewport as opened
- 29 This completes the first viewport exercise. If you want to save the exercise (with the viewport configurations) **DO NOT USE THE NAME 3DWFM**

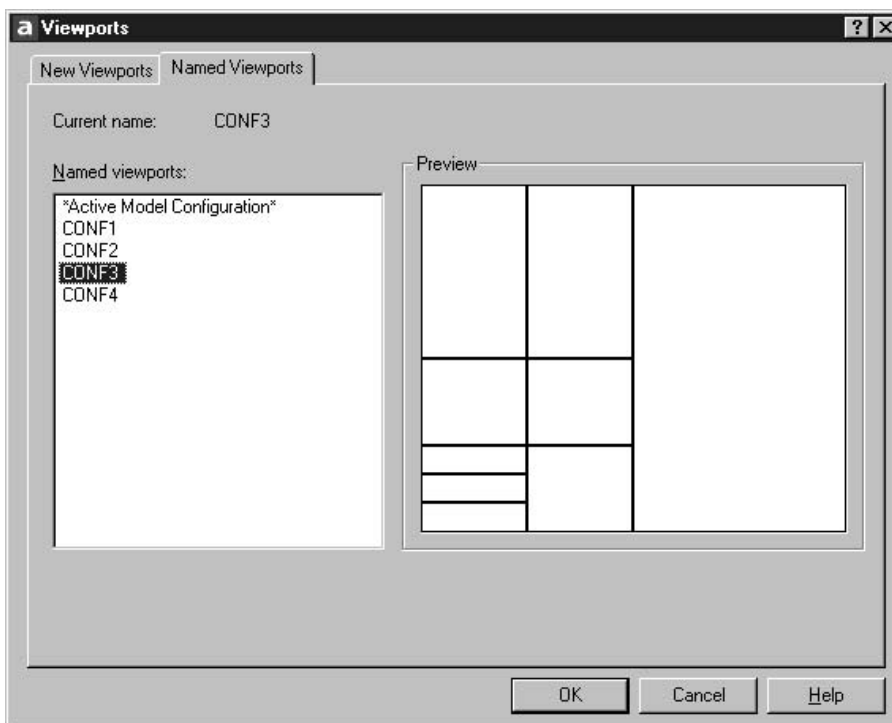


Figure 9.3 The Viewports (Named Viewports tab) dialogue box.

Example 2

The first exercise used an already created 3D model to investigate the viewport command and configurations. This current exercise will create a new 3D wire-frame model interactively using a four viewport configuration with preset 3D viewpoints. This will allow the user to 'see' the model being created in all four viewports at the one time.

- 1 Open your **3DSTDA3** template file to display the black border at a 3D viewpoint with layer MODEL current.
- 2 Menu bar with **View-Display-UCS Icon** and check both On and Origin are active (tick) – they should be!
- 3 Menu bar with **Tools-New UCS-Origin** and:
prompt Specify new origin point
enter **50,50,0 <R>**
and icon moves to the entered point and is displayed as a UCS icon
- 4 Save this UCS position as BASE
- 5 Menu bar with **View-Viewports-New Viewports** and:
prompt Viewports dialogue box with New Viewports tab active
respond
 1. New name: enter SCREEN DISPLAY 1
 2. Standard viewports: pick Four: Equal
 3. Apply to: Display
 4. Setup: scroll and pick 3D
 5. Change view to: do not alter (Fig. 9.4)
 6. pick OK

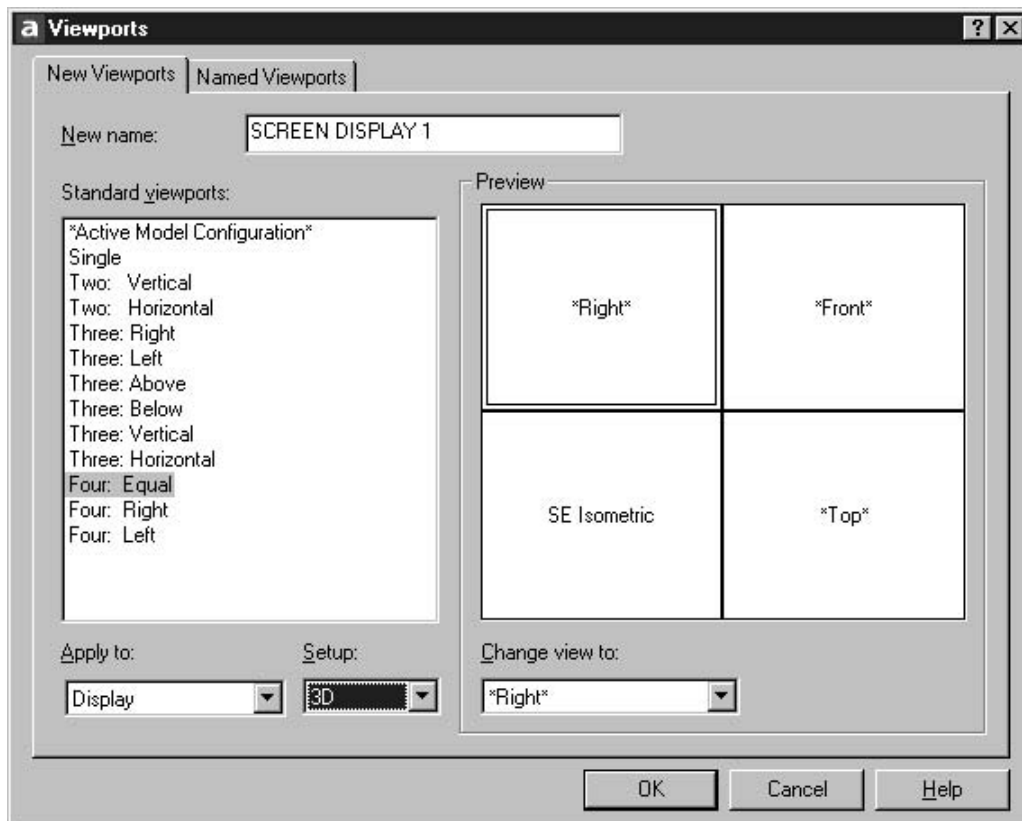


Figure 9.4 Viewports (New Viewports tab) dialogue box.

- 6 The screen will display a four viewport configuration with the black border displayed in each. Note the 'appearance' of the icon in the top two, and lower right viewports – it has the same configuration in each, despite the different viewpoints set in the New Viewports dialogue box (respond 4 above)
- 7 Making each viewport active in turn enter the following:
 - a) at the command line enter **UCSV $\langle R \rangle$** and:
prompt Enter new value for UCSVP<1>
enter **0 $\langle R \rangle$**
 - b) at the command line enter **ZOOM $\langle R \rangle$** then **0.9 $\langle R \rangle$**
- 8 The screen layout at this stage is similar to Fig. 9.5(a)
- 9 *Note:* we will discuss UCSVP in a later chapter
- 10 With the lower left viewport active, construct the model base using the LINE icon with with:

<i>Start point</i>	0,0,0 $\langle R \rangle$	pt1
<i>Next point</i>	@200,0,0 $\langle R \rangle$	pt2
<i>Next point</i>	@0,120,0 $\langle R \rangle$	pt3
<i>Next point</i>	@200<180,0 $\langle R \rangle$	pt4
<i>Next point</i>	close – fig(b) in 3D	
- 11 Using the LINE command construct the front vertical side with:

<i>Start point</i>	Intersection icon of pt1	
<i>Next point</i>	@20,0,100 $\langle R \rangle$	pt5
<i>Next point</i>	@120,0,0 $\langle R \rangle$	pt6
<i>Next point</i>	Intersection icon of pt2	
<i>Next point</i>	right-click/enter – fig(c) in 3D	

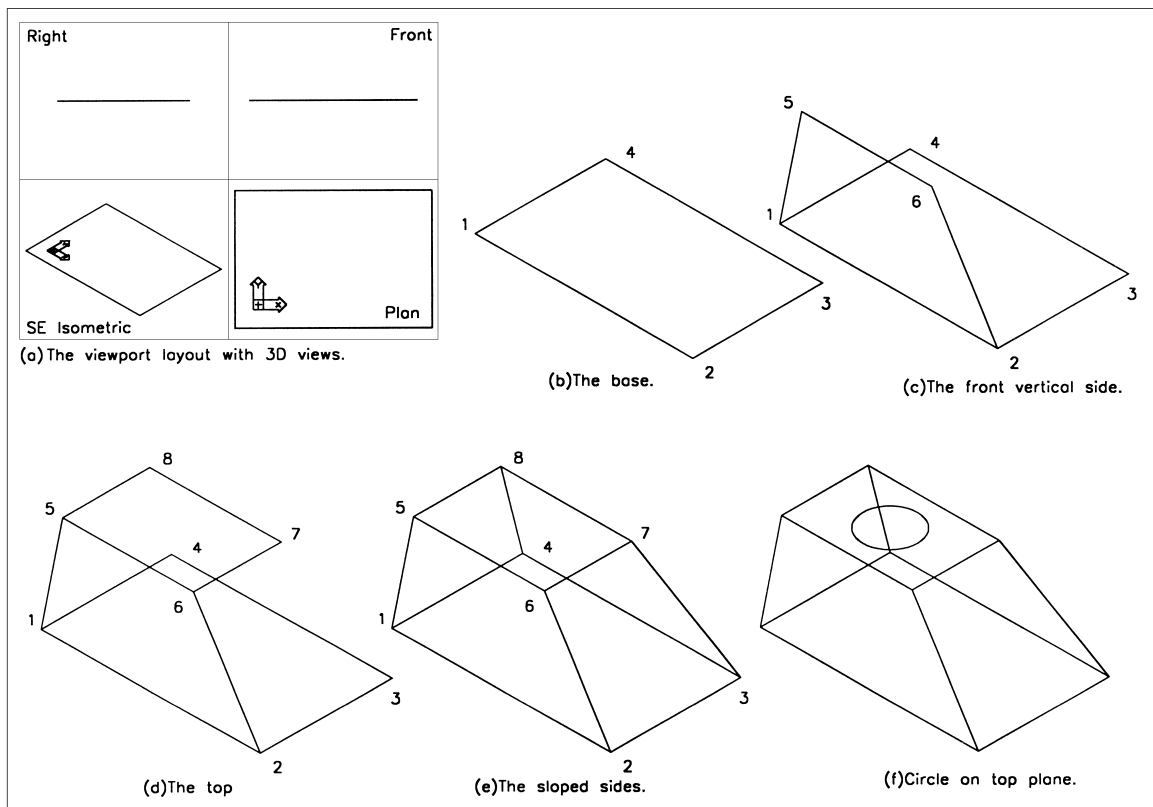


Figure 9.5 Construction of model for viewport example 2.

- 12 The top surface is created with the LINE command and:
- | | | |
|--------------------|---------------------------------|----------------|
| <i>Start point</i> | Intersection icon of pt6 | |
| <i>Next point</i> | @0,80,0 <R> | pt7 |
| <i>Next point</i> | @-120,0,0 <R> | pt8 |
| <i>Next point</i> | Intersection icon of pt5 | |
| <i>Next point</i> | right-click/enter | – fig(d) in 3D |
- 13 Add the sloped sides with lines joining points 3–7 and 4–8 as fig(e) in 3D
- 14 Make layer OBJECTS (blue) current and draw a circle with centre at 80,40,100 and with a radius of 25 – fig(f) in 3D
- 15 Menu bar with **Draw-Surfaces-3D Surfaces-Box3d** and:
- | | |
|---------------|---|
| <i>prompt</i> | Specify corner of box and enter: 80,30,0 |
| <i>prompt</i> | Specify length of box and enter: 50 |
| <i>prompt</i> | Specify width of box and enter: 40 |
| <i>prompt</i> | Specify height of box and enter: 30 |
| <i>prompt</i> | Specify rotation angle of box about Z axis and enter: 20 |
- 16 a) Make layer TEXT current
 b) Rotate UCS about X axis by 90 and save as FRONT
 c) Menu bar with **Draw-Text-Single Line Text** and add the text item AutoCAD, centred on 80,50 with height 20 and rotation 0
- 17 a) Set a 3 point UCS on the right sloped surface with:
1. origin: midpoint of line 23 – Fig. 9.5(e)
 2. x axis: intersection of pt 3
 3. y axis: perpendicular to line 67
 4. save UCS as SLOPE
- b) Add the single line text item R2002, centred on -5,50 with a height of 15 and a rotation angle of 0
- 18 The complete four viewport configuration display should be similar to Fig. 9.6
- 19 Save the drawing as **MODR2002\TEST3D**
- 20 This completes the two exercises on viewports
- 21 Notes
1. A new system variable was used during this exercise, this being **UCSVP**. This variable determines whether the UCS in an active viewport will 'reflect' the UCS orientation of that active viewport and:
 - a) UCSVP 0: unlocked, i.e. the UCS will reflect the UCS of the current active viewport
 - b) UCSVP 1: locked, i.e. UCS is independent of the UCS in the current active viewport
 2. The default UCSVP value is 1, i.e. locked
 3. It is **recommended that the UCSVP is set to 0**, i.e. it always reflects the UCS position in any active viewport
 4. The UCSVP must be set in every created viewport

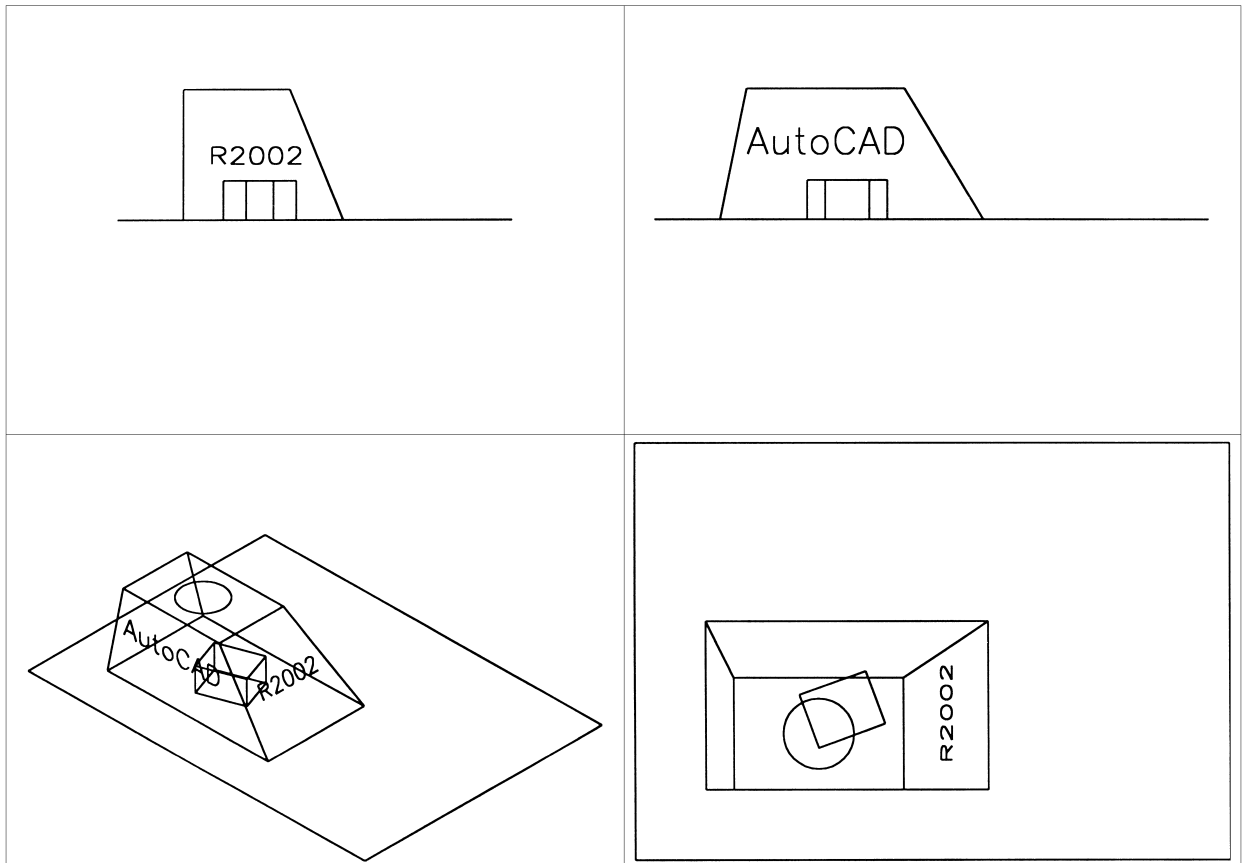


Figure 9.6 Completed viewport example 2 – TEST3D.

Summary

- 1 Viewports allow multi-screen configurations to be set
- 2 There are two types of viewport – TILED and UNTILED
- 3 The viewport type is controlled by the system variable TILEMODE and:
 - a) TILEMODE 1: tiled viewports (fixed)
 - b) TILEMODE 0: untiled viewports (movable) – more later
- 4 Tiled viewports can have between 1 and 4 ‘divisions’ and ‘fill the screen drawing area’
- 5 Multi-screen viewports are generally used with the viewpoint command and their full benefit will not be appreciated until the various viewpoint options are discussed
- 6 Multiple viewport layouts are essential to 3D modelling.

3D views (Viewpoint)

3D Views (or viewpoints) determine how the user 'looks' at a model and has been used in previous chapters without any discussion about how it is used. In this chapter we will investigate the command in detail using previously created models. When combined with viewports, the user has a very powerful draughting aid – multiple viewports displaying different views of a model.

The viewpoint command has the following selection options:

- a) Isometric views: SW, SE, NE, NW
- b) Orthographic views: Top, Bottom, Left, Right, Front, Back
- c) Plan view: to current UCS, WCS, named UCS
- d) Viewpoint: with rotate, compass and tripod, vector options
- e) Viewpoint Presets: dialogue box selection
- f) Real-time 3D rotation
- g) New Viewports dialogue box

In this chapter we will investigate all of the above selections.

The Viewpoint ROTATE option

This option requires two angles to be entered by the user:

- a) the angle in XY plane from the X-axis – the **view** direction
- b) the angle from XY plane – the **inclination (tilt)**

- 1 Open your MODR2002\3DWFM drawing and:
 - a) erase any dimensions and hatching.
 - b) leave all text items – they will act as a 'reference' as the model is viewed from different angles.
- 2 Layer MODEL current, UCS BASE and SE Isometric viewpoint
- 3 Refer to Fig. 10.1 section A
- 4 At the command line enter **VPOINT <R>** and:

```
prompt ***Switching to WCS***
and Current view direction
then Specify a view point or [Rotate]<display compass and
tripod>
enter R <R> – the rotate option
prompt Enter angle in XY plane from X-axis
enter 40 <R>
prompt Enter angle from XY plane
enter 0 <R>
prompt ***Returning to UCS***
then Regenerating drawing
and model displayed as fig(al), i.e. looking towards the right-rear side from a
horizontal 'stand-point' – the view direction.
```

- 5 At the command line enter **VPOINT <R>** and:
- ```
prompt Specify a view point or [Rotate]
enter R <R> – the rotate option
prompt Enter angle in XY plane from X-axis and enter: 90 <R>
prompt Enter angle from XY plane and enter: 0 <R>
and model displayed as fig(a2)
```
- 6 Repeat the VPOINT-Rotate command from the command line and enter the following angle values at the prompts:
- ```
prompt 1   prompt 2   fig
215       0           a3
330       0           a4
```
- 7 Restore the original SE Isometric viewpoint and refer to Fig. 10.1 section B
- 8 Use the VPOINT-Rotate command and enter the following angles at the prompts:
- ```
prompt 1 prompt 2 fig
0 45 b1
0 135 b2
0 270 (-90) b3
0 -45 (315) b4
```
- 9 Restore the SE Isometric viewpoint and refer to Fig. 10.1 section C. Activate the VPOINT-Rotate command and enter the following angles:
- ```
prompt 1   prompt 2   fig
40         70         c1
40         -30        c2
-20        20         c3
```
- 10 Restore the original SE Isometric viewpoint
- 11 *Task*
Restore some other UCS settings, e.g. SLOPE1, VERT1, etc. and repeat the viewpoint rotate command using some of the above angle entries. The model display should be unaffected by the UCS position Think about the prompt *****Switching to the WCS*****
- 12 *Explanation of option*
- a) **Prompt 1: angle in XY plane from the X-axis**
This is the viewer's standpoint **on the XY horizontal plane** looking towards the model, i.e. it is your view direction. If this angle is 0degs you are looking at the model from the right side. If the angle is 270degs you are looking onto the front of the model. The value of this angle can be between 0degs and 360degs. It can also be positive or negative and remember that 270degs is the same as -90degs
- b) **Prompt 2: angle from the XY plane**
This is the viewer's 'head inclination' looking at the model, i.e. it is the **angle of tilt**. A 0degs value means that you are looking at the model horizontally and a 90degs value is looking vertically down. The angle of tilt can vary between 0degs and 360degs and be positive or negative and: positive tilt: looking down on the model negative tilt: looking up at the model
- 13 *Note*
The reader must realise that the displays in Fig. 10.1 have been 'scaled' to fit the one sheet, and that your model displays will be larger than those illustrated.

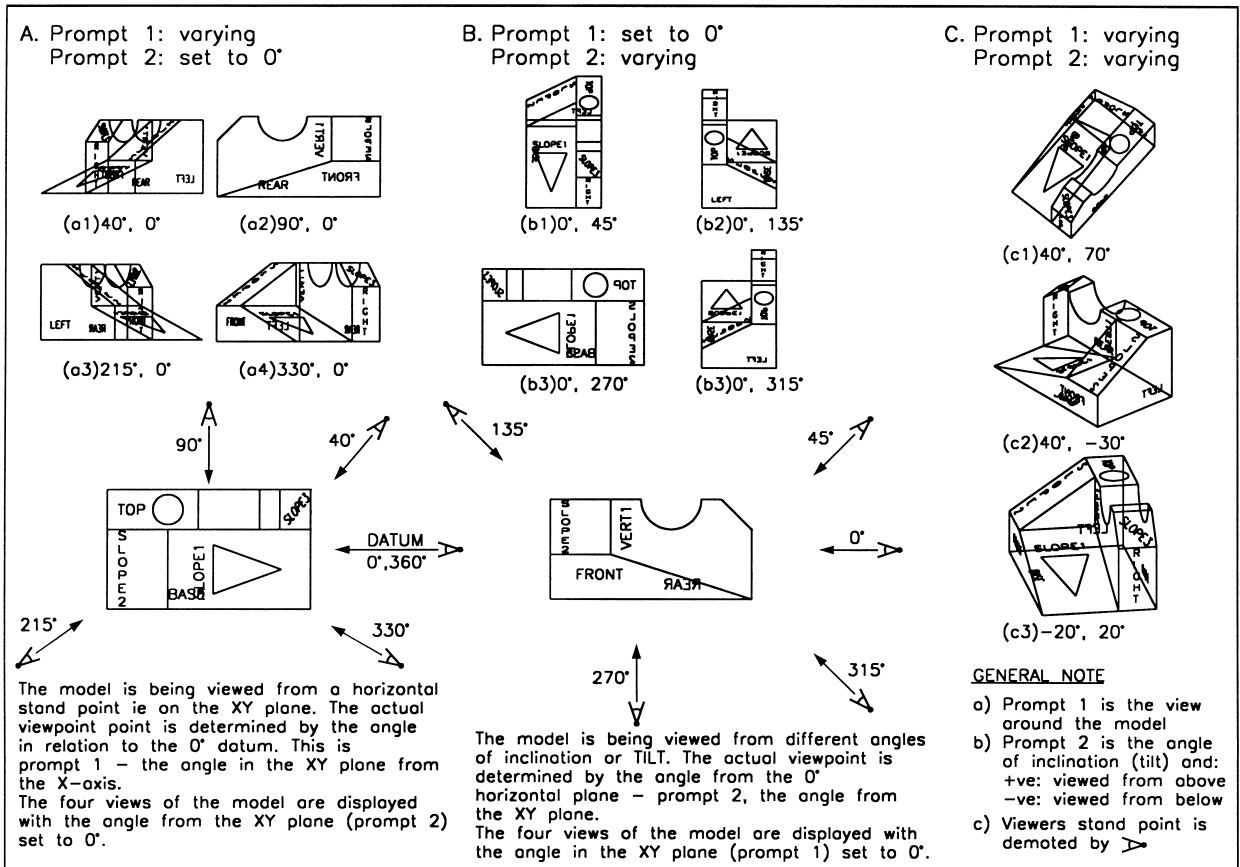


Figure 10.1 3D Views – the VPOINT ROTATE option with 3DWF.

VPOINT ROTATE using the presets dialogue box

- 1 3DWF displayed at SE Isometric setting with UCS BASE?
- 2 Menu bar with **View-3D Views-Viewpoint Presets** and:
prompt Viewpoint Presets dialogue box – Fig. 10.2
 with
 1. viewing angle: absolute to WCS
 2. angle from X-axis: 315 – left-hand ‘clock’
 3. angle from XY plane: 35.3 – right-hand ‘arc’
- 3 This dialogue box allows:
 - a) viewing angle to be absolute to WCS or relative to UCS
 - b) angles to be set by selecting circle/arc position
 - c) angles to be set by altering values at **From:** line
 - d) plan views to be set
- 4 Respond to the dialogue box with:
 - a) do not change to absolute to WCS
 - b) change the X-axis angle from 315 to 150
 - c) change the XY plane angle from 35.3 to 10
 - d) pick OK
 - e) the model will be displayed at the entered viewpoint angles
- 5 Make UCS SLOPE1 current

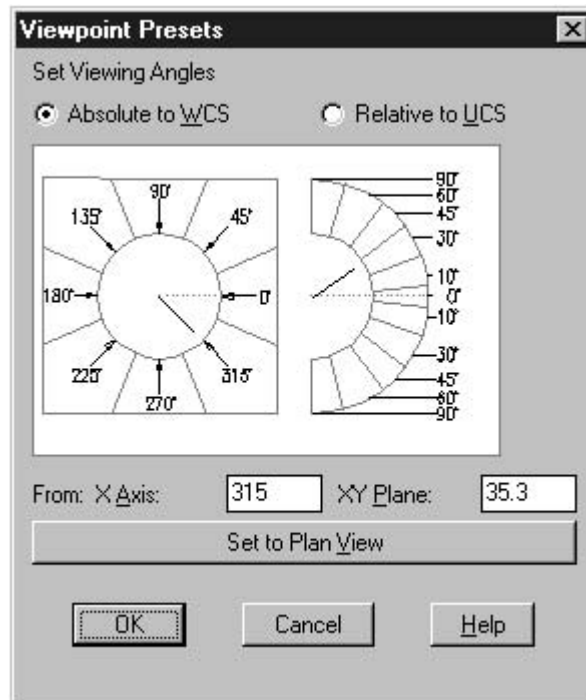


Figure 10.2 The Viewpoint Presets dialogue box.

- 6 Menu bar with View-3D Views-Viewpoint Presets and:
 - a) make Relative to UCS active – black dot
 - b) leave the two angle values as 150 and 10
 - c) pick OK
 - d) the model is displayed at the entered viewpoint angles but differs from the step 4 display due to the UCS setting
- 7 *Task*
 - a) Try some other entries from the Viewpoint Presets dialogue box using both selection methods, i.e. the clock/arc and altering the angles.
 - b) Investigate the difference in the display with the Absolute to WCS and Relative to UCS selections
 - c) Restore Absolute to WCS
 - d) Restore UCS BASE and the SE Isometric viewpoint
- 8 This completes the viewpoint rotate exercise. Do not save any changes to the 3DWFM model.

The Viewpoint COMPASS and TRIPOD option

This option allows the user to set 'infinite viewpoints' and has been called the bulls-eye and target method – for obvious reasons. We will demonstrate the command with a different model so:

- 1 Open the MODR2002\TEST3D model created during the viewport exercise and refer to Fig. 10.3
- 2 Ensure UCS BASE is current and make the lower left viewport active, i.e. the 3D viewport
- 3 Menu bar with View-Viewports-1 Viewport to display a single viewport of the model at a 3D viewpoint. This model 'fills the screen'.
- 4 Menu bar with **View-3D Views-Viewpoint** and:
 - prompt* 1. model 'disappears'
 - 2. screen displays the XYZ tripod and the compass, which was previously termed the bulls-eye target
 - 3. cursor replaced by a small cross (+)
 - 4. axes and cross(+) move as mouse is moved
 - respond* move the cross (+) into the circle quadrant indicated in Fig. 10.3(a) and left-click
 - and* model displayed at this viewpoint, and is viewed from above

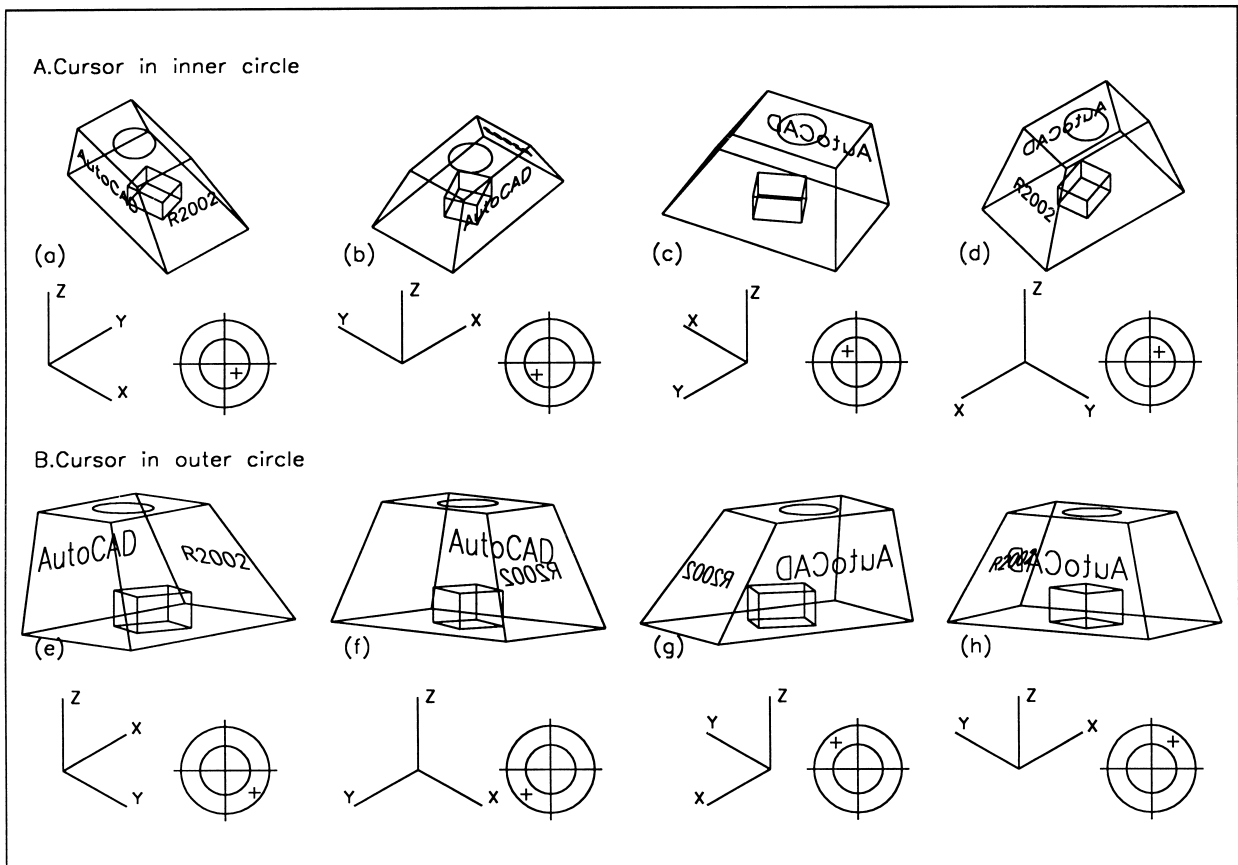


Figure 10.3 3D Views – the Viewpoint compass and tripod option with TEST3D.

- 5 At the command line enter **VPOINT <R>** and:
- ```

prompt ***Switching to WCS***
then Specify a viewpoint or [Rotate]<display compass and tripod>
respond press <RETURN>
prompt tripod and compass displayed
respond move the cross (+) into the circle quadrant indicated in Fig. 10.3(b)
then left-click
and model displayed at this new viewpoint

```
- 6 Repeat the tripod viewpoint option (menu bar or command line) and position the cross (+) in the quadrants indicated in Fig. 10.3  
i.e. (c)–(d): within the inner circle  
(e)–(h): between the inner and outer circles
- 7 *Task*  
When you are capable of using the compass and tripod, try the following:  
a) position the + at different points on the two axes and observe the resultant displays  
b) position the + at different points on the circle circumferences and note the displays
- 8 *Explanation of option*  
a) The 'bulls-eye' is in reality a representation of a glassglobe and the model is located at the centre of the globe. The XY plane is positioned at the equator. The north pole of the globe is at the circle centres and the two concentric circles represent the surface of the world, stretched out onto a flat plane with:  
circle centre : the north pole  
inner circle : the equator  
outer circle : the south pole
- b) As the cross(+) is moved about the circles, the user is moving around the surfaces of the globe and:
- | <i>Cross (+) position</i> | <i>View result</i>                   |
|---------------------------|--------------------------------------|
| in inner circle           | above equator, looking down on model |
| in outer circle           | below equator, looking up at model   |
| on inner circle           | looking horizontally at model        |
| below horizontal          | viewing from the front               |
| above horizontal          | viewing from the rear                |
- 9 This completes the tripod option exercise. Do not save changes.

## The Viewpoint VECTOR option

- 1 Open MODR2002\3DWFM with UCS BASE and SE Isometric viewpoint.
- 2 Erase any dimensions and hatching, but leave the text items as they will act as a 'reference' as the model viewpoint is altered.
- 3 Refer to Fig. 10.4
- 4 Menu bar with **View-3D Views-Viewpoint** and:  
*prompt* \*\*\*Switching to WCS\*\*\*  
*then* Current view direction  
*and* Specify a view point or [Rotate]  
*enter* **0,0,1 <R>**  
*prompt* \*\*\*Returning to the UCS\*\*\*  
*and*
  1. the model will be displayed at the entered viewpoint
  2. it is a top view – fig(a)
  3. it 'fills the screen'
- 6 At the command line enter **VPOINT <R>** and:  
*prompt* Specify a view point or [Rotate]  
*enter* **0,-1,0 <R>**  
*and* model displayed as fig(b) – a front view

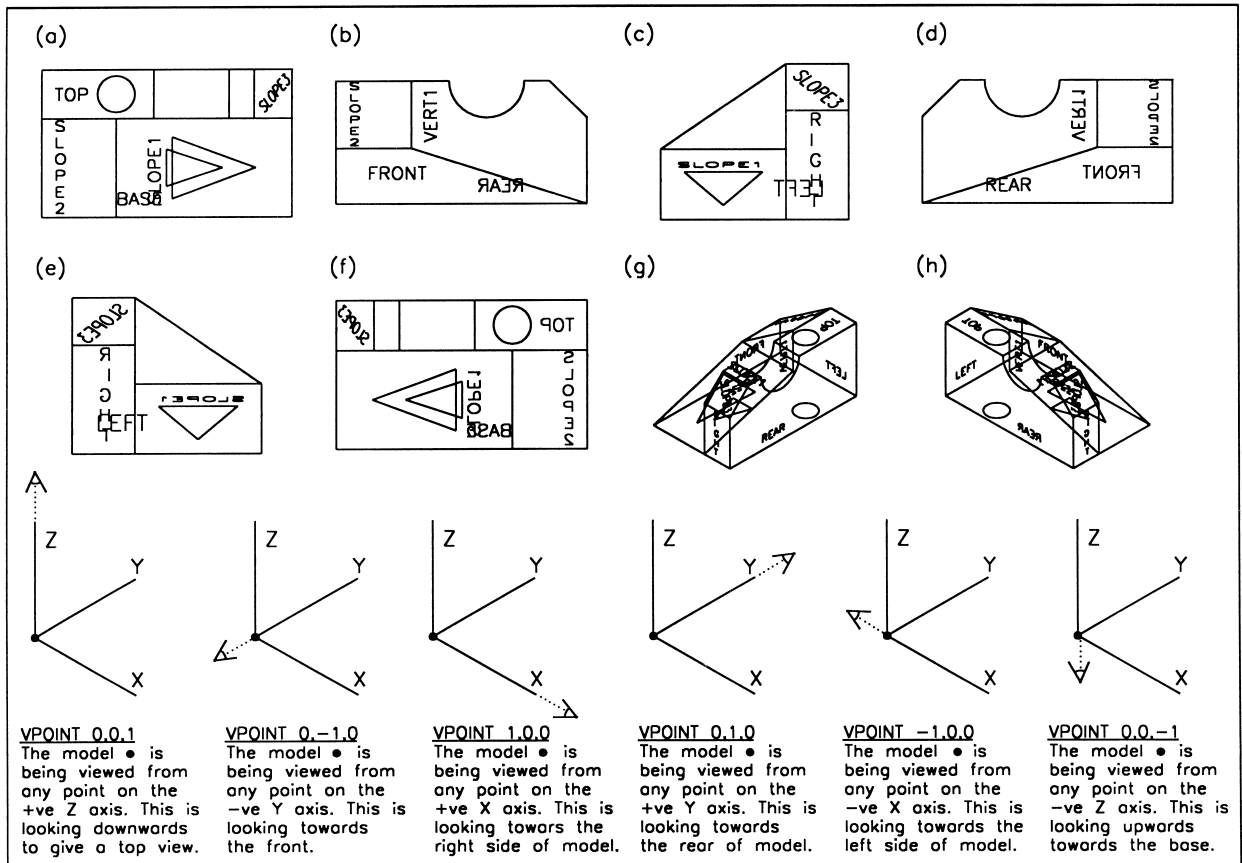


Figure 10.4 3D Views – the Viewpoint VECTOR option with 3DWFM.

- 7 Repeat the viewpoint vector option (menu bar or command line) and enter the following coordinates at the prompt line:

| <i>co-ords</i> | <i>view</i>   | <i>fig</i> |
|----------------|---------------|------------|
| 1,0,0          | from right    | c          |
| 0,1,0          | from rear     | d          |
| -1,0,0         | from left     | e          |
| 0,0,-1         | from below    | f          |
| 1,1,1          | 3D from above | g          |
| -1,-1,-1       | 3D from below | h          |

- 8 Restore the original SE Isometric viewpoint

- 9 Task

Try some vector entries for yourself then restore the original SE Isometric viewpoint

- 10 *Explanation of option*

a) The vector option allows the user to enter x,y,z coordinates. These are the coordinates of the viewers 'stand point' looking at the model which is considered to be at the origin. Thus if you enter 0,0,1 you are 'standing' at the point 0,0,1 looking towards the origin. As this point is on the positive Z axis you are looking down on the model, i.e. a top view.

b) The actual numerical value of the vector entered does not matter, i.e. 0,0,1; 0,0,12; 0,0,99.99; 0,0,3456 are all the same viewpoint entries. I prefer to use the number 1, hence 0,0,1; -1,0,0 etc

c) Certain vector entries give the same display as rotate options and the following lists some of these similarities:

| <i>vector</i> | <i>rotate</i> | <i>view</i>   |
|---------------|---------------|---------------|
| 0,0,1         | 0,90          | top           |
| 0,-1,0        | 270,0         | front         |
| 1,0,0         | 0,0           | right         |
| 0,1,0         | 90,0          | rear          |
| -1,0,0        | 180,0         | left          |
| 0,0,-1        | 0,-90         | bottom        |
| 1,1,1         | 45,35         | 3D from above |
| -1,-1,-1      | -135,-35      | 3D from below |

- 11 This completes the vector option. Do not save any changes.

## The Isometric viewpoints

The isometric 3D views allow the user to view a model from four 'preset' viewpoints, these being SW, SE, NE and NW. These four viewpoints are used extensively as they allow easy access to viewing a model in 3D.

- 1 Open model TEST3D to display the four viewport configuration saved from the previous chapter.
- 2 Restore UCS BASE with layer MODEL current
- 3 Making the appropriate viewport active, menu bar with Views-3Dviews and set the following viewpoints:

| <i>viewport</i> | <i>viewpoint</i> |
|-----------------|------------------|
| top left        | SW Isometric     |
| top right       | SE Isometric     |
| lower right     | NE Isometric     |
| lower left      | NW Isometric     |

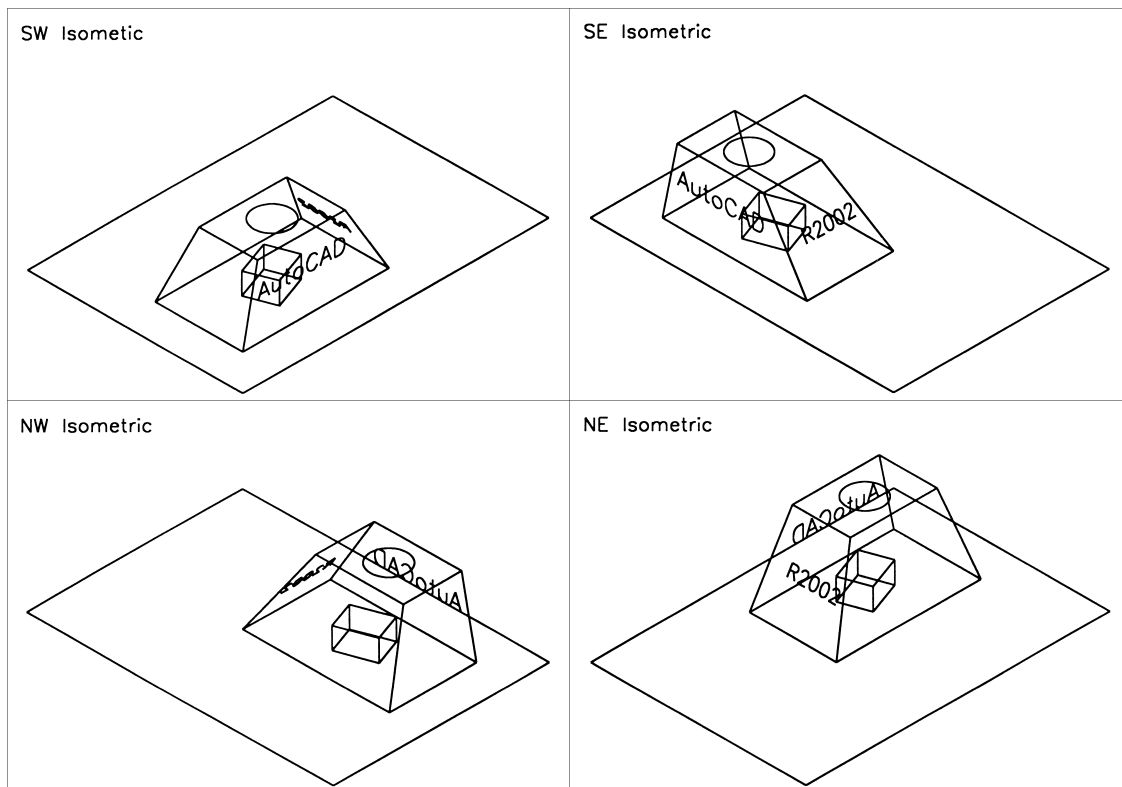
- 4 When the viewpoints have been entered, zoom-all in each viewport and the result should be Fig. 10.5. This does not need to be saved

- 5 *Notes:*

a) The four preset isometric viewpoints only allow viewing from above. If a model is to be viewed from below, another option is required. My choice for this is VPOINT-Rotate with a negative second angle value.

b) The equivalent VPOINT-Rotate angles for the four isometric presets are:

| <i>3D View</i> | <i>angle in XY plane</i> | <i>angle from XY plane</i> |
|----------------|--------------------------|----------------------------|
| SW Isometric   | 225                      | 35.3                       |
| SE Isometric   | 315                      | 35.3                       |
| NE Isometric   | 45                       | 35.3                       |
| NW Isometric   | 135                      | 35.3                       |

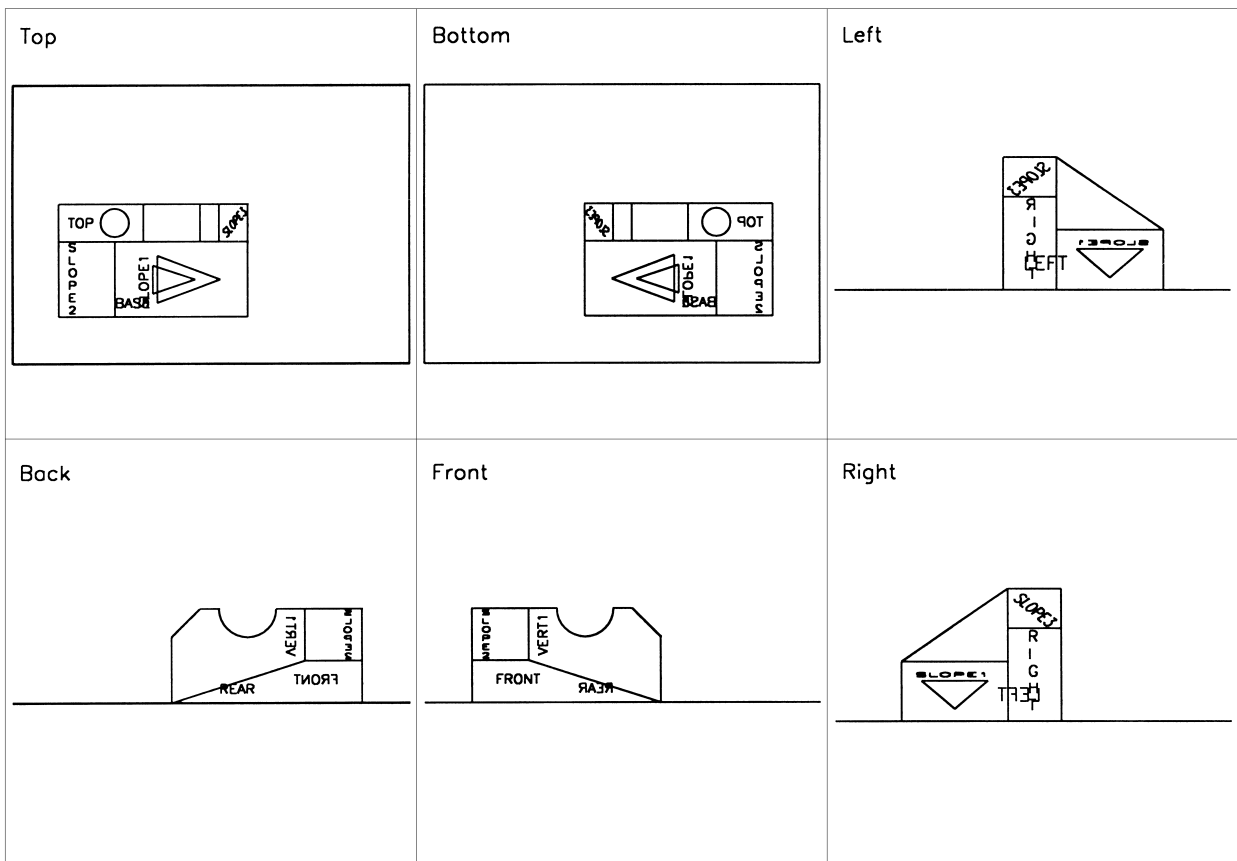


**Figure 10.5** 3D Views – the ISOMETRIC Presnts with the TEST3D model.

## The Orthographic viewpoints

There are six 'preset' orthographic viewpoints these being Top, Bottom, Left, Right, Front and Back. The options are independent of the UCS position.

- 1 Open model 3DWF1 and erase any dimensions and hatching
- 2 Restore UCS BASE with layer MODEL current. Refer to Fig. 10.6
- 3 Menu bar with **View-Viewports-2 Viewports** and:  
*prompt* Enter a configuration option and enter: **H <R>**
- 4 With the top viewport active, menu bar with **View-Viewports-3 Viewports** and:  
*prompt* Enter a configuration option and enter: **V <R>**
- 5 With the bottom viewport active, repeat step 4
- 6 Making each viewport active, menu bar with **View-3D Views** and set the following orthographic viewpoints:  
*viewport*                      *viewpoint*  
top left                      top  
top middle                      bottom  
top right                      left  
lower right                      right  
lower middle                      front  
lower right                      back
- 7 This layout does not need to be saved.



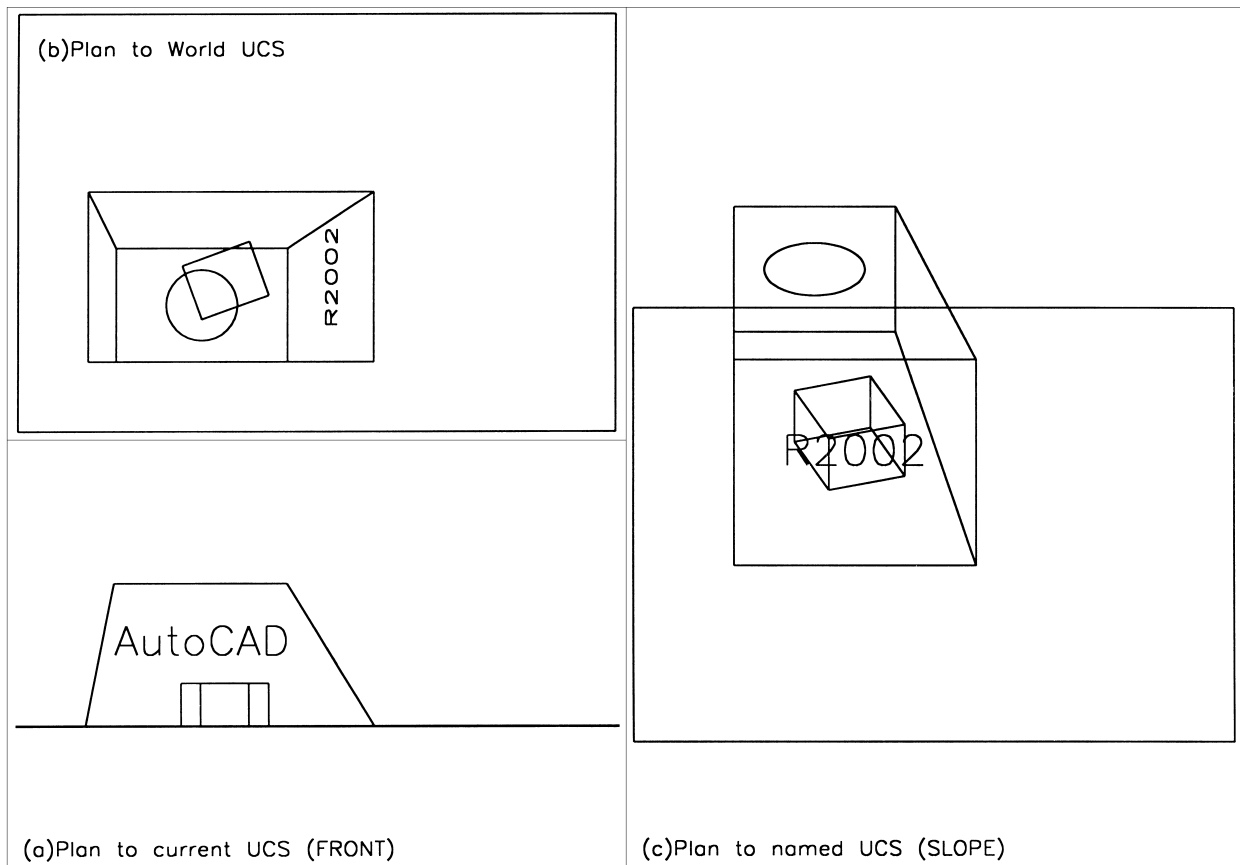
**Figure 10.6** 3D Views – the six orthographic presets with the 3DWF1 model.

## Viewpoint PLAN

This viewpoint selection was discussed during the chapter on the UCS and gives a view perpendicular to a UCS position.

- 1 Open the four viewport configuration TEST3D and:
  - a) create a single viewport configuration of the 3D view
  - b) set any 3 viewport configuration
  - c) UCS FRONT current and refer to Fig. 10.7
- 2 Make any viewport active and menu bar with **View-3D Views-Plan View-Current UCS** and the model will be displayed as fig(a).
- 3 It is a plan view perpendicular to the UCS FRONT XY plane.
- 4 With another viewport active, menu bar with View-3D Views-Plan **View-World UCS** and the model will be displayed as fig(b)
- 5 Make the third viewport active and select the menu bar sequence **View-3D Views-Plan View-Named UCS** and:
 

*prompt* Enter name of UCS and enter: **SLOPE <R>**
- 6 The model will be displayed as a view perpendicular to the XY plane of UCS SLOPE.
- 7 This layout does not need to be saved.



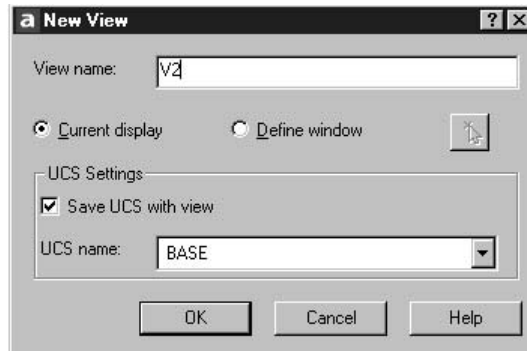
**Figure 10.7** 3D Views – the PLAN options with model TEST3D.

## The VIEW command

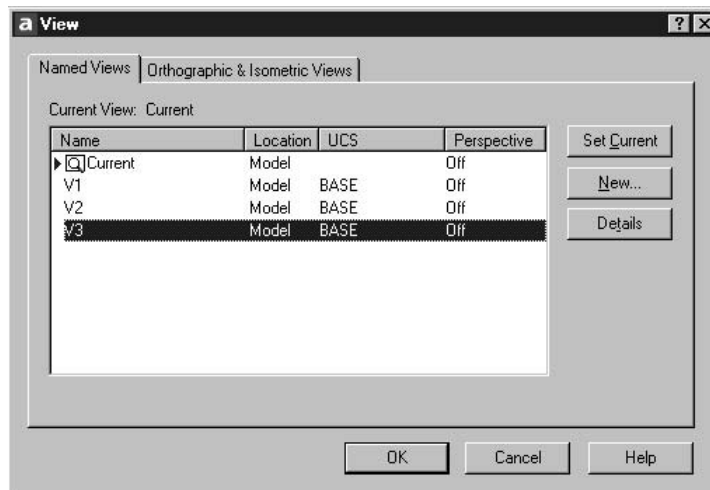
Different views of a model can be saved within the current drawing, thus allowing the operator to create a series of 'pictures'. These could be of the model being constructed, of a completed model at differing viewpoints etc. These views (pictures) can be recalled at any time.

- 1 Open the 3DWFM model and erase any dimensions and hatching (or freeze the appropriate layers)
- 2 Model displayed at SE Isometric viewpoint with UCS BASE and layer MODEL current
- 3 At the command line enter **-VIEW <R>** and:
  - prompt* Enter an option [?/Orthographic/Delete/Restore/Save..
  - enter* **S <R>** – the save option
  - prompt* Enter view name to save
  - enter* **VI <R>**
- 4 Menu bar with View-3D Views and set to NW Isometric
- 5 Menu bar with **View-Named Views** and:
  - prompt* View dialogue box – Named Views tab active
  - respond* **pick New**
  - prompt* New View dialogue box
  - respond*
    1. enter View name: V2
    2. ensure Current display active, i.e. black dot
    3. Save UCS with view active (tick)
    4. UCS name: BASE – Fig. 10.8
    5. pick OK
  - prompt* View dialogue box with V2 added to list
  - respond* **pick OK**
- 6 Menu bar with View-3D Views and set to SW Isometric
- 7 Menu bar with **View-Named Views** and from the View dialogue box:
  - a) pick New
  - b) View name: V3 with current display, UCS BASE then OK
  - c) View dialogue box as Fig. 10.9
  - d) pick OK
- 8 At the command line enter **-VIEW <R>** and:
  - prompt* Enter an option
  - enter* **R <R>** – the restore option
  - prompt* Enter view name to restore
  - enter* **VI <R>**
  - and* saved view V1 displayed
- 9 Menu bar with **View-Named Views** and:
  - prompt* View dialogue box
  - respond*
    1. pick V3
    2. pick Set Current
    3. pick OK
  - and* screen displays the saved V3 view
- 10 *Task*
  - a) create, save and display some other views
  - b) investigate the Details option from the Named Views tab
  - c) investigate the Orthographic & Isometric Views tab
  - d) investigate altering the UCS with saved views
- 11 When complete, restore the SE Isometric viewpoint. Save if required but not as 3DWFM.
- 12 *Note*

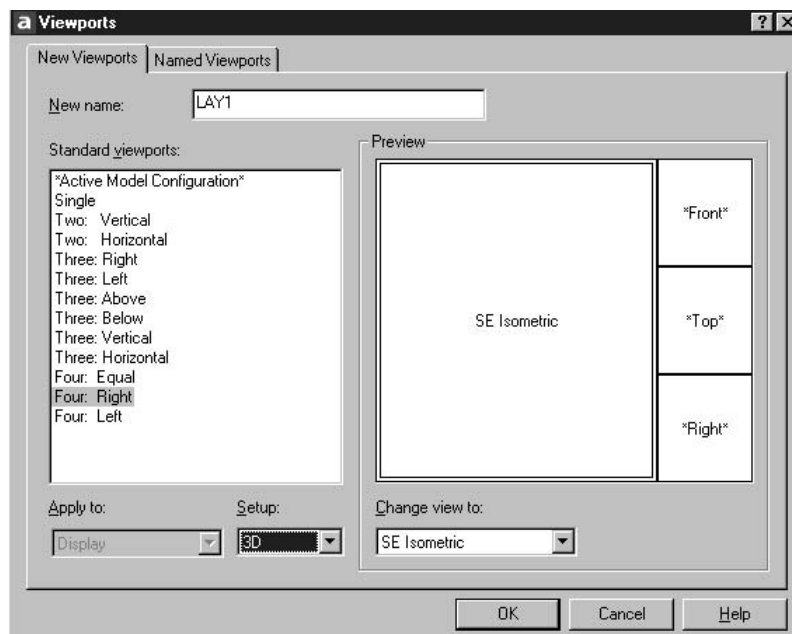
Do not confuse the VIEW command with the View option of the UCS command. They are two entirely different concepts.



**Figure 10.8** The New View dialog box.



**Figure 10.9** The View dialog box (Named Views tab).

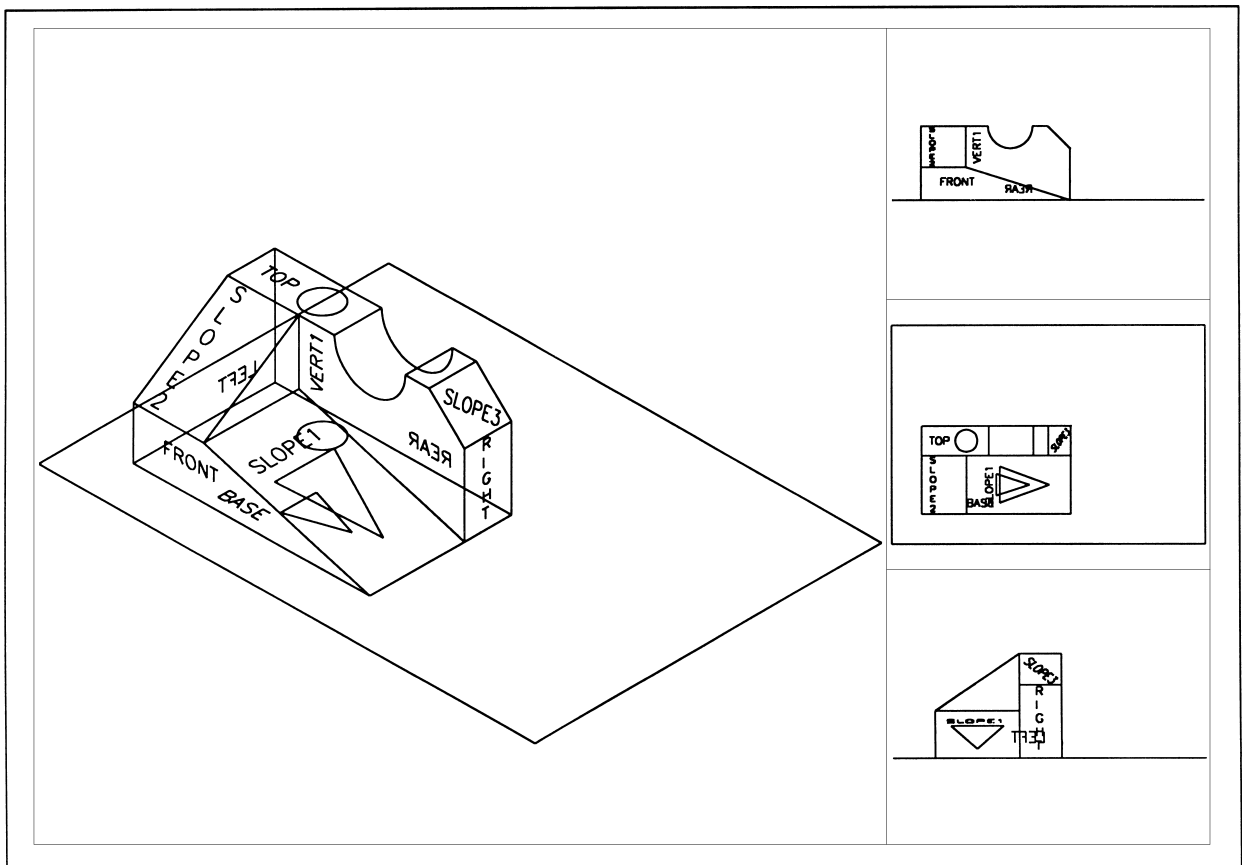


**Figure 10.10** The Viewports dialog box (New Viewports tab).

## Viewports dialogue box

Viewpoints can be set using the various options of the 3D View command as well as from the Viewports dialogue box. To investigate the Viewports dialogue box further:

- 1 Open the 3DWFMM model – no dimensions or hatching
- 2 UCS BASE and layer MODEL both current
- 3 Menu bar with **View-Viewports-Named Viewports** and:  
*prompt* Viewport dialogue box  
*respond*
  1. pick the New Viewports tab
  2. enter View name: **LAY1**
  3. pick Standard viewports: **Four: Right**
  4. Setup: 3D
  5. note Preview
  6. leave view names as given – Fig. 10.10
  7. pick OK
- 4 The screen will display a four viewport configuration with the appropriate views of the model as Fig. 10.11. This is easier than creating a viewport layout then setting the viewpoint?



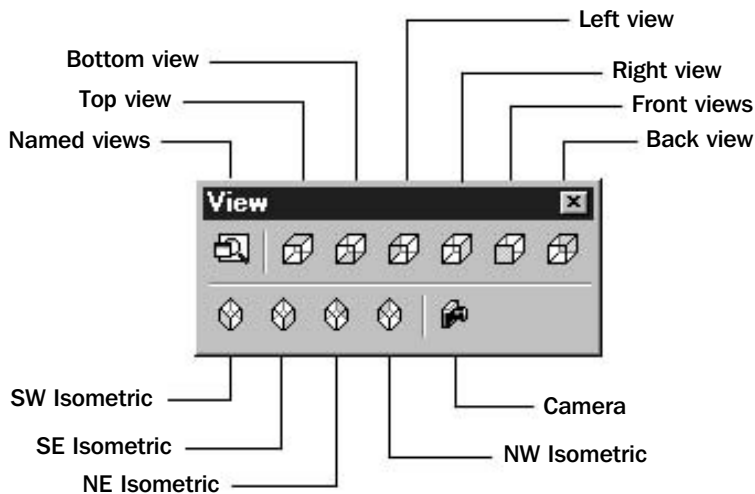
**Figure 10.11** 3DWFMM using the Viewports dialogue box (New Viewports tab).

- 5 Repeat the menu bar sequence View-Viewports-Named Viewports and:
  - a) pick the New Viewport tab
  - b) enter View name as LAY2
  - c) pick a Four Left standard viewport
  - d) apply to display with 3D setup
  - e) pick OK
  - f) screen displays the same four views of the model as before, but in a different viewport configuration.
- 6 Now restore the original single SE isometric viewport layout
- 7 This completes this exercise

## The View toolbar

All commands have so far been activated from the command line or the menu bar. The view toolbar has several icons which can be used to obtain several viewpoints of a model. The toolbar is displayed in Fig. 10.12 and it is user preference as to which method is used.

The view toolbar can be activated from the menu bar with **View-Toolbars** and 'ticking' the View toolbar. The Toolbars dialogue box can then be closed, and the View toolbar positioned to suit.



**Figure 10.12** The View toolbar.

## Centring models in viewports

When 3D models are displayed in multiple viewport configurations, three 'problems' may initially occur:

- a) the model may 'fill the viewport'
- b) the model may be displayed at different sizes in the viewports
- c) the model views may not 'line up' between viewports

These 'problems' are easily overcome by zooming each viewport by a scale factor or about a specified centre point determined by the user, who then decides on the 'scale effect' in the viewports. We will demonstrate the concept with two previously created models.

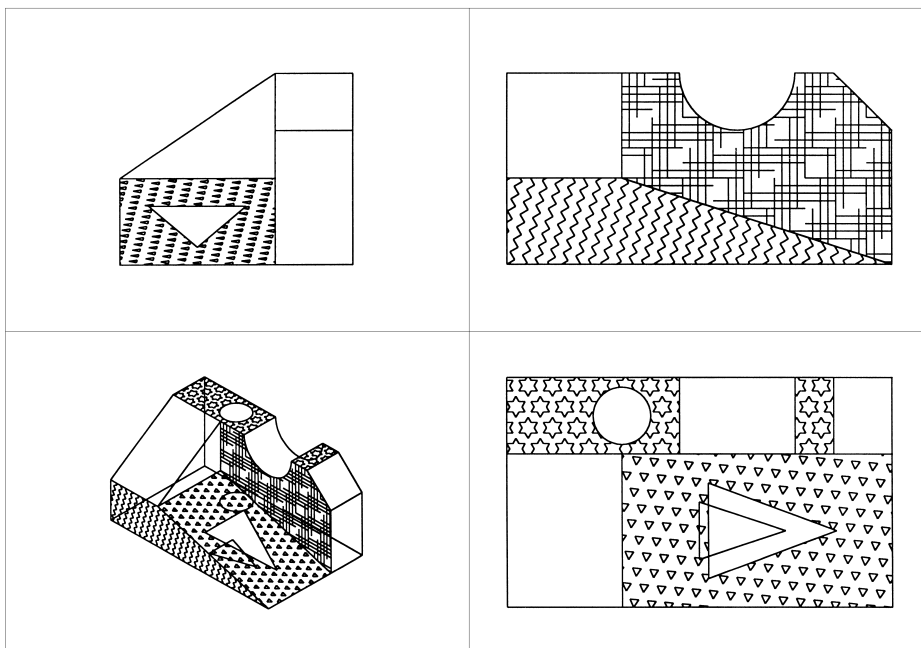
## Example 1 – centring by scale

- 1 Open MODR2002\3DWF3 of the wire-frame model and:
  - a) erase/freeze any dimensions and text
  - b) leave the hatching displayed
  - c) erase the black border
  - d) Zoom-all and the model 'fills the screen'
  - e) ensure UCS Icon is on and at origin with menu bar View-Display-UCS Icon (these should be active)
  - f) ensure UCS BASE is current
  - g) command line with **UCSVP <R>** and:
 

```
prompt Enter new value for UCSVP
respond enter: 0 <R>
```
- 2 Menu bar with **View-Viewports-4 Viewports** and the model is displayed in 3D in each viewport
- 3 By entering **VPOINT <R>** at the command line, set the following vector viewpoints in the named active viewports:
 

| <i>viewport</i> | <i>viewpoint</i> |
|-----------------|------------------|
| top left        | 1,0,0            |
| top right       | 0,-1,0           |
| lower right     | 0,0,1            |
| lower left      | 1,-1,1           |
- 4 The model will be displayed at the viewpoints entered and will be of differing sizes in each viewport. The model now needs to be centred about a specific point
- 5 With the top left viewport active, menu bar with **View-Zoom-Scale** and:
 

```
prompt Enter a scale factor
respond enter: 1.75 <R>
```
- 6 Repeat the zoom-scale selection in the other three viewports, and enter a scale factor of 1.75 in the top right and lower right viewports, but 1 in the lower left (3D) viewport.
- 7 When the zoom-scale command has been completed, the model will be 'neatly centred' in all viewports as Fig. 10.13
- 8 Save as MODR2002\MV3DWF3

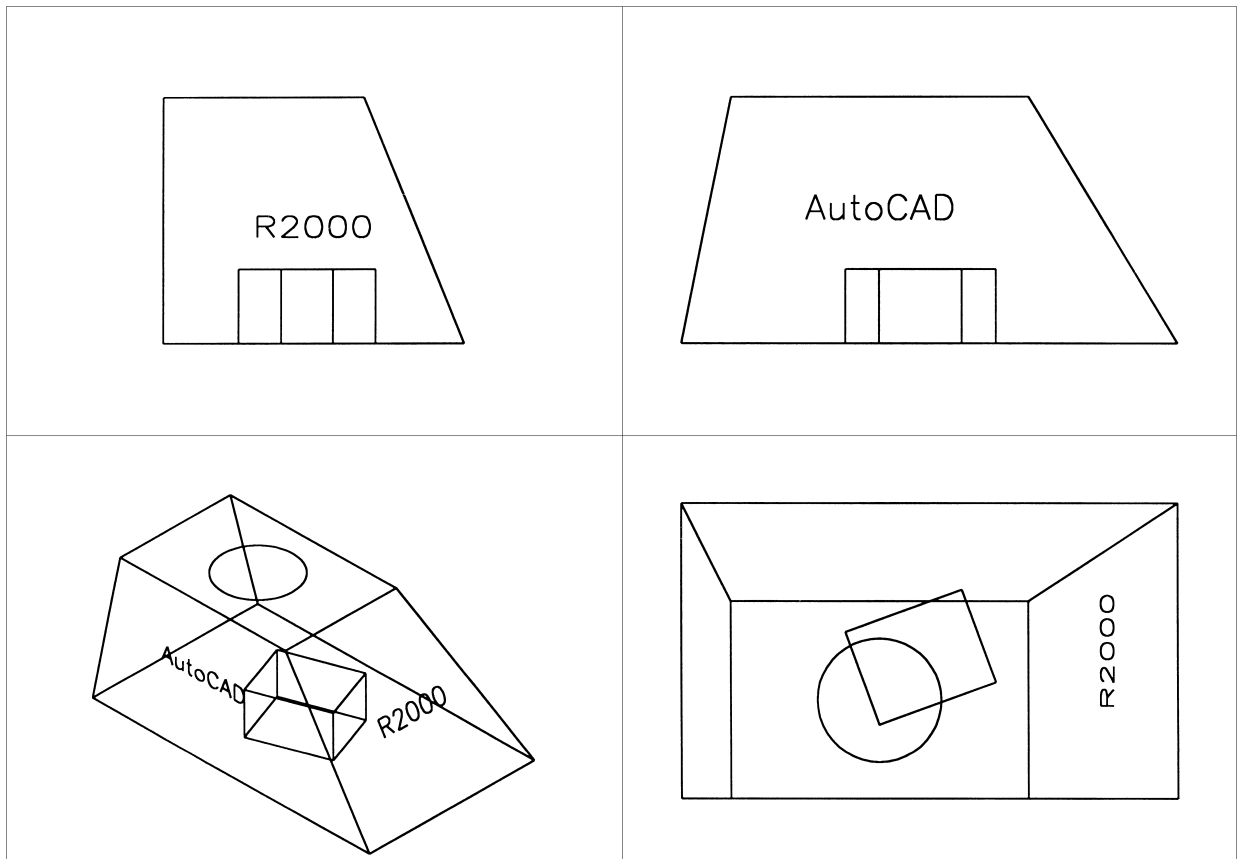


**Figure 10.13** Centre viewport example 1 – 3DWF3 by scale factor.

**Example 2 – centring about a user specified point**

- 1 Open drawing TEST3D to display the four viewport configuration of the created model with text on two 'planes'
- 2 Erase the black border and zoom-all in each viewport and the model will be displayed at different 'sizes' in the viewports
- 3 Ensure UCS BASE is current
- 4 a) ensure UCS Icon is on and at origin  
b) in each viewport use the command line UCSVP command and set the variable to 0
- 5 The model has a basic overall cuboid sizes of 200x120x100 and its 'centre point' *relative to UCS BASE* is at 100,60,50.
- 6 With the top left viewport active, menu bar with **View-Zoom-Center** and:  

```
prompt Specify centre point
enter 100,60,50 <R>
prompt Enter magnification or height <some value>
enter 175 <R>
```
- 7 Repeat the zoom-center command in the other three viewports and enter the center point as 100,60,50 and the magnification as 175 but 225 in the 3D viewport
- 8 The model will be centred in each viewport as Fig. 10.14
- 9 Save this display as MODR2002\MVTEST3D.



**Figure 10.14** Centre viewport example 2 – TEST3D by centre/magnification.

## Summary

- 1 The viewpoint command allows models to be viewed from different 'stand points'
- 2 The command has several selection options including:
  - a) four preset isometric views – SW, SE, NE and NW
  - b) six orthographic preset views – top bottom, left, right, front and back
  - c) three plan views – current UCS, world UCS, named UCS
  - d) VPOINT with three options – rotate, compass and tripod, vector
- 3 The VPOINT rotate option requires two angles:
  - a) the angle 'around' the model – the direction
  - b) the angle of inclination – the tilt
- 4 The VPOINT rotate option can be set from a dialogue box
- 5 The VPOINT compass/tripod option allows unlimited viewpoints
- 6 The VPOINT vector option requires an x,y,z coordinate entry
- 7 Viewpoints are generally set **absolute to the WCS** and the relative to the UCS option is **not recommended**
- 8 All wire-frame models exhibit **ambiguity** when the viewpoint command is used, i.e. viewed from above or from below?
- 9 The VIEW command allows different views of a model to be saved in the current drawing for future recall. This is useful when the model is being displayed at various viewpoints.
- 10 Models can be centred in multiple tiled viewports using:
  - a) Zoom-scale, the user entering the scale factor
  - b) Zoom-center, the user entering the centre point and the magnification
- 11 The centre point entered with zoom-center is **dependent on the UCS position**
- 12 I like to zoom-centre relative to UCS BASE which is usually 'set' at a convenient base vertex. It is then easy for me to work out the model centre point in relation to the overall size of the model.
- 13 The magnification value entered is a 'scale' effect and is relative to the given default, e.g. if the default is <180> then:
  - a) a value less than 180 will increase the model size
  - b) a value greater than 180 will decrease the model size.

## Assignments

Two activities have been included at this stage, both of which involve creating multiple tiled viewports, setting viewpoints and scaling/centring the models. The models have already been created (hopefully) during the hatching activities.

*Activity 8:* The hatched shaped block of Macfaramus.

- 1 Open the drawing of the hatched block from activity 6 and erase the border
- 2 Create a four viewport configuration to display top, front, right and a 3D view.
- 3 Zoom-all in each viewport
- 4 With UCS BASE current, zoom centre about the point 100,60,90 at 225 magnification in the three orthographic viewports, but 275 in the 3D viewport.
- 5 When complete, save as MODR2002\MVBLOCK

*Activity 9:* The hatched pyramid of Macfaramus.

- 1 Open hatched pyramid model from activity 7 and erase the border
- 2 Create a four viewport (left) configuration to display top, front, right and a SE Isometric view
- 3 Either:
  - a) Zoom-scale in each viewport with a scale factor of 1, but with a different scale factor in the 3D viewport
  - b) with UCS BASE current, zoom centre about the point 100,100,125 at 300 magnification in the three orthogonal viewports and your own entered magnification in the 3D viewport
- 4 When complete, save as MODR2002\MVPYR

# Model space and paper space and untiled viewports

AutoCAD 2002 has multi-view capabilities which allow the user to layout, organise and plot multiple views of any 3D model. The multiple viewport concept has already been used with our wire-frame models, these viewports being TILED, i.e. fixed.

In this chapter we will investigate how to create UNTILED or FLOATING viewports which are used in the same way that the tiled viewports were used. The creation of untiled viewports requires an understanding of the two AutoCAD drawing environments – model space and paper space.

The model/paper space concepts required by the user are:

1 *Model space*

This is the drawing environment that exists in any viewport and is the default. All models which have been created have been completed in model space. Model space is used for all draughting and design work and for setting 3D viewpoints. Multiple viewports are possible in model space but are TILED, i.e. they cannot be moved or altered in size – Fig. 11.1(a). While model space multi-views are useful, they have one major disadvantage – only the active viewport can be plotted, i.e. model space multiple viewports cannot be plotted on one sheet of paper.

2 *Paper space*

This is a drawing environment which is independent of model space. In paper space the user creates the drawing sheet, i.e. border, title box, etc. as well as arranging the multiple viewport layout. The viewports created in paper space are UNTILED, i.e. they can be positioned to suit, altered in size and additional viewports can be added to the layout – Fig. 11.1(b). In paper space the 3D viewpoint command is not valid although objects (particularly text) can be added to the sheet layout. The real advantage of working with paper space multiple viewports is that any viewport configuration can be plotted on the one sheet of paper.

3 *Tilemode*

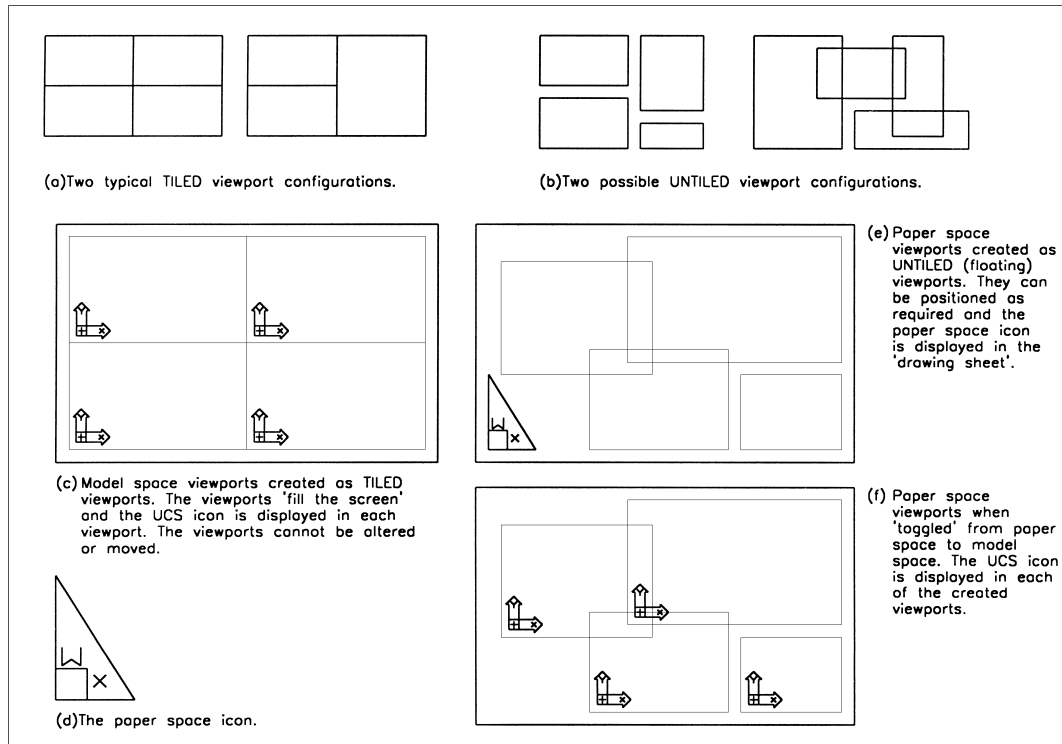
The system variable which controls the ‘type’ of viewport to be created is **TILEMODE** and:

- a) TILEMODE 1: model space (FIXED) viewports and paper space is not available
- b) TILEMODE 0: paper space (FLOATING) viewports and model space is available

Tiled (model space) viewports are always displayed as edge-to-edge and fill the screen like a tiled wall. Untiled (paper space) viewports can be positioned anywhere within the screen area with spaces between them if required. They can also be copied, moved, stretched etc.

4 *Icons*

When working in model space the normal WCS/UCS icon will be displayed in all viewports, orientated to the viewport viewpoint as Fig. 11.1(c). In paper space, the paper space icon – Fig. 11.1(d) is displayed.



**Figure 11.1** Model and paper space concepts.

When viewports are created in paper space, the paper space icon is displayed in the lower left corner of the screen as Fig. 11.1(e), but when model space 'is entered' the UCS icon is again displayed in all created viewports as Fig. 11.1(f).

#### 5 *Toggling between model and paper space*

All AutoCAD users will be familiar with the toggle concept, e.g. toggling the grid on/off with the F7 key or from the Status bar with a left-click on the word GRID. It is possible to toggle between model space and paper space but only if the TILEMODE system variable has been set to 0. The toggle effect can be activated by:

##### 1. *Command line*

- a) if in paper space, toggle to model space with **MS <R>**
- b) if in model space, toggle to paper space with **PS <R>**

##### 2. *Status bar*

- a) left-click on PAPER to toggle to model space
- b) left-click on MODEL to toggle to paper space

#### 6 *UCSVP*

UCSVP is a system variable which determines the orientation of the UCS in multiple viewports and:

- a) UCSVP 0 : the UCS icon reflects the UCS setting of the current viewport
- b) UCSVP 1 : the UCS icon is independent of the UCS setting of the current viewport This means that if the UCSVP is set to 0 and the UCS is altered, the icon will reflect this altered position. If UCSVP is set to 1, and alteration of the UCS will not be reflected by the icon. For all our multiple viewport work, **UCSVP will be set to 0**, as I believe the icon should reflect the UCS setting.

#### 7 *Layout tabs*

AutoCAD has a layout tab line which allows the user to create several layouts of a model in the current drawing. The layout tab line has a Model tab display and usually two layout tabs – Layout1 and Layout2. We will use the Model and Layout tabs with our multiple viewports.

## Model/paper space example – untiled viewports

This example will use a model created in model space to demonstrate the paper space multiple viewport concept. The example is quite long but if you are unsure of paper space, persevere with it – it is important that you understand how to create and use paper space viewports.

Note

- a) Although you may be familiar with the model/paper space concept it is advisable that you complete the exercise.
- b) AutoCAD 2002 assumes that the user has access to a printer or plotter, this being connected to the computer. As all printer/plotters differ in their configurations, paper size etc, I have *assumed* that no printer is configured. This does not exclude the user from obtaining a hard copy of completed drawings, it is simply a matter of convenience for me when detailing the various steps in the exercises.
- c) There are different methods of creating paper space untiled viewports from a model space drawing. This example demonstrates one of these methods.

- 1 Open drawing MODR2002\3DWF.M and:
  - a) erase the black border, hatching and any dimensions
  - b) leave the text items – they will ‘act as a reference’
  - c) layer MODEL and UCS BASE current
  - d) zoom-all and model ‘fills the screen’
  - e) refer to Fig. 11.2

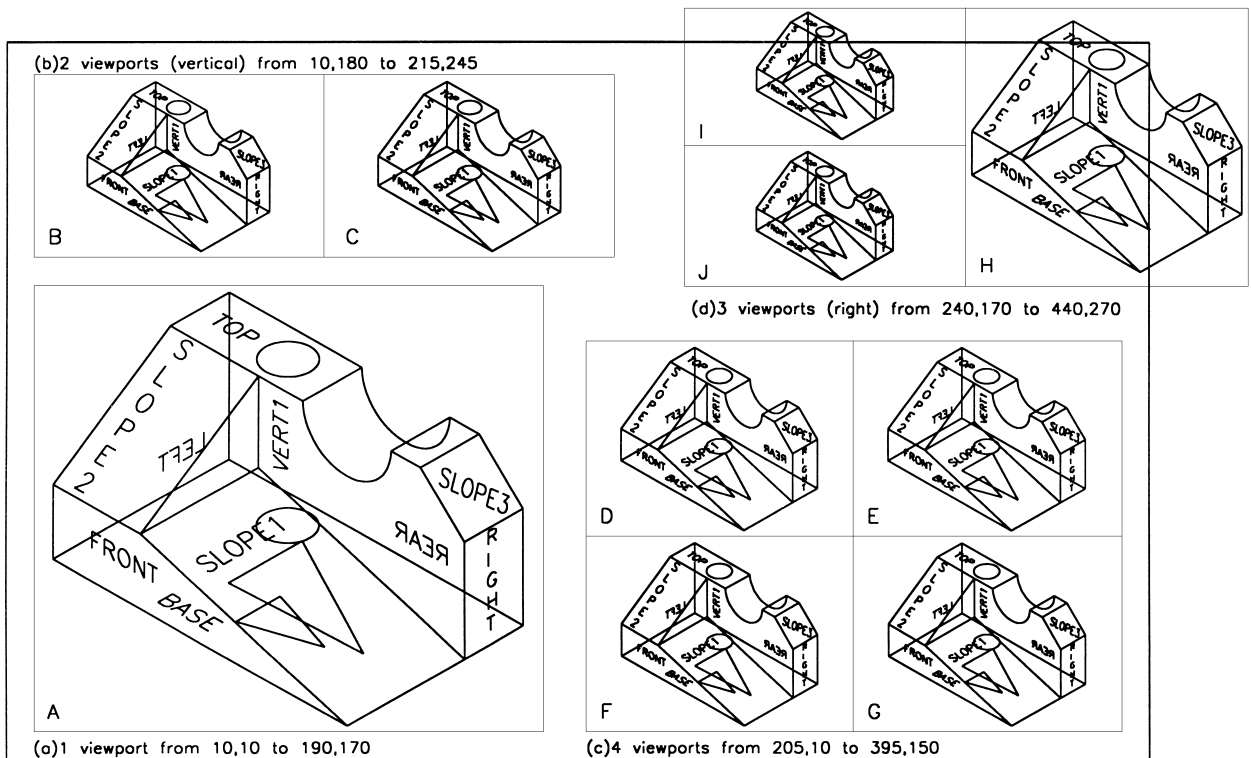


Figure 11.2 Creating the paper space multiple viewport layout with 3DWF.M.

- 2 Make a new layer called VP, linetype continuous, colour 12
- 3 At the command line enter **PS <R>** and prompt line will display:  
*\*\* Command not allowed in Model tab \*\**
- 4 Menu bar with **Tools-Wizards-Create Layout** and:
  - prompt* Create Layout - Begin dialogue box
  - respond* enter layout name: **MYFIRST** then pick Next
  - prompt* Create Layout - Printer dialogue box
  - respond* pick **None** then pick Next
  - prompt* Create Layout - Paper Size dialogue box
  - respond*
    1. scroll and pick **ISOA3 (420.00x297.00MM)**
    2. Drawing units: Millimeters
    3. pick Next
  - prompt* Create Layout - Orientation dialogue box
  - respond* ensure **Landscape** active then pick Next
  - prompt* Create Layout - Title Block dialogue box
  - respond* pick **None** then pick Next
  - prompt* Create Layout - Define Viewports dialogue box
  - respond* pick **None** then pick Next
  - prompt* Create Layout -Finish dialogue box
  - respond* pick **Finish**
  - and*
    1. screen returned in paper space – note the icon
    2. the white area is the A3 paper
    3. the dotted area is the permitted drawing area within the white A3 paper – for plot purposes
    4. a new layout name MYFIRST is added to the tab bar
- 5 Make layer 0 current and menu bar with **Draw-Rectangle** and:
  - prompt* Specify first corner point
  - enter* **0,0 <R>**
  - prompt* Specify other corner point
  - enter* **405,257 <R>**
- 6 This rectangle is our drawing area outline
- 7 Make layer VP current
- 8 Menu bar with **View-Viewports-1 Viewport** and:
  - prompt* Specify corner of viewport and enter: **10,10 <R>**
  - prompt* Specify opposite corner and enter: **190,170 <R>**
  - and* a viewport (A) is created with the model 3DWFM displayed as fig(a)
- 9 Menu bar with **View-Viewports-2 Viewports** and:
  - prompt* Enter viewport arrangement and enter: **V <R>**
  - prompt* Specify first corner and enter: **10,180 <R>**
  - prompt* Specify opposite and enter: **215,245 <R>**
  - and* two additional viewports (B and C) are created, each displaying the model in 3D – fig(b)
- 10 Menu bar again with **View-Viewports-4 Viewports** and:
  - prompt* Specify first corner and enter: **205,10 <R>**
  - prompt* Specify opposite corner and enter: **395,150 <R>**
  - and* four new viewports (D,E,F,G) are created with the model displayed in each – fig(c)
- 11 Final menu bar selection with **View-Viewports-3 Viewports** (Right option) and:
  - prompt* Specify first corner and enter: **240,170 <R>**
  - prompt* Specify opposite corner and enter: **440,270 <R>**
  - and* three viewports (H,I,J) are created each with the model displayed as fig(d). These viewports extend 'outwith' the drawing paper.

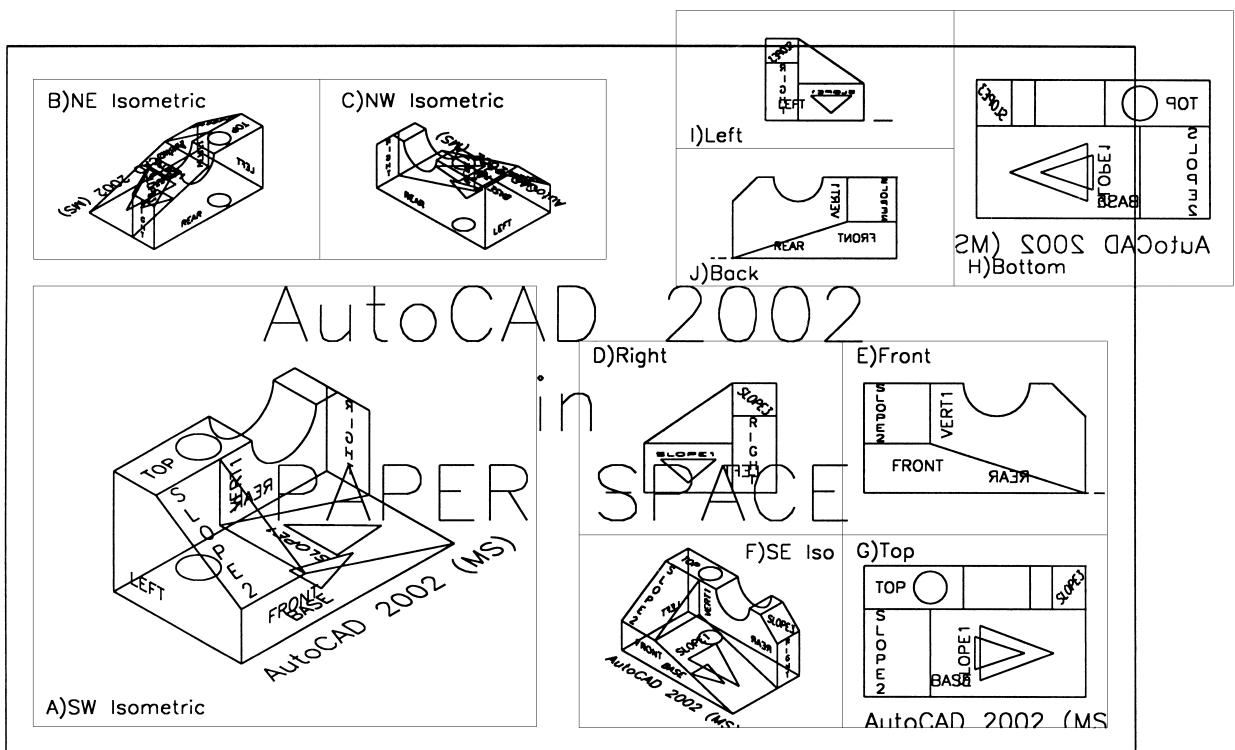
- 12 *What has been achieved?*
- ten different sized viewports have been created
  - the viewports have been created in paper space
  - each viewport displays the original 3DWFM model
- 13 At the command line enter MS <R> and:
- toggled to model space
  - the last viewport (top right) is active?
  - each viewport displays the UCS icon at BASE?
  - any viewport can be made active as before, i.e. move to required viewport and left-click
- 14 Investigate the paper/model space toggle:
- from the command line with PS and MS
  - status bar with left-click on MODEL/PAPER
  - decide on which toggle method you prefer
- 15 Investigate the model/layout tab line:
- pick Model – the original 3DWFM model is displayed
  - pick MYFIRST – our multiple viewport layout is displayed.
- 16 At this stage the layout is as Fig. 11.2

## Setting the viewpoints

- 1 Enter model space and set the following viewpoints in the named viewports:
- | <i>viewport</i> | <i>viewpoint</i> | <i>viewport</i> | <i>viewpoint</i> |
|-----------------|------------------|-----------------|------------------|
| A               | SW Isometric     | F               | SE Isometric     |
| B               | NE Isometric     | G               | Top              |
| C               | NW Isometric     | H               | Bottom           |
| D               | Right            | I               | Left             |
| E               | Front            | J               | Back             |
- 2 The model is now displayed at a different viewpoint in each viewport. Using the Zoom-Scale command, enter the following scale factors in the appropriate viewport:
- scale factor of 1.75 in orthogonal view viewports
  - scale factor of 1.25 in 3D view viewports.
- 3 The UCS icon may not be displayed as expected so:
- in each viewport, enter **UCSV P <R>** then **0 <R>**
  - make the largest viewport (A) current
  - restore UCS BASE and the icon should be displayed as expected.

## Adding text

- 1 In model space, make layer TEXT current and viewport (A) active
- 2 Using the menu bar sequence, **Draw-Text-Single Line Text**, add an item of text using the following:
  - a) start point: 0,-30
  - b) height: 15 and rotation: 0
  - c) text item: AutoCAD 2002 (MS)
- 3 This item of text will be displayed in the ten viewports at different orientations due to the viewpoints. In some viewports (D,E,I,J) the text is viewed 'end-on'
- 4 Enter paper space with PS <R>
- 5 Menu bar with **Draw-Text-Single Line Text** and add an item of text using:
  - a) start point: centred on 200,150
  - b) height: 20 and rotation: 0
  - c) text: AutoCAD 2002 <R>  
text: in <R>  
text: PAPER SPACE <R><R> – two returns
- 6 At this stage your screen layout should resemble Fig. 11.3



**Figure 11.3** Working with the created paper space viewports.

## Modifying the layout

- 1 In paper space try and erase the model – you cannot
- 2 In model space try and erase the paper space text – not possible
- 3 In paper space, activate the ZOOM command and window viewport A. The viewport will be enlarged and by entering model space, it is easier to work on the model.
- 4 In paper space, zoom previous to restore the original layout
- 5 In paper space, select the SCALE icon from the Modify toolbar and:
  - prompt* Select objects
  - respond* **completely window viewports H,I,J then right-click**
  - prompt* Specify base point
  - respond* **Intersection of lower left corner of viewport J**
  - prompt* Specify scale factor
  - enter* **0.75 <R>**
- 6 In model space:
  - a) zoom-all in viewports H, I and J
  - b) then zoom-scale with the same 1.75 scale factor as before.
- 7 Enter paper space and freeze the layer VP
- 8 The ten views of the model will be displayed with text but without the viewport borders – Fig. 11.4
- 9 If you have access to a printer/plotter:
  - a) print from any viewport in model space
  - b) print from paper space
- 10 This completes this exercise – save if required.

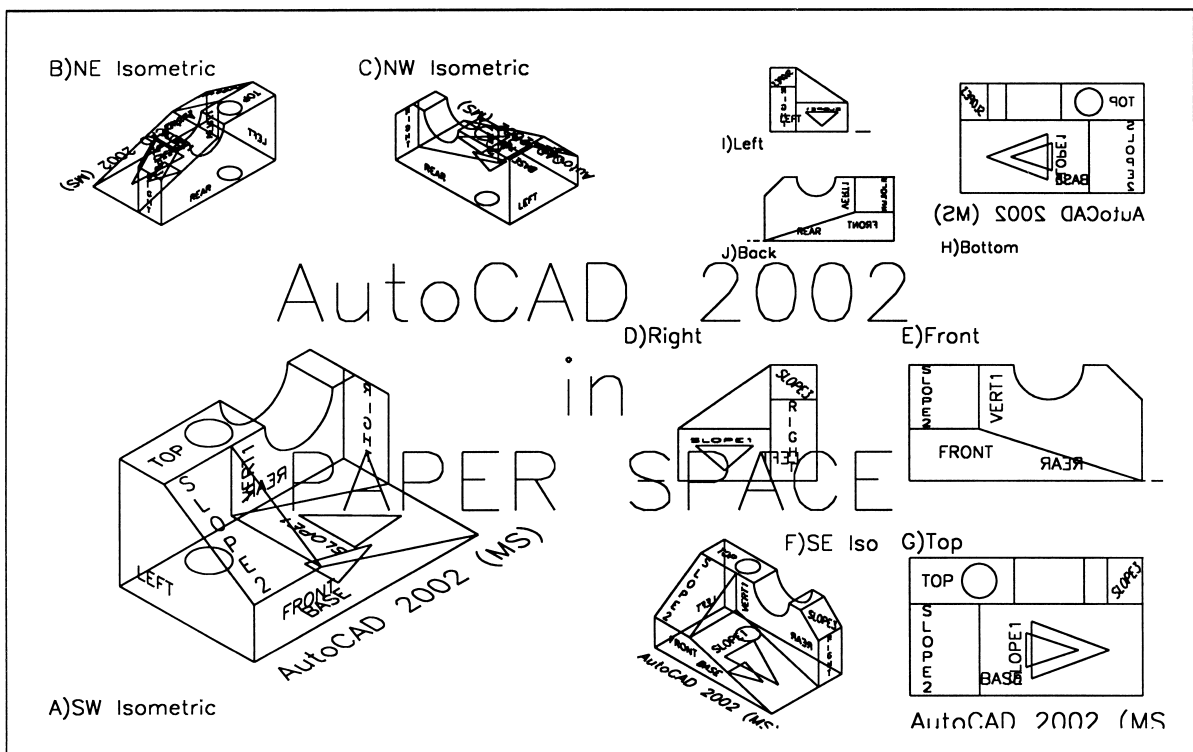


Figure 11.4 Completed paper space exercise with 3DWF.

## Comparison between model space and paper space

The following table gives a brief comparison between the model and paper space drawing environments:

### *Model space*

used to create the model  
 model can be modified  
 tiled (fixed) viewports  
 tilemode: 1  
 viewports restricted in size  
 viewports 'fill screen'  
 viewports cannot be altered  
 cannot add viewports  
 plot only active viewport  
 3D views active  
 WCS or UCS icon  
 zoom in active viewport

### *Paper space*

used to create the paper layout  
 model cannot be modified  
 untiled (floating) viewports  
 tilemode: 0  
 viewports to any size  
 viewports positioned to suit  
 viewports can be moved, copied etc  
 additional viewports can be created  
 all viewports can be plotted  
 cannot use 3D viewpoint  
 paper space icon  
 zoom complete viewports

## Finally

One of the major benefits of paper space is the ability to zoom a complete viewport. This allows the zoomed viewport to be enlarged, thus allowing the user to 'see more clearly' the model being created/worked on when in model space. This is a concept with which the user should become familiar and the procedure is:

- 1 Enter paper space with PS <R>
- 2 Zoom-window a specific viewport, e.g. the 3D viewport
- 3 Return to model space with MS <R>
- 4 Complete (or modify) the model
- 5 Enter paper space with PS <R>
- 6 Zoom-previous (or All) to restore the complete layout
- 7 Return to model space with MS <R>

This zoom a viewport in paper space is a very useful concept and will be used in all future exercises.

# New 3D multiple viewport standard sheet

Now that the model/paper space concept has been discussed, a new standard sheet will be created which will allow all future models (surface and solid) to be displayed in multiple viewports. This standard sheet will be created with three layout tab displays, and the original 3DSTDA3 sheet will be modified as it:

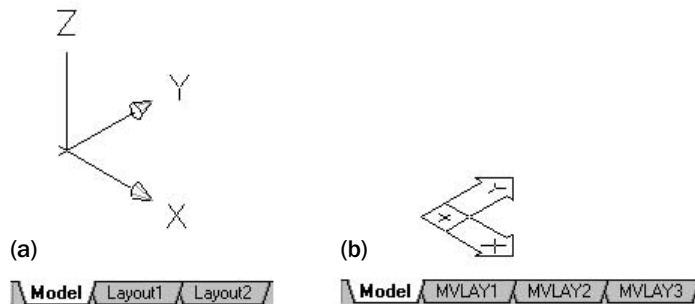
- a) already has layers, e.g. MODEL, OBJECTS, TEXT, etc.
- b) has a created dimension style – 3DSTD
- c) has other variables set

## Getting ready

- 1 Close any existing drawings and open your 3DSTDA3 standard sheet – template or drawing (it does not matter) to display:
  - a) the black border as a SE Isometric viewpoint
  - b) the WCS icon at the left vertex of the border
- 2 Erase the black border – we will not use it again
- 3 Menu bar with **View-Display-UCS Icon** and:
  - a) On and Origin active (tick)
  - b) Properties and decide on a 2D or 3D icon display
- 4 Menu bar with **Format-Layer** and create two new layers:
 

| name  | colour     | linetype   |
|-------|------------|------------|
| VP    | number 14  | continuous |
| SHEET | number 212 | continuous |
- 5 At the command line enter **UCSVP <R>** and:
 

```
prompt Enter new value for UCSVP
enter 0 <R>
```
- 6 Note the layout tab line – similar to Fig. 12.1(a)



**Figure 12.1** The layout tab displays. (a) Original tab with a 3D icon. (b) The layout tab display after creating three new viewport configurations with a 3D icon.

## The first layout tab display

- 1 Menu bar with **Tools-Wizards>Create Layout** and with the Create Layout dialogue box set the following:
  - a) Begin: enter **MVLAY1** then Next
  - b) Printer: None then Next
  - c) Paper Size: Millimeters, ISO A3 (420×297) then Next
  - d) Orientation: Landscape then Next
  - e) Title Block: None then Next
  - f) Define viewports: None then Next
  - g) Finish: pick Finish
- 2 A white paper area will be returned with a dotted drawing area within, this being in paper space.
- 3 Make layer SHEET current
- 4 Menu bar with **Draw-Rectangle** and:
 

*prompt* Specify first corner point and enter: **0,0 <R>**

*prompt* Specify other corner point and enter: **405,257 <R>**
- 5 Zoom-all and PAN to suit
- 6 Draw a line from: 0,15 to: @405,0
- 7 The area at the bottom of the 'paper' is for you to 'customise' as required. Use layer SHEET for this.
- 8 Make layer VP current
- 9 Menu bar with **View-Viewports-New Viewports** and:
 

*prompt* New Viewports dialogue box

*respond*

  1. pick Four: Equal
  2. Viewport spacing: 0
  3. Setup: 3D
  4. Change views to: leave unaltered
  5. pick OK

*prompt* Specify first corner and enter: 10,20<R>

*prompt* Specify opposite corner and enter: 395,247<R>
- 10 The screen will display four paper space viewports within the sheet border
- 11 To set and save some UCS positions, enter model space with the command line entry **MS <R>**
- 12 Making each viewport active, enter **UCSVP <R>** and:
 

*prompt* Enter new value for UCSVP

*enter* **0 <R>**
- 13 Make the lower left viewport active then menu bar with Tools-New UCS-World
- 14 Menu bar with **Tools-New UCS-Origin** and:
 

*prompt* Specify new origin point

*enter* **10,10,0 <R>**
- 15 At command line enter **UCS <R>** then **S <R>** and:
 

*prompt* Enter name to save current UCS

*enter* **BASE <R>**
- 16 Menu bar with **Tools-New UCS-X** and:
 

*prompt* Specify rotation angle about X axis

*enter* **90 <R>**
- 17 Command line with **UCS <R>** then **S <R>** and enter: **FRONT <R>**
- 18 Menu bar with **Tools-New UCS-Y** and:
 

*prompt* Specify rotation angle about Y axis

*enter* **90 <R>**
- 19 Command line with **UCS <R>** then **S <R>** and enter: **RIGHT <R>**
- 20 We have now set and saved three UCS positions – BASE, FRONT and RIGHT. These three UCS positions will assist with future model creation.
- 21 Restore UCS BASE and make the lower left viewport active

## A second layout tab display

- 1 Still with the MVLAY1 tab and the lower left viewport active with UCS BASE current
- 2 Left-click on the Layout1 tab and:
  - prompt* Page Setup – Layout1 tab dialogue box
  - respond*
    1. Plot Device tab active and set:
      - a) Plotter configuration: None
    2. Layout Settings tab active and set:
      - a) Paper size: ISO A3 (420×297)
      - b) Drawing orientation: Landscape
      - c) Plot area: Layout
      - d) Plot scale: 1:1
      - e) pick OK
  - and* Paper space entered with:
    - a) white drawing paper
    - b) dashed line plotting area
    - c) coloured rectangle – a new viewport
  - respond* erase the viewport
- 3 Make layer SHEET current
- 4 Menu bar with **Draw-Rectangle** and:
  - prompt* Specify first corner point and enter: **0,0 <R>**
  - prompt* Specify other corner point and enter: **405,257 <R>**
- 5 Zoom-all and PAN to suit
- 6 Draw a line from: 0,15 to: @405,0 and customise the area below this line using layer SHEET
- 7 Make layer VP current and menu bar with **View-Viewports-New Viewports** and set:
  - a) Configuration: Two vertical
  - b) Spacing: 0
  - c) Setup: 3D
  - d) Change view to: leave at present
  - e) first corner: 10,20
  - f) opposite corner: 395,247
- 8 Enter model space and:
  - a) set UCSVP to 0 in each viewport
  - b) restore UCS BASE
- 9 Make the right viewport active and with **VPOINT <R>** at the command line:
  - a) enter R <R> for the rotate option
  - b) enter 210 as the angle in the XY plane
  - c) enter 30 as the angle from the XY plane
- 10 With the left viewport active, **VPOINT <R>** again and:
  - a) enter R <R> for the rotate option
  - b) enter 210 as the angle in the XY plane
  - c) enter -30 as the angle from the XY plane
- 11 We have set two viewpoints in this layout tab, one from above (the right viewport) and one from below (the left)
- 12 Right-click on Layout1 tab name and:
  - prompt* Shortcut menu
  - respond* **pick Rename**
  - prompt* Rename Layout dialogue box
  - respond*
    1. alter name to **MVLAY2**
    2. pick OK
- 13 Now have created two layout settings, MVLAY1 and MVLAY2 with the Model tab still available.

## A third layout tab display

Although we have created two layout tab displays, we will create a third layout tab configuration which is slightly different from the two 'traditional' layouts MVLAY1 and MVLAY2. The sequence is long and uses coordinates for most of the entries. I would ask the user to persevere with this exercise, as the end result is worth the time and effort spent.

- 1 Still with the MVLAY2 tab and the right viewport active with UCS BASE current
- 2 Left-click on the Layout2 tab and:
  - prompt* Page Setup - Layout2 tab dialogue box
  - respond* 1. Plot Device tab active and set:
    - a) Plotter configuration: None
  - 2. Layout Settings tab active and set:
    - a) Paper size: ISO A3 (420x297)
    - b) Drawing orientation: Landscape
    - c) Plot area: Layout
    - d) Plot scale: 1:1
    - e) pick OK
  - and* Paper space entered with:
    - a) white drawing paper
    - b) dashed line plotting area
    - c) coloured rectangle - a new viewport
  - respond* erase the viewport
- 3 Make layer SHEET current
- 4 Menu bar with **Draw-Rectangle** and:
  - prompt* Specify first corner point and enter: **0,0 <R>**
  - prompt* Specify other corner point and enter: **405,257 <R>**
- 5 Zoom-all and PAN to suit
- 6 Draw a line from: 0,15 to: @405,0 and customise the area below this line using layer SHEET
- 7 Make layer VP current
- 8 Menu bar with **View-Viewports-Polygonal Viewport** and:
  - prompt* Specify start point and enter: **5,140 <R>**
  - prompt* Specify next point and enter: **140,140 <R>**
  - prompt* Specify next point and enter: **170,195 <R>**
  - prompt* Specify next point and enter: **170,250 <R>**
  - prompt* Specify next point and enter: **5,250 <R>**
  - prompt* Specify next point and enter: **C <R>** - the close option
- 9 A five sided viewport will be displayed at the top left corner of the drawing area
- 10 Menu bar with **Modify-Mirror** and:
  - prompt* Select objects
  - respond* **pick any point on viewport** then right-click
  - prompt* First point of mirror line and enter: **202.5,15 <R>**
  - prompt* Second point of mirror line and enter: **202.5,250 <R>**
  - prompt* Delete source objects and enter: **N <R>**

- 11 Repeat the menu bar with **Modify-Mirror** selection and:
  - prompt* Select objects
  - respond* **pick any point on the two viewports** then right-click
  - prompt* First point of mirror line and enter: **0,136 <R>**
  - prompt* Second point of mirror line and enter: **405,136 <R>**
  - prompt* Delete source objects and enter: **N <R>**
- 12 We have now created four polygonal viewports
- 13 Menu bar with **View-Viewports-Polygonal Viewport** and:
  - prompt* Specify start point and enter: **152.5,136 <R>**
  - prompt* Specify next point or [Arc/Length/Undo]
  - enter* **A <R>** the arc option
  - prompt* Enter arc boundary option
  - enter* **CE <R>** – the centre point option
  - prompt* Specify center point of arc and enter: **202.5,136 <R>**
  - prompt* Specify endpoint of arc and enter: **252.5,136 <R>**
  - prompt* Specify endpoint of arc and enter: **152.5,136 <R>**
  - prompt* Specify endpoint of arc and **<RETURN>** to end command
- 14 We have now added a circular viewport to our Layout3 display
- 15 Enter model space and make any of the new viewports active
- 16 Menu bar with **View-Display-UCS Icon** and ensure that On and Origin are both active. I had to complete this sequence in all the five viewports
- 17 Enter UCSVP and 0 in all viewports
- 18 Make layer MODEL current and restore UCS BASE
- 19 Set five viewpoints to your own specification. I set the viewpoints to SE Isometric, Left, Right, Front and Top, but it is your choice
- 20 Right-click on Layout2 tab and:
  - a) pick Rename from shortcut menu
  - b) enter **MVLAY3** as the layout name
  - c) pick OK
- 21 At this stage the layout tab should display Model, MVLAY1, MVLAY2 and MVLAY3 similar to Fig. 12.1(b)
- 22 If the layout tab names are 'not in order' then right-click an MVLAY name and use the Move/Copy option of the shortcut menu to position as required.

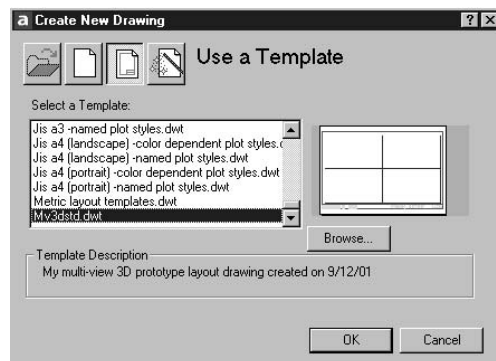
## Saving the layouts as a new standard sheet

- 1 Make any layout tab MVLAY1 (for example) current
- 2 Enter model space with the lower left viewport active
- 3 Layer MODEL and UCS BASE current
- 4 Menu bar with **File-Save As** and:
  - prompt* Save Drawing As dialogue box
  - respond*
    1. scroll at Files of Type
    2. pick **AutoCAD Drawing Template File (\*.dwt)**
  - prompt* list of existing template files in the AutoCAD template folder
  - respond*
    1. enter File name as: **MV3DSTD**
    2. Measurement: Metric
    3. pick Save
  - prompt* Template Description dialogue box
  - respond*
    1. enter: **My multi-view 3D prototype layout drawing created on XX/YY/ZZ**
    2. pick OK
- 5 Repeat step 2, but save as a template to your named folder with the same name, i.e. MV3DSTD
- 6 Menu bar with **File-Save As** and:
  - prompt* Save Drawing As dialogue box
  - respond*
    1. scroll and pick **AutoCAD 2000 Drawing (\*.dwg)**
    2. scroll and pick your named folder (MODR2002)
    3. enter file name: **MV3DSTD**
    4. pick Save
- 7 The template/drawing file will be used extensively when starting new model exercises.

## Checking the new MV3DSTD layout

Now that the MV3DSTD template file has been created we will add some 3D objects to 'check' the layout. Try and reason out the coordinate entries.

- 1 Close any existing drawings
- 2 Menu bar with **File-New** and:
  - prompt* Create New Drawing dialogue box
  - respond*
    1. pick Use a Template
    2. scroll and pick **MV3DSTD.dwt** – Fig. 12.2
    3. pick OK



**Figure 12.2** The Use a Template dialogue box with MV3DSTD selected.

- 3 The created multiple viewport drawing should be displayed with the layout tab current that you saved as.
- 4 Display the Object Snap and Surfaces toolbars
- 5 Menu bar with **Draw-Surfaces-3D Surfaces** and:
  - prompt* 3D Objects dialogue box
  - respond* **pick Box3d then OK**
  - prompt* Specify corner of box and enter: **0,0,0 <R>**
  - prompt* Specify length of box and enter: **200 <R>**
  - prompt* Specify width of box and enter: **100 <R>**
  - prompt* Specify height and enter: **80 <R>**
  - prompt* Specify rotation angle about Z axis and enter: **0 <R>**
- 6 Menu bar with **Draw-Surfaces-3D Surfaces** and:
  - prompt* 3D Objects dialogue box
  - respond* **pick Wedge then OK**
  - prompt* Specify corner point of wedge and enter: **0,0,0 <R>**
  - prompt* Specify length of wedge and enter: **100 <R>**
  - prompt* Specify width of wedge and enter: **100 <R>**
  - prompt* Specify height of wedge and enter: **100 <R>**
  - prompt* Specify rotation angle of wedge about Z axis and enter: **-90 <R>**
- 7 At the command line enter **CHANGE <R>** and:
  - prompt* Select objects
  - respond* **pick the wedge then right-click**
  - prompt* Specify change point or [Properties] and enter: **P <R>**
  - prompt* Enter property to change and enter: **C <R>**
  - prompt* Enter new color and enter: **BLUE <R><R>** – two returns
- 8 Select the CONE icon from the Surfaces toolbar and:
  - prompt* Specify center point for base of cone and enter: **70,50,80 <R>** 
  - prompt* Specify radius for base of cone and enter: **50 <R>**
  - prompt* Specify radius for top of cone and enter: **0 <R>**
  - prompt* Specify height of cone and enter: **100 <R>**
  - prompt* Enter number of segments for surface of cone and enter: **16 <R>**
- 9 Change the colour of the cone to green
- 10 Select the DISH icon from the Surface toolbar and:
  - prompt* Specify center point of dish and enter: **150,50,0 <R>** 
  - prompt* Specify radius of dish and enter: **50 <R>**
  - prompt* Enter number of longitudinal segments for surface of dish  
*and* – enter: **16<R>**
  - prompt* Enter number of latitudinal segments for surface of dish  
*and* – enter: **8 <R>**
- 11 Change the colour of the dish to magenta
- 12 Make each viewport current and:
  - a) zoom to extents
  - b) zoom to a scale of 1
- 13 With UCS BASE make layer TEXT current and menu bar with **Draw-Text-Single Line Text** and:
  - prompt* Specify start point of text and enter: **130,80,80 <R>**
  - prompt* Specify height and enter: **10 <R>**
  - prompt* Specify rotation angle of text and enter: **0 <R>**
  - prompt* Text and enter: **AutoCAD <R><R>**

14 Add two other text items using the following information:

|             | <i>first item</i> | <i>second item</i> |
|-------------|-------------------|--------------------|
| Named UCS   | FRONT             | RIGHT              |
| start point | 110,40,0          | 15,15,200          |
| height      | 15                | 20                 |
| rotation    | 0                 | 0                  |
| item        | Release           | 2002               |

15 Restore UCS BASE

16 Enter paper space with **PS <R>**

17 At the command line enter **DTEXT <R>** and add the following text using the information given:

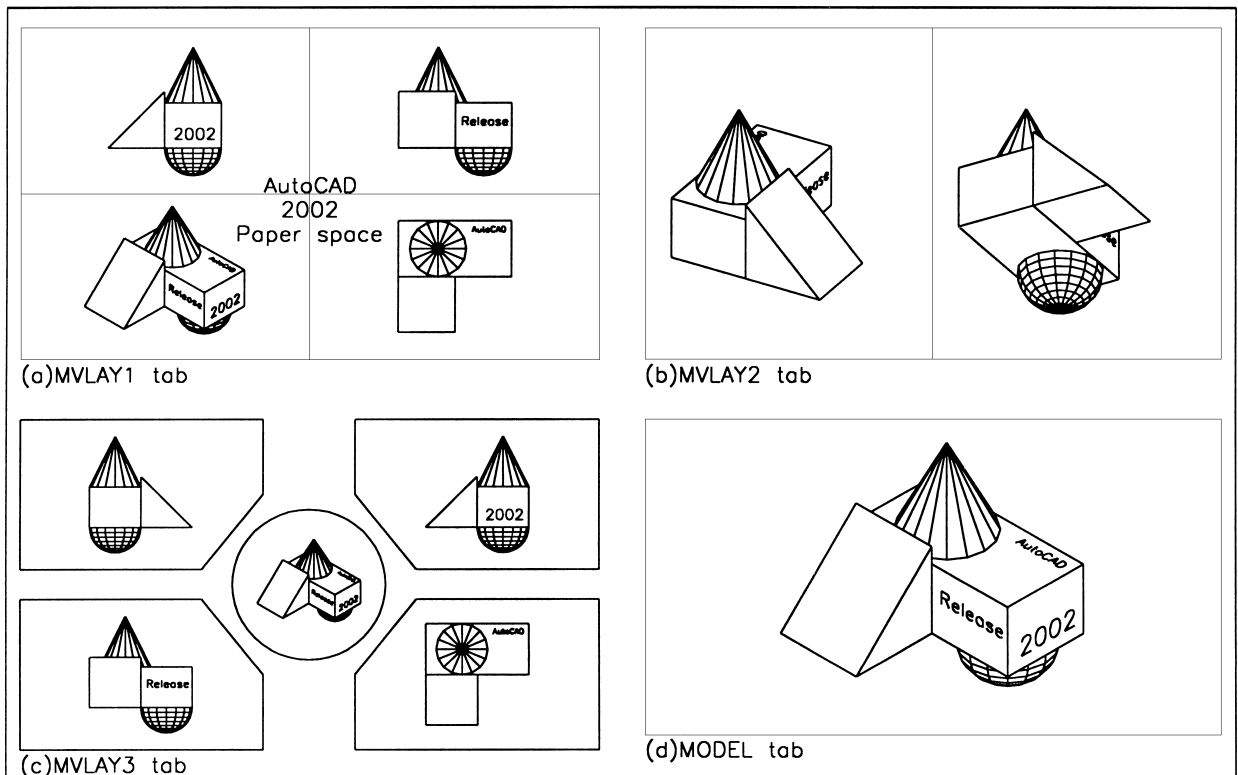
- a) start point: centred on 202.5,133.5
- b) height: 10 and rotation: 0
- c) enter text: AutoCAD <R>  
     enter text: 2002 <R>  
     enter text: Paper space <R><R>

18 Return to model space with **MS <R>**

19 Menu bar with **View-Hide** in each viewport and your model will be displayed with hidden line removal – Fig. 12.3(a)

20 Now select the MVLAY2 tab and:

- a) in each viewport Zoom-Extents
- b) Zoom-Scale with a 1 scale factor
- c) hide each viewport – Fig. 12.3(b)



**Figure 12.3** The layout tabs for checking the MV3DSTD standard sheet.

- 21 Select the MVLAY3 tab and repeat the three entries (a), (b) and (c) as step 20 – Fig. 12.3(c)
- 22 Select the Model tab to display the original 3D view of the model and hide – Fig. 12.3(d)
- 23 *Task*
  - a) Using each tab name and in model space, make each viewport active and menu bar with **View-Shade-Gouraud Shaded** to display the model in colour. Note the effect of the shading on the model space and paper space text.
  - b) When you have completed shading, menu bar with **View-Shade-2D Wireframe** to restore the original views of the model. This will also restore the paper space text – interesting?

This (long) chapter is now complete. You do not have to save the drawing, but if you do be careful not to over-write your standard sheet.

We can concentrate on surface and solid modelling.

# Surface modelling

The best way of describing a surface model is to think of a wire-frame model with 'skins' covering all the wires from which the model is constructed. The 'skins' convert a wire-frame model into a surface model with several advantages:

- 1 The model can be displayed with hidden line removal
- 2 There is no ambiguity
- 3 The model can be shaded and rendered

AutoCAD 2002 adds **FACETED** surfaces using a polygon mesh technique, but this mesh only approximates to curved surfaces. The mesh density (the number of facets) is controlled by certain system variables which will be discussed in the appropriate chapter.

The different types of surface models available with AutoCAD 2002 are:

- 3D faces
- 3D meshes
- polyface meshes
- ruled surfaces
- tabulated surfaces
- revolved surfaces
- edge surfaces
- 3D objects

Each surface type will be demonstrated by worked examples in their own chapter.

Surface model commands can be activated:

- 1 From the menu bar with **Draw-Surfaces**
- 2 In icon form from the Surfaces toolbar
- 3 By direct keyboard entry, e.g. **3DFACE <R>**

The various exercises will use all three methods.

# 3DFACE and PFACE

These two commands appear similar in operation, both adding faces (skins) to wire-frame models. If these added faces are in colour, the final model display can be quite impressive.


## 3Dface example 1

**A 3Dface is a three or four sided surface added to an object and is independent of the UCS position.**

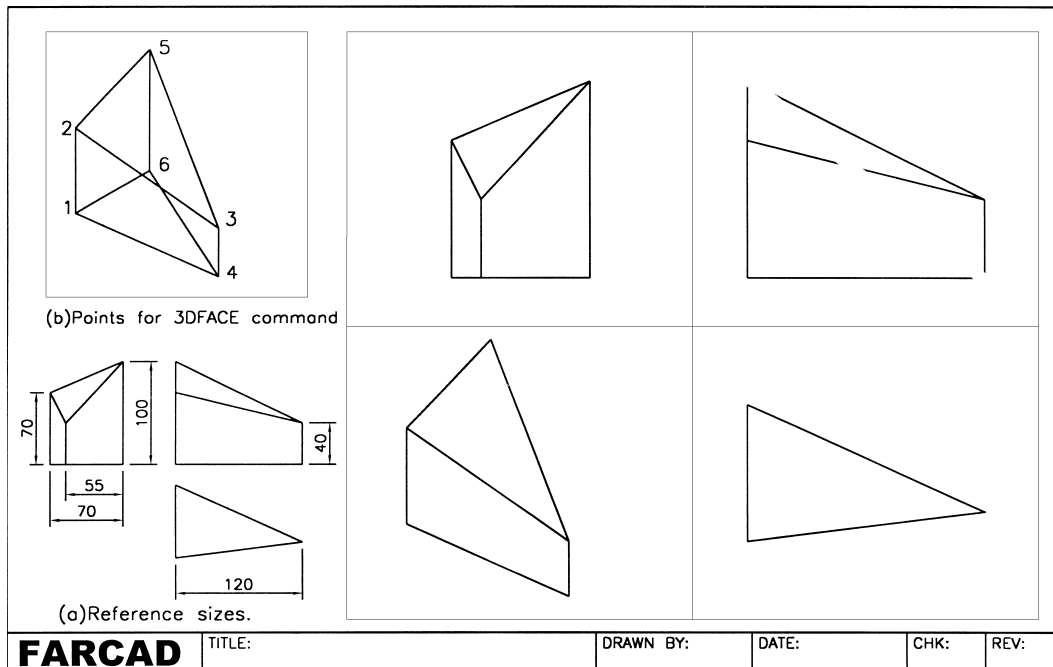
- 1 Close any existing drawings then menu bar with **File-New** and 'open' your **MV3DSTD** template file with layer MODEL and UCS BASE both current
- 2 Ensure MVLAY1 tab is current and:
  - a) display toolbars to suit
  - b) make the lower left (3D) viewport active
  - c) zoom-centre about the point 60,35,50 at 175 magnification
- 3 Create a wire-frame model using the LINE command and the reference sizes in Fig. 14(a) Ensure that point 1 is (0,0,0)
- 4 When the 3D wire-frame model is complete:
  - a) make each of the other viewports active and zoom-extents then zoom to a factor of 2 in each
  - b) make the 3D viewport active
- 5 The created model has five 'planes' so make five new layers: F1 red, F2 blue, F3 green, F4 magenta and F5 colour 14.
- 6 Make layer F1 current
- 7 Still with the 3D viewport active, menu bar with **Draw-Surfaces-3Dface** and:
 

```
prompt Specify first point
respond Intersection icon and pick pt1
prompt Specify second point
respond Intersection icon and pick pt2
prompt Specify third point
respond Intersection icon and pick pt3
prompt Specify fourth point
respond Intersection icon and pick pt4
prompt Specify third point
respond right-click/enter
```
- 8 Make layer F2 current and select the 3DFACE icon from the Surfaces toolbar and:
 

```
prompt Specify first point and pick Intersection pt2
prompt Specify second point and pick Intersection pt3
prompt Specify third point and pick Intersection pt5
prompt Specify fourth point and pick Intersection pt2 then <R>
```


- 9 Make layer F3 current, enter **3DFACE <R>** at the command line and:
 

```
prompt First point and pick Intersection pt3
prompt Second point and pick Intersection pt4
prompt Third point and pick Intersection pt6
prompt Fourth point and pick Intersection pt5 then <R>
```



**Figure 14.1** 3DFACE example 1.

- 10 Use the 3DFACE command and add faces to:
  - a) face: 1256 with layer F4 current
  - b) face: 146 with layer F5 current
- 11 Menu bar with **View-Hide** in each viewport – Fig. 14.1
- 12 Remove the hide effect with the menu bar sequence **View-RegenAll**
- 13 In each viewport select from the menu bar:
  - a) **View-Shade-Flat Shaded** to give a colour effect
  - b) **View-Shade-2D Wireframe** to remove the shade effect
- 14 Make MVLAY2 tab current then in each viewport
  - a) Zoom-Extents
  - b) Zoom to a factor of 3
- 15 Make MVLAY3 tab current and in each viewport zoom-extents then zoom to a factor of 3
- 16 Now make the MODEL tab active and:
  - a) Zoom-All the zoom to a factor of 1.5
  - b) Save the model as **MODR2002\CHEESE**
- 17 Task
  - a) Shade the model with the model tab active
  - b) Select the 3D Orbit icon from the Standard toolbar
  - c) Interactively rotate the 3D shade model to 'see' the surfaces not displayed in the four viewports
  - d) Select the Undo icon to restore the 3D view
- 18 This first 3D FACE exercise is now complete.
- 19 *Note*
  - a) The 3D orbit command will be discussed in detail in a later chapter. At this stage interactively rotate the model but holding down the left button and moving the pointing device
  - b) In this example I have referred to the MVLAY2 and MVLAY3 tabs. In general most of the future exercises will be completed with the MVLAY1 tab active, i.e. the traditional four viewport configuration. The other layout and model tabs will occasionally be mentioned. The user should investigate these tabs in their own time.



## 3D Face example 2

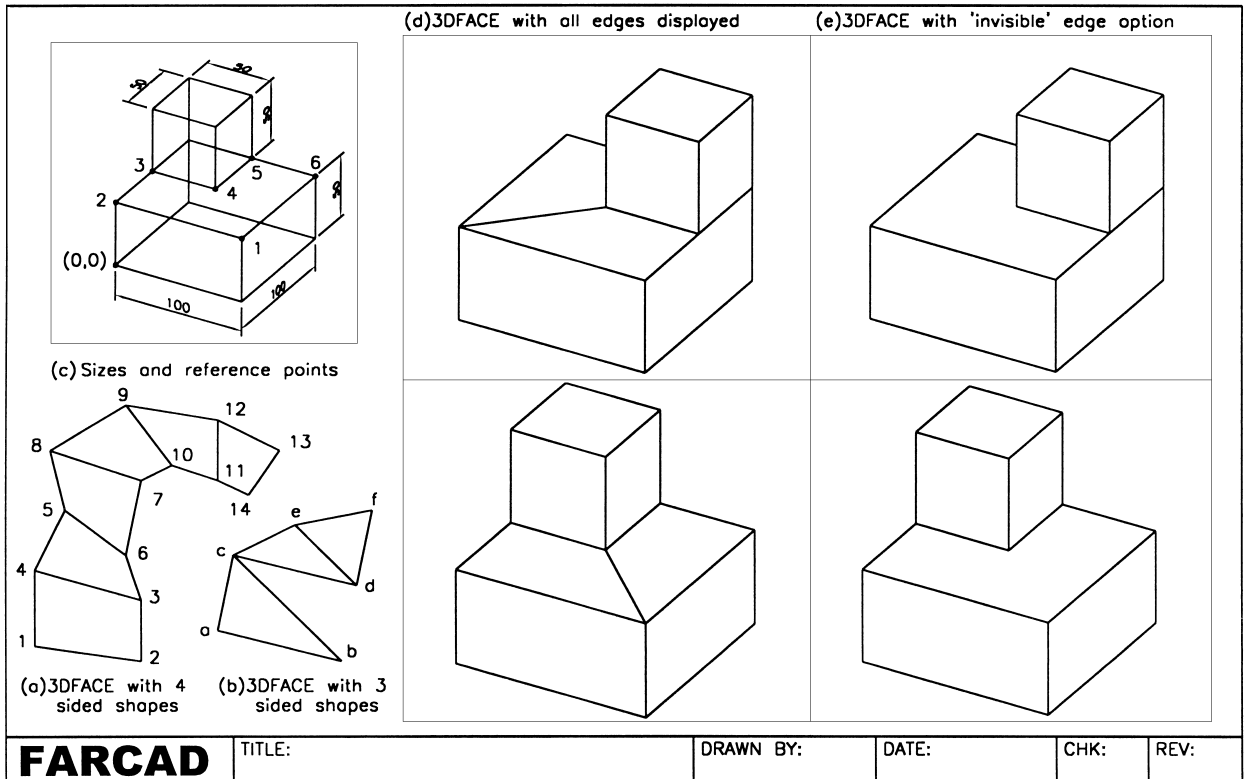
The 3DFACE command can be used to face any three or four sided 'plane'. The command allows 'continuous' faces to be created and will be demonstrated with a 2D example (although the procedure is valid in 3D) so:

- 1 With the MODEL tab active:
  - a) erase the model (saved?)
  - b) restore a plan view with the menu bar sequence **View-3D Views-Plan View-Current UCS**
  - c) set the snap to 5 and the grid to 10
  - d) zoom-all and refer to Fig. 14.2
- 2 Activate the 3DFACE command and:
 

*prompt* Specify first point or [Invisible]  
*respond* pick any pt1 on the screen  
*prompt* Specify second point or [Invisible]  
*respond* pick any pt2 on the screen  
*prompt* Specify third point or [Invisible]<exit>  
*respond* pick any pt3 (which is the 1st pt of the next face)  
*prompt* Specify fourth point or [Invisible]<create three-sided face>  
*respond* pick any pt4 (which is the 2nd pt of the next face)  
*and* Face 1-2-3-4 displayed  
*prompt* Specify third point  
*respond* pick any pt5 (3rd pt of face and 1st pt of next face)  
*prompt* Specify fourth point  
*respond* pick any pt6 (4th pt of face and 2nd pt of next face)  
*and* Face 3-4-5-6 displayed  
*prompt* Specify third point  
*respond* pick any pt7 (3rd pt of face and 1st pt of next face)  
*prompt* Specify fourth point  
*respond* pick any pt8 (4th pt of face and 2nd pt of next face)  
*and* Face 5-6-7-8 displayed  
*prompt* Specify third point..  
*respond* in response to the third and fourth point prompts:  
  - a) pick any points 9 and 10
  - b) then pick any points 11 and 12
  - c) then pick any points 13 and 14 in response
  - d) then right-click/enter

*and* Faces 7-8-9-10; 9-10-11-12; 11-12-13-14 will be displayed as fig(a)
- 3 Activate the 3DFACE command again and:
 

*prompt* Specify first point and pick any pta  
*prompt* Specify second point and pick any ptb  
*prompt* Specify third point and pick any ptc (1st of next face)  
*prompt* Specify fourth point and press <RETURN> – note the prompt  
*and* Face a-b-c displayed  
*prompt* Specify third point and pick any ptd (2nd pt of face)  
*prompt* Specify fourth point and pick any pte (3rd pt of face)  
*and* Face c-d-e displayed  
*prompt* Specify third point and pick any ptf  
*prompt* Specify fourth point and <R><R> – two returns  
*and* Face d-e-f displayed as fig(b)
- 4 Now erase the two continuous 3D faces.



**Figure 14.2** Continuous 3DFACE example.

### The invisible edge (3D Face example 3)

When the 3DFACE command is used with continuous three/four-sided 'shapes', all three/four sides of the face are displayed. It is possible to create a 3DFACE with an 'invisible edge' which will be demonstrated with the following example.

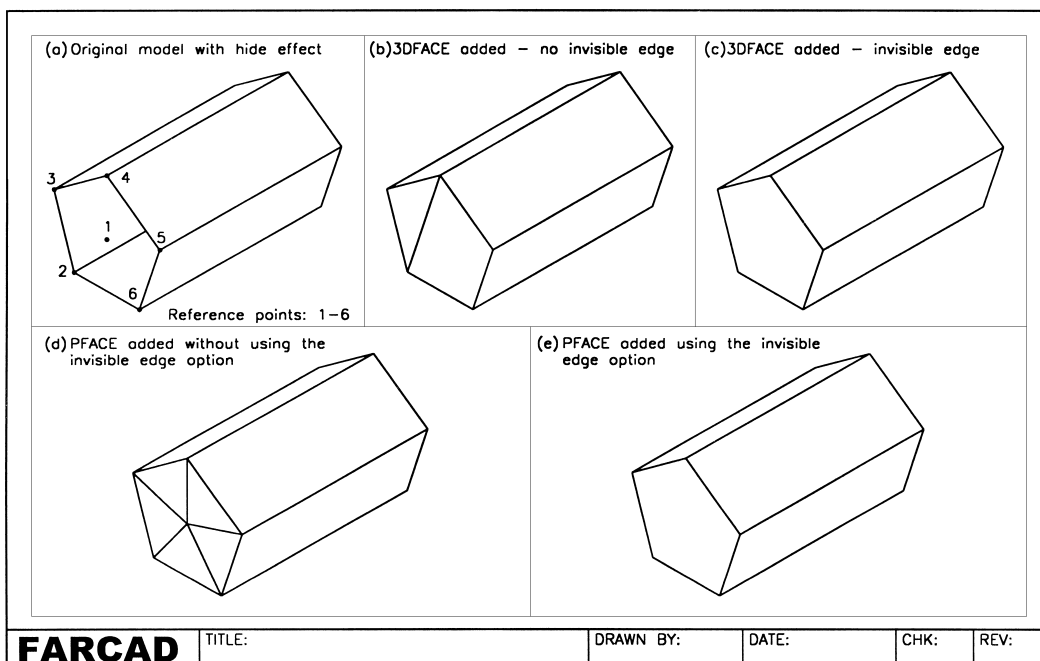
- Refer to Fig. 14.2(c) and with the Model tab still active:
  - make MVLAY1 tab active
  - enter model space
  - layer MODEL and UCS BASE current
  - toggle the grid and snap off
- Change the viewpoint in the following viewpoints:
  - lower left: VPOINT 'Rotate' at 300degs and 30degs
  - upper left: VPOINT 'Rotate' at 30degs and 30 degs
  - upper right and lower right: your own VPOINT 'Rotate' values
- Set the running object snap to INTERSECTION
- Create a 3D wire-frame model using the sizes given. This should be easy for you (?) but use your discretion if unsure of sizes
- With layer F1 current, 3DFACE the four vertical sides of the lower part of the model
- With layer F3 current, 3DFACE the four vertical sides of the upper part of the model
- With layer F4 current, 3DFACE the top of the model

- 8 With layer F2 current:
  - a) activate the 3DFACE command and:
    - prompt* Specify first point and pick pt1
    - prompt* Specify second point and pick pt2
    - prompt* Specify third point and pick pt3
    - prompt* Specify fourth point and pick pt4 the right-click/enter
  - b) activate the 3DFACE command and:
    - prompt* Specify first point and pick pt1
    - prompt* Specify second point and pick pt6
    - prompt* Specify third point and pick pt5
    - prompt* Specify fourth point and pick pt4 then right-click/enter
- 9 Menu bar with **View-Hide** and the model will be displayed with hidden line removal. The edge between point 1 and 4 is visible as displayed in fig(d)
- 10 Menu bar with **View-Shade-Flat Shaded** and the model is displayed with coloured 3D faces. No edge is visible between points 1 and 4 due to the two added faces having the same colour
- 11 Restore the original wire-frame model with the menu bar sequence **View-Shade-2D Wireframe**
- 12 Erase the two blue 3D face surfaces by picking the edges between points 1 and 4
- 13 Still with layer F2 current and running object snap set:
  - a) activate the 3DFACE command and:
    - prompt* Specify first point and pick pt1
    - prompt* Specify second point and pick pt2
    - prompt* Specify third point and pick pt3
    - prompt* Specify fourth point
    - enter* **I <R>** – the invisible edge option
    - prompt* Specify fourth point and pick pt4
    - prompt* Specify third point and right-click/enter
    - and* the 3D face is displayed without edge 1–4
  - b) 3DFACE again and:
    - prompt* Specify first point and pick pt1
    - prompt* Specify second point and pick pt6
    - prompt* Specify third point and pick pt5
    - prompt* Specify fourth point and enter: **I <R>**
    - prompt* Specify fourth point pick pt4
    - prompt* Specify third point and right-click/enter
    - and* the 3D face is displayed without edge 1–4
- 14 Display the model with hidden line removal – fig(e)
- 15 Try some different shade effects then restore the original wire- frame model
- 16 Cancel the object snap intersection mode.
- 17 Save if required, but it will not be used again.

## PFACE

A PFACE is a polygon mesh and is similar to a 3DFACE. It allows the user to define a number of vertices for the surface to be faced, not the 3 or 4 allowed with the 3DFACE command. The following example has a rather long set of prompts:

- 1 Open your MV3DSTD template file and:
  - a) MVLAY1 tab active
  - b) make four new layers: F1 blue, F2 green, F3 magenta, F4 cyan
  - c) layer MODEL current, restore UCS FRONT, OSNAP INT
  - d) lower left viewport (3D) active
- 2 Set the elevation to 0 and the thickness to -200
- 3 Select the POLYGON icon from the Draw toolbar and:
  - a) number of sides: 5
  - b) centre of polygon: 0,0
  - c) circumscribed circle radius: 50
- 4 Zoom centre about the point 0,0,-100 at 225 magnification
- 5 Refer to Fig. 14.3 which only displays the 3D viewport of the model
- 6 Menu bar with **View-Hide** to display the pentagonal prism without a 'top surface' as fig(a)
- 7 Set the object snap to Intersection, make layer F1 current and:
  - a) select the 3DFACE icon and:
    - First point pick pt2
    - Second point pick pt3
    - Third point pick pt4
    - Fourth point <R><R> - two returns
  - b) select the 3DFACE icon and:
    - First point pick pt2
    - Second point pick pt6
    - Third point pick pt5
    - Fourth point pick pt4
    - Third point right-click/enter


**FARCAD**

TITLE:

DRAWN BY:

DATE:

CHK:

REV:

**Figure 14.3** 3DFACE example.

- 8 Hide the model to display the 'top surface' as fig(b) i.e. with hidden line removal. Restore the original wire-model from the menu bar with View-Regen
- 9 Make layer F2 current and freeze layer F1 then:
- a) activate the 3DFACE command and:
    - First point pick pt2
    - Second point pick pt3
    - Third point enter **I <R>** then pick pt4
    - Fourth point <R><R> – two returns
  - b) 3DFACE again and:
    - First point pick pt2
    - Second point pick pt6
    - Third point pick pt5
    - Fourth point enter **I <R>** then pick pt4
    - Third point right-click/enter
- 10 Hide to display the model with the 3D face invisible edge option as fig(c)
- 11 Make layer F3 current and freeze layer F2
- 12 At the command line enter **PFACE <R>** and:
- ```

prompt Specify location for vertex 1 and enter: 0,0 <R> i.e. pt1
prompt Specify location for vertex 2 and pick pt2
prompt Specify location for vertex 3 and pick pt3
prompt Specify location for vertex 4 and pick pt4
prompt Specify location for vertex 5 and pick pt5
prompt Specify location for vertex 6 and pick pt6
prompt Specify location for vertex 7 and right-click: no more vertices
prompt Face 1,vertex 1
then Enter a vertex number and enter1 <R>
and Enter the following in response to the vertex number
prompts:
prompt Face 1,vertex 2 and enter: 2 <R>
prompt Face 1,vertex 3 and enter: 3 <R>
prompt Face 1,vertex 4 and <RETURN> i.e. end of face 1
prompt Face 2, vertex 1 and enter: 1
prompt Face 2, vertex 2 and enter: 3
prompt Face 2, vertex 3 and enter: 4
prompt Face 2, vertex 4 and <RETURN> i.e. end of face 2
prompt Face 3, vertex 1 and enter: 1
prompt Face 3, vertex 2 and enter: 4
prompt Face 3, vertex 3 and enter: 5
prompt Face 3, vertex 4 and <RETURN> i.e. end of face 3
prompt Face 4, vertex 1 and enter: 1
prompt Face 4, vertex 2 and enter: 5
prompt Face 4, vertex 3 and enter: 6
prompt Face 4, vertex 4 and <RETURN> i.e. end of face 4
prompt Face 5, vertex 1 and enter: 1
prompt Face 5, vertex 2 and enter: 6
prompt Face 5, vertex 3 and enter: 2
prompt Face 5, vertex 4 and <RETURN> i.e. end of face 5
prompt Face 6, vertex 1 and <RETURN> to end command.

```
- 13 Menu bar with View-Hide to display the end of the prism with a PFACE surface – fig(d)
- 14 Make layer F4 current and freeze layer F3

15 Repeat the PFACE command line entry and:

```

prompt Specify location for vertex 1 and enter: 0,0 <R>
prompt Specify location for vertex 2 and pick pt2
prompt Specify location for vertex 3 and pick pt3
prompt Specify location for vertex 4 and pick pt4
prompt Specify location for vertex 5 and pick pt5
prompt Specify location for vertex 6 and pick pt6
prompt Specify location for vertex 7 and <RETURN>
prompt Face 1, vertex 1, Enter a vertex number and enter: -1 <R>
prompt Face 1, vertex 2 and enter: 2 <R>
prompt Face 1, vertex 3 and enter: -3 <R>
prompt Face 1, vertex 4 and <RETURN>
prompt Face 2, vertex 1 and enter: -1
prompt Face 2, vertex 2 and enter: 3
prompt Face 2, vertex 3 and enter: -4
prompt Face 2, vertex 4 and <RETURN>
prompt Face 3, vertex 1 and enter: -1
prompt Face 3, vertex 2 and enter: 4
prompt Face 3, vertex 3 and enter: -5
prompt Face 3, vertex 4 and <RETURN>
prompt Face 4, vertex 1 and enter: -1
prompt Face 4, vertex 2 and enter: 5
prompt Face 4, vertex 3 and enter: -6
prompt Face 4, vertex 4 and <RETURN>
prompt Face 5, vertex 1 and enter: -1
prompt Face 5, vertex 2 and enter: 6
prompt Face 5, vertex 3 and enter: -2
prompt Face 5, vertex 4 and <RETURN>
prompt Face 6, vertex 1 and <RETURN>

```

16 Now View-Hide to display the model with the hidden edge option of the PFACE command.

17 The negative (-1, etc.) entry for the face/vertex prompt is the invisible edge option. Think this out for the vertices being entered

18 Save if required, but this model will not be used again.

Summary

- 1 The 3DFACE and PFACE commands allow **surface models** to be created by drawing 'skins' over wire-frame models
- 2 The HIDE command allows surface models to be displayed with hidden line removal
- 3 The 3DFACE command can only be used with three or four sided 'shapes' although continuous faces can be created
- 4 The PFACE command can be used with multi-sided figures
- 5 Both commands have an invisible edge option
- 6 The commands can be activated:
 - a) 3DFACE: command line, menu bar or icon
 - b) PFACE: command line only
- 7 Using separate coloured layers for different faces allows models to be displayed in colour using the SHADE command
- 8 If the HIDE command has been used, the original model can be restored with:
 - a) View-Regen: in the active viewport
 - b) View-Regenall for all viewports
- 9 If the SHADE command has been used then REGEN will not restore the original model. The menu bar sequence View-Shade-2D Wireframe must be selected. This must be applied in the viewports which used the SHADE command.

Assignment

One of MACFARAMUS's structures was a temple created from a series of hexagonal shaped columns, these prisms having two different sloped surfaces. It is one of these columns which you have to create as a 3DFACED surface model.

Activity 10: Hexagonal column of MACFARAMUS.

- 1 Start with your MV3DSTD template/drawing file
- 2 MVLAY1 tab, layer MODEL current and UCS BASE current
- 3 Create a wire-frame model of the hexagonal prism using the sizes given then zoom-centre about 0,0,75 at 200 magnification
- 4 Make three new coloured layers for:
 - a) vertical sides: blue
 - b) slopes: green
 - c) horizontal surfaces: red
- 5 Use the 3DFACE command with the coloured layers to convert the wire-frame model into a surface model. Note that two of the vertical sides have five vertices while all the other surfaces to be faced have four (except of course the base)
- 6 Freeze layer MODEL, then hide and shade the model
- 7 Save as **MODR2002\HEXCOL**


3DMESH

A 3D mesh (or more correctly, a 3D polygon mesh) consists of a series of 3D faces in a rectangular array pattern. This mesh matrix is defined by **M x N vertices** and:

- a) M is the number of 'columns' in the x direction
- b) N is the number of 'rows' in the y direction
- c) the user enters the x,y,z coordinates of every vertex in the matrix.

3D mesh example

- 1 Open your MV3DSTD template file with MVLAY1 tab, layer MODEL and UCS BASE current
- 2 Enter model space and make the lower right viewport active
- 3 Display the Surfaces toolbar and refer to Fig. 15.1
- 4 Zoom-centre about the point 150,150,25 at 250 magnification
- 5 Select the 3DMESH icon from the Surfaces toolbar and:



```

prompt Enter size of mesh in M direction
enter 5 <R>
prompt Enter size of mesh in N direction
enter 4 <R>
prompt Specify location for vertex (0,0)
enter 50,50,30 <R>
prompt Specify location for vertex (0,1)
enter 50,120,15 <R>
prompt Specify location for vertex (0,2)
respond refer to Fig. 15.1 and enter the required coordinates in response to the vertex prompts
      
```
- 6 When the last vertex coordinate is entered – Vertex (4,3), the 3D mesh will be displayed as fig(a)

Modifying the mesh

Two of the vertices have been entered wrongly, these being:

- a) vertex (0,3): entered value of (50,200,-25)
- b) vertex (4,0): entered value of (260,90,100)

To modify the mesh:

- 1 Menu bar with **Modify-Object-Polyline** and:


```

prompt Select polyline
respond pick any point on the mesh
prompt Enter an option [Edit vertex/Smooth surface/..
enter E <R> – the edit vertex option
prompt Current vertex (0,0) and a small X displayed at vertex (0,0)
then Enter an option [Next/Previous/Left/Right/..
      
```

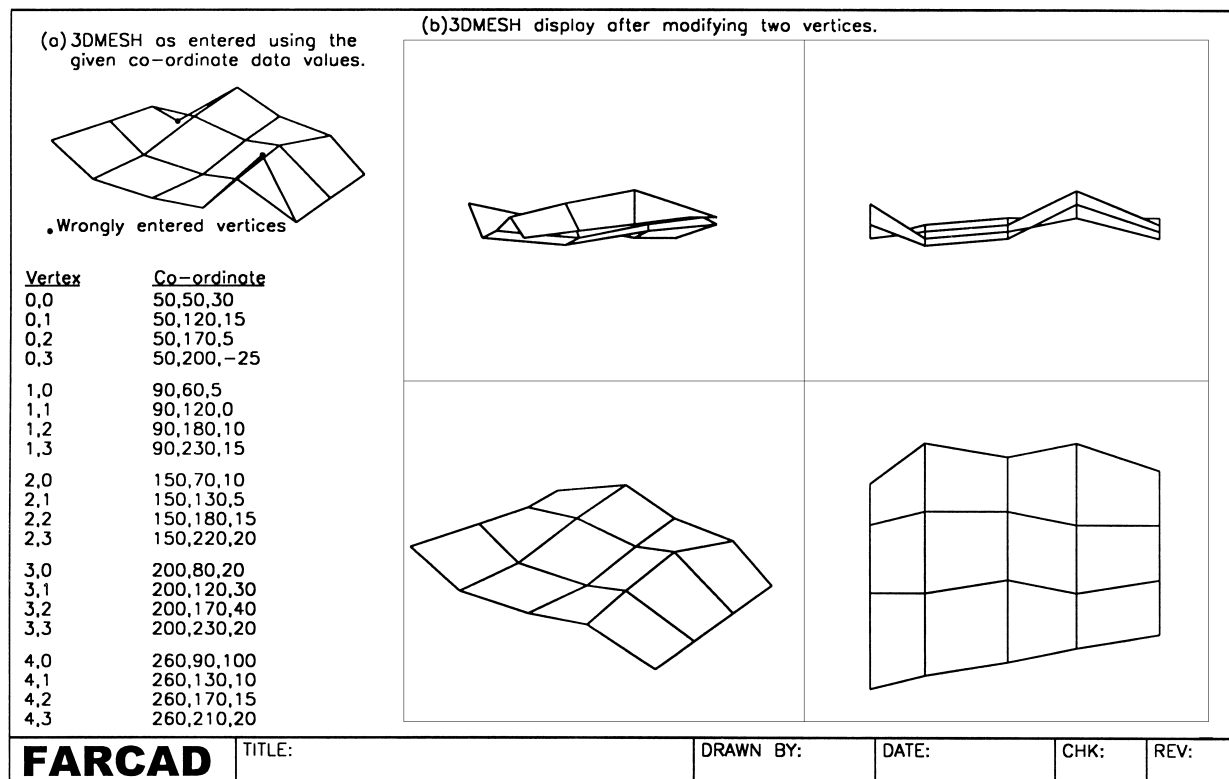


Figure 15.1 3DMESH exercise.

and a small X displayed at vertex (0,0)
 enter **N <R>** until current vertex is (0,3)
 prompt Enter an option [Next/Previous/..
 enter **M <R>** – the move vertex option
 prompt Specify new location for marked vertex
 enter **50,50,5 <R>** – absolute coordinate entry
 prompt Current vertex (0,3) Enter an option [Next/Previous..
 enter **N <R>** until current vertex is (4,0)
 prompt Enter an option [Next/Previous/..
 enter **M <R>**
 prompt Specify new location for marked vertex
 enter **@0,0,-95 <R>** – relative coordinate entry
 prompt Current vertex (4,0) Enter an option [Next/Previous/..
 enter **X <R>** – to exit edit vertex options
 prompt Enter an option [Edit vertex/Smooth surface/..
 and **right-click/enter** – to exit command

- 2 The mesh will be displayed as fig(b), with the two wrong vertices have been 'repositioned'
- 3 Use the hide command and note the effect
- 4 Save the drawing as the exercise is now complete.

Notes

- 1 This example has been a brief introduction to the 3DMESH command
- 2 The command requires the user to enter all vertex values as coordinates and is therefore very tedious to use. You can also reference existing objects if these are displayed
- 3 The Modify-Polyline command can be used with 3D meshes. This is the same as the 2D modify-polyline command
- 4 Selecting MESH from the 3D Objects dialogue box, will allow the user to create a 2D mesh by:
 - a) specifying the four corners of the mesh
 - b) entering the M and N sizes
- 5 The 3D MESH values M and N can be between 2 and 256
- 6 A 3D MESH is a single object
- 7 The command can be activated:
 - a) by icon selection from the Surfaces toolbar
 - b) from the menu bar with Draw-Surfaces
 - c) by entering 3DMESH at the command line.

Ruled surface

A ruled surface is a polygon mesh created between two defined boundaries selected by the user. The objects which can be used to define the boundaries are lines, arcs, circles, points and 2D/3D polylines. The surface created is a 'one-way' mesh of straight lines drawn between the two selected boundaries. The number of straight line meshes is controlled by the system variable **SURFTAB1** which has an initial value of 6.

The ruled surface effect will be demonstrated by worked examples, the first being in 2D to allow the user to become familiar with the basic terminology.

Example 1

- 1 Begin a new 2D drawing from scratch (metric) and create two layers, MOD (red) and RULSUR (blue). Refer to Fig. 16.1
- 2 Display the Draw, Modify and Surfaces toolbars

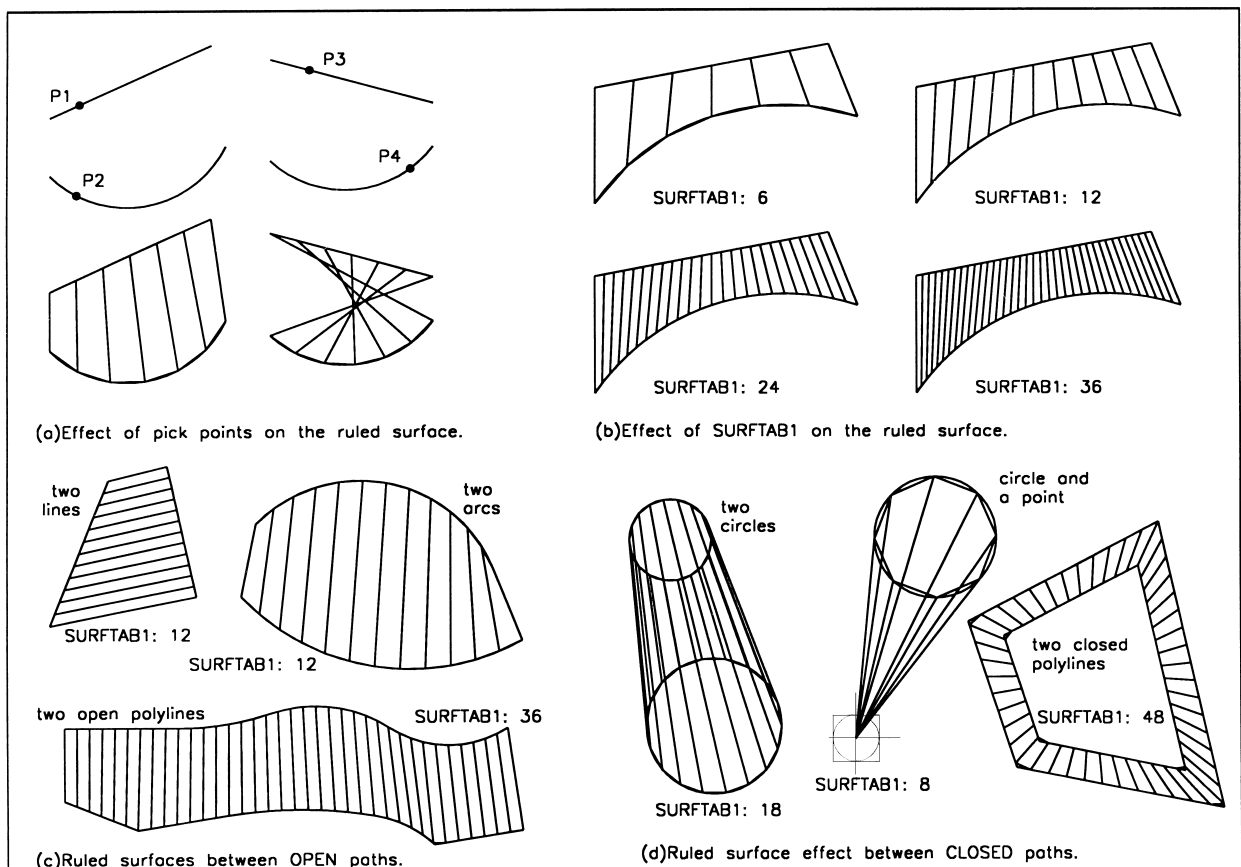



Figure 16.1 Ruled surface example 1 – usage and basic terminology.

Pick points effect

- 1 With layer MOD current, draw two lines and two three point arcs as fig(a) then make layer RULSUR current
- 2 Select the RULED SURFACE icon from the Surfaces toolbar and:



```

prompt Select first defining curve
respond pick a point P1 on the first line
prompt Select second defining curve
respond pick a point P2 on the first arc
and a blue ruled surface is drawn between the two objects
      
```
- 3 Menu bar with **Draw-Surfaces-Ruled Surface** and:
 - a) first defining curve prompt: pick point P3 on second line
 - b) second defining curve prompt: pick point P4 on second arc
 - c) ruled surface drawn between the line and arc
- 4 The ruled surface drawn between selected objects is thus dependent on the pick point positions.

Effect of the SURFTAB1 system variable

- 1 With layer MOD current draw a line and three point arc as fig(b)
- 2 Copy the line and arc to three other places on screen
- 3 Make layer RULSUR current
- 4 At the command line enter **SURFTAB1 <R>** and:


```

prompt Enter new value for SURFTAB1<?>
enter 6 <R>
      
```
- 5 At the command line enter **RULESURF <R>** and:


```

prompt Select first defining curve
respond pick a point on the first line
prompt Select second defining curve
respond pick a point on the first arc
      
```
- 6 By entering SURFTAB1 at the command line, enter new values of 12, 24 and 36 and add a ruled surface between the other lines and arcs.
- 7 *Note*
 - a) The system variable **SURFTAB1** controls the display of the ruled surface effect, i.e. the number of 'strips' added between the defining curves.
 - b) The default value is 6.
 - c) The value of SURFTAB1 to be used is dependent on the 'size' of the defining curves.

Open paths

- 1 An open path is defined as a line, arc or open polyline
- 2 With layer MOD current draw some open paths as fig(c)
- 3 Using the ruled surface command and with SURTFAB1 set to your own value, add ruled surfaces between the drawn open paths.

Closed paths

- 1 A closed path is defined as a circle or closed polyline.
- 2 Draw some closed paths as fig(d) and add ruled surfaces between them.

Note

- 1 A ruled surface can only be drawn/added between:
 - a) TWO OPEN paths
 - b) TWO CLOSED paths
- 2 A ruled surface **cannot** be created between an open and a closed path. If a line and a circle are selected as the defining curves, the following message will be displayed:
Cannot mix closed and open paths
- 3 A point can be used as a defining curve with either an open path (e.g. line) or closed path (e.g. circle)
- 4 The defining curves are also called boundaries
- 5 This first exercise is now complete and need not be saved.

Example 2

- 1 Open your MV3DSTD template file and refer to Fig. 16.2. Note that in Fig. 16.2 I have only displayed the 3D viewpoint.

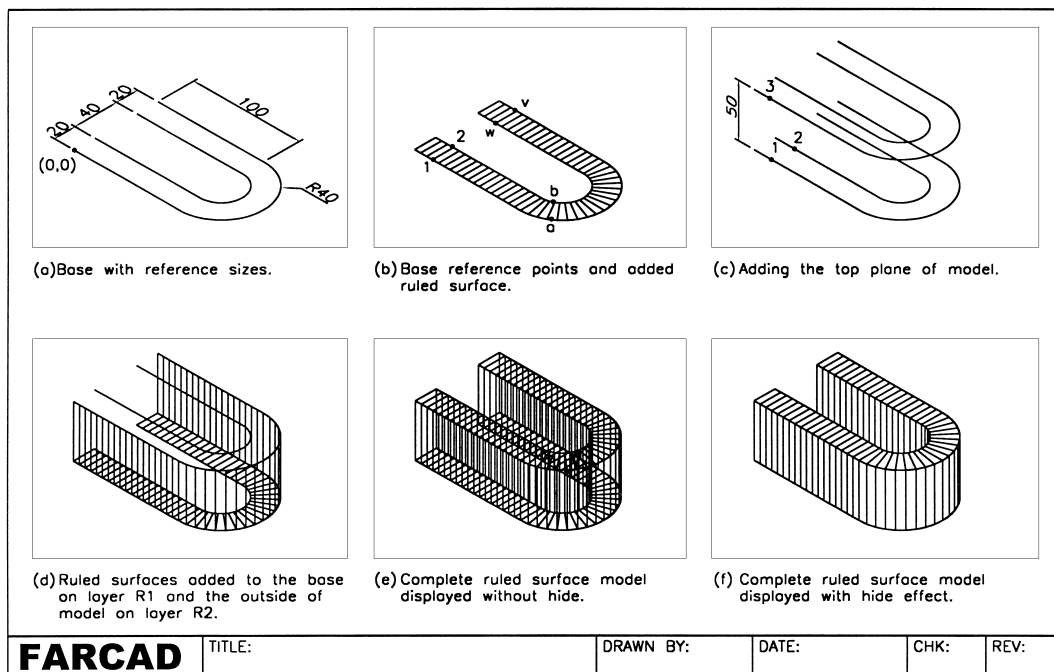


Figure 16.2 Ruled surface example 2 – 3D wire-frame model.

- 2 With MVLAY1 tab, layer MODEL and UCS BASE current, zoom centre about the point 70,40,25 at 150 magnification in all viewpoints
- 3 Create the model base from lines and trimmed circles using the sizes given in fig(a). Use the (0,0) start point indicated.
- 4 Make a new layer RULSRF, colour blue and current
- 5 Set the SURFTAB1 system variable to 18
- 6 Using the ruled surface icon (three times) from the Surfaces toolbar, select the following defining curves:
 - a) lines 1 and 2
 - b) arcs a and b
 - c) lines v and w
 - d) effect as fig(b)

- 7 Erase the ruled surface and create the top surface of the model by copying the base objects:
 - a) from the point 0,0,0
 - b) by @0,0,50 – fig(c)
- 8 With layer RULSRF still current, select the ruled surface icon and select the following defining curves as fig(c):
 - a) lines 1 and 2 – ruled surface added
 - b) lines 3 and 1 – no ruled surface added and following message displayed:
Object not usable to define ruled surface – why?
 - c) Explanation: when the second set of defined curves was being selected:
 1. point 3 was picked satisfactorily
 2. point 1 could not be picked – you were picking the previous ruled surface added between lines 1 and 2
 - d) cancel the ruled surface command (ESC) and erase the added ruled surface
- 9 Make the following four new layers:
R1 – red; R2 – blue; R3 – green; R4 – magenta
- 10 a) Make layer R1 current
b) Add a ruled surface to the base of the model (three needed)
- 11 a) Make layer R2 current
b) Freeze layer R1
c) Add a ruled surface to the three ‘outside’ vertical planes of the model
d) Thaw layer R1 – fig(d)
- 12 a) Make layer R3 current
b) Freeze layers R1 and R2
c) Rule surface the top three defining curves of the model
- 13 a) Make layer R4 current and freeze layer R3
b) Add a ruled surface to the three ‘inside’ vertical planes
- 14 a) Thaw layers R1, R2 and R3
b) Model displayed a fig(e)
- 15 Menu bar with **View-Hide** to give fig(f)
- 16 Menu bar with **View-Shade-Gouraud Shaded** – impressive?
- 17 Return the model to wire-frame then save as **MODR2002\RSRF1**, it may be used in a later exercise
- 18 *Note*
When the ruled surface command is being used with adjacent surfaces, it is recommended that:
 - a) a layer be made for each ruled surface to be added
 - b) once a ruled surface has been added, that layer should be frozen before the next surface is added
 - c) the new surface layers should be coloured for effect
- 19 *Task*
 - a) Try the 3D orbit with the 3D viewport active
 - b) The two ‘ends’ of the model are ‘open’. A 3DFACE could be added to these ends?
 - c) The original model was created from lines and circles/arcs. The base could have been created from a single polyline and then offset. Try this and add a ruled surface and note that only one set of defining curves is required. What about SURFTAB1 with a polyline?

Example 3

This example will investigate how a ruled surface can be added to a surface which has a circular/slotted hole in it. The example will be in 2D, but the procedure is identical for a 3D model.

- 1 Begin a new 2D metric drawing from scratch and refer to Fig. 16.3
- 2 Make two new layers, MOD red (current) and RULSRF blue and set SURFTAB1 to 24
- 3 Using the LINE icon draw a square of side 60 with a 15 radius circle at the square 'centre' – snap on helps
- 4 Using the Ruled Surface icon, pick any line of the square and the circle as the defining curves. No ruled surface can be added because of the open/closed path effect – fig(a)
- 5 With the Polyline icon, draw a 60 sided square from 1–2–3–4-<R> as fig(b) and draw the 15 radius circle. Add a ruled surface and the open/closed path message is displayed and no ruled surface is added.
- 6 Draw a closed polyline square using the points 1–2–3–4–close in the order given in fig(c). Draw the circle. With the ruled surface icon pick the defining curves indicated and a ruled surface is added, but not as expected.
- 7 Draw a 60 sided square as a closed polyline and select the points 1–2–3–4–close in the order given in fig(d). Draw the circle then add a ruled surface picking the defining curves indicated. The added ruled surface is not quite 'correct' at the circle.

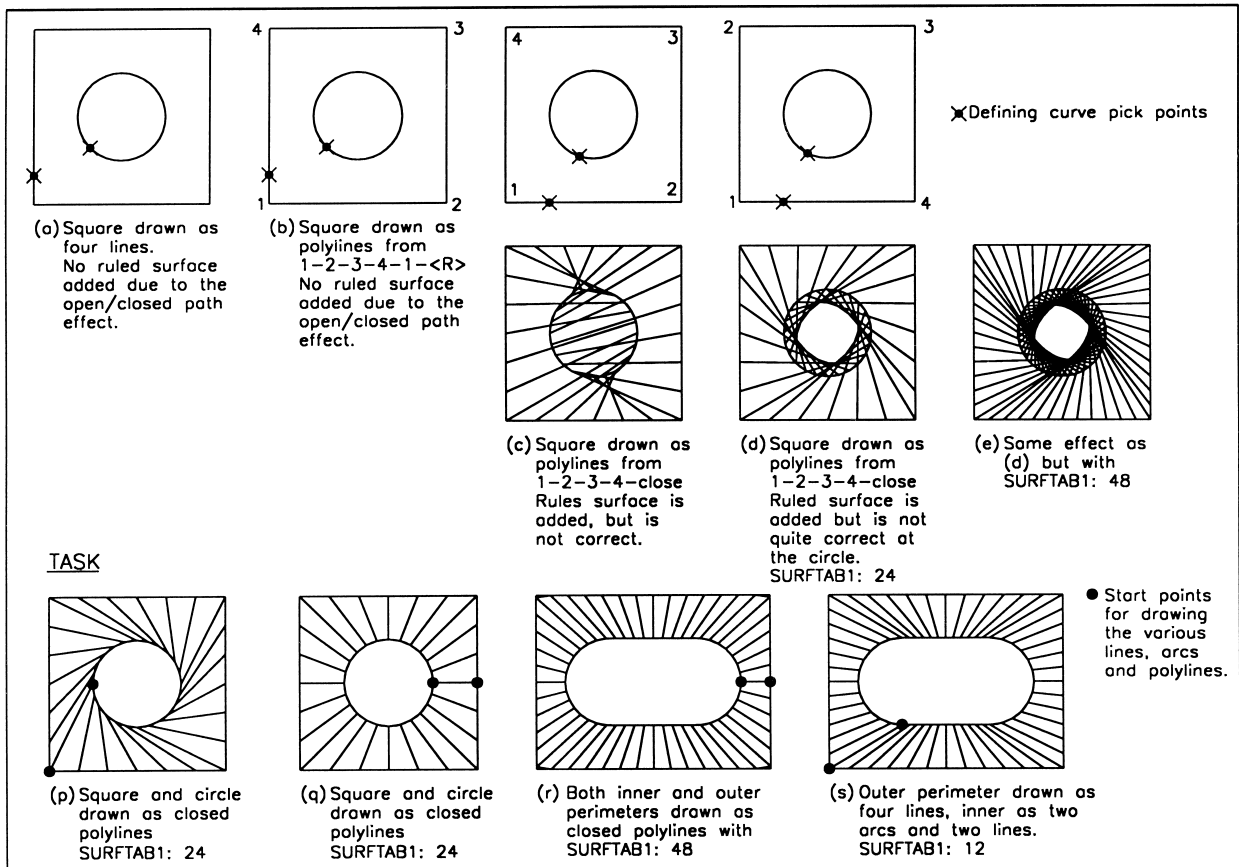


Figure 16.3 Ruled surface example 3 – polylines and circles.

- 8 Erase the ruled surface effect, set SURFTAB1 to 48 and repeat the ruled surface command to give fig(e). Set SURFTAB1 back to 24.
- 9 *Note*
- a) When a ruled surface is added between two defined curves, the surface 'begins at the defined curve start points'. It is thus essential that the defined curves are:
- 1 DRAWN IN THE SAME DIRECTION**
2 DRAWN FROM THE SAME 'RELATIVE' START POINT
- b) Circular holes require to be drawn as two closed polyarcs
- 10 *Task*
- Using the information given in step 9, add ruled surfaces to the following models displayed in Fig. 16.3:
- a) fig(p): square drawn as a closed polyline and circle drawn as two closed polyarcs. Note start points
- b) fig(q): square drawn as a closed polyline and circle drawn as two closed polyarcs. Note that the start points differ from those in fig(p)
- c) fig(r): both the outer and inner perimeters are drawn as closed polylines/polyarcs. Note the start points
- d) fig(s): the outer perimeter is drawn as four lines, and the inner as two arcs and two lines.
- 11 When this task is complete, the exercise is finished. Save?

Example 4

A ruled surface is one of the most effective surface modelling techniques, and I have included another 3D model to demonstrate how it is used. The procedure when adding a ruled surface is basically the same with all models, this being:

- a) create the 3D wire-frame model
 b) make new coloured layers for the surfaces to be added
 c) use the ruled surface command with layers current as required.

- 1 Open your MV3DSTD template file and refer to Fig. 16.4
- 2 Make four new layers, R1 red, R2 blue, R3 green and R4 magenta
- 3 With MVLAY1 tab and layer MODEL current, restore UCS FRONT and make the lower left (3D) viewport active.
- 4 Select the POLYLINE icon and draw:
Start point: 0,0
Next point: @0,100
Next point: **Arc option**, i.e. enter A <R>
Arc endpt: @50,50 then right-click/enter
- 5 Centre each viewport about the point 50,75,0 at 175 mag
- 6 Offset the polyline by 20 'inwards'
- 7 Copy the two polylines from: 0,0, by: @0,0,-20
- 8 Change the viewport in the lower left viewport with the rotate option and angles:
 a) first prompt: 300
 b) second prompt: 30
- 9 Set SURFTAB1 to 18
- 10 Making each layer R1-R4 current, add a ruled surface to each 'side' of the model, remembering to freeze layers as in the second example.

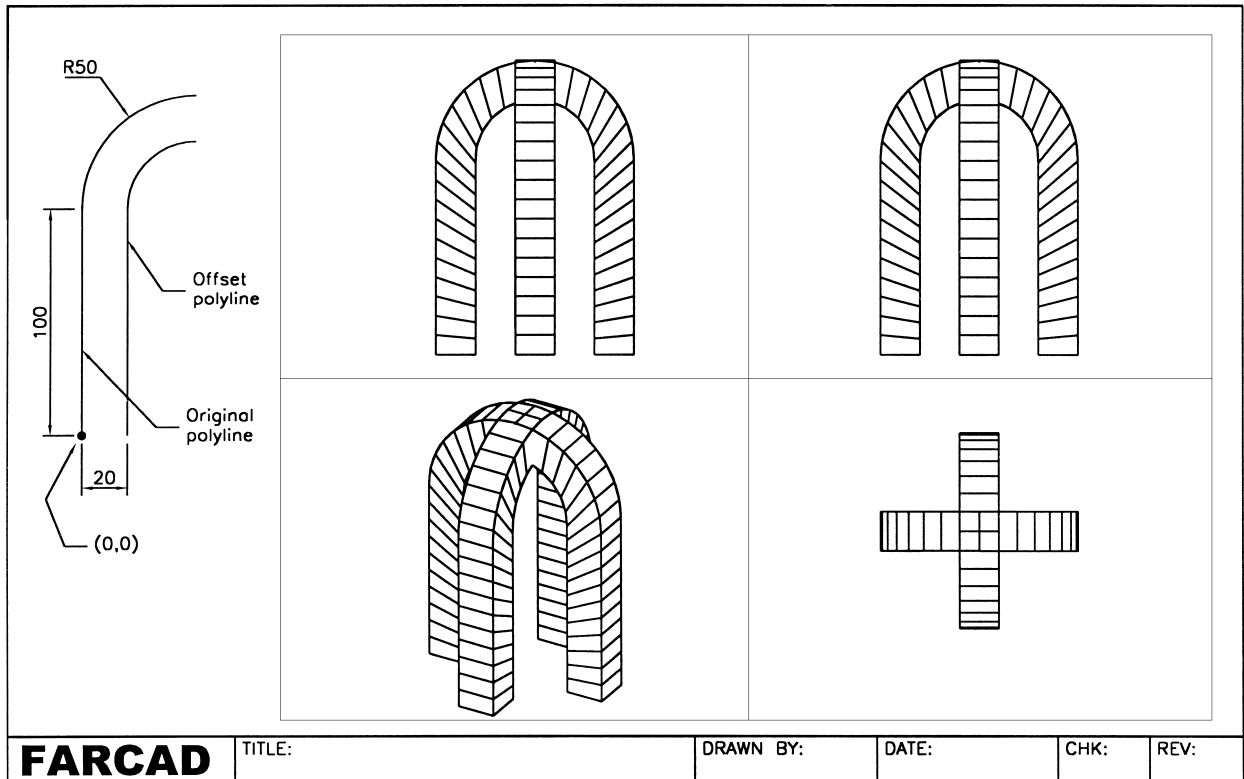


Figure 16.4 Ruled surface example 4 – ARCHES.

- 11 Restore UCS BASE and polar array the complete model (crossing selection) using:
 - a) Method: Total number of items & Angle to fill
 - b) Centre point: X: 50 and Y: 10
 - c) Total number of items: 4
 - d) Angle to fill: 360
 - e) Rotate items as copied: active
- 12 Hide, shade etc – impressive result?
- 13 Save the complete model as MODR2002\ARCHES for future recall
- 14 *Note*
The top 'square' of the arrayed arches – comments?

Summary

- 1 A ruled surface can be added between lines, circles, arcs, points and polylines
- 2 The command can be activated in icon form, from the menu bar or by keyboard entry
- 3 The command can be used in 2D or 3D
- 4 A ruled surface **CAN ONLY** be added between:
 - a) two open paths, e.g. lines, arcs, polylines (not closed)
 - b) two closed paths, e.g. circles, closed polylines
- 5 Points can be used with open and closed paths

- 6 With closed paths, the correct effect can only be obtained if:
 - a) the paths are drawn in the same direction
 - b) the paths start at the 'same relative point'
- 7 The system variable SURFTAB1 controls the number of ruled surface 'strips' added between the two defining curves.
- 8 The default SURFTAB1 value is 6

Assignment

Activity 11: Ornamental flower bed of MACFARAMUS

MACFARAMUS designed some interesting artefacts for the famous lost city of CADOPOLIS. One of his least known creations has the 'hanging gardens' for which he made several unusual ornamental flower beds. It is one of these which you have to create as a 3D ruled surface model, the procedure being the same as in the examples:

- 1 Open your MV3DSTD template file, MVLAY1 tab, layer MODEL active
- 2 Create the wire-frame model from lines and trimmed circles using the sizes given with the (0,0) start point. The vertical R50 arch requires the UCS RIGHT to be current and the R30 side curve requires UCS FRONT. Use your discretion for any sizes omitted.
- 3 With UCS BASE, zoom centre about 90,50,50 at 200 mag.
- 4 Make four coloured layers
- 5 Add ruled surfaces to the 'four sides' of the model using the four new layers correctly. Use a SURFTAB1 value of 18 for most of the defining curves, but 6 for the 'side' line/arc selection.
- 6 Hide, shade, 3D orbit, save.
- 7 *Note*
 - a) I suggest that you enter paper space and zoom-window the lower left viewport then return to model space. This will make creating the wire-frame model and selecting the defining curves easier
 - b) As an alternative to (a), create the model with the MODEL tab active

Tabulated surface

A tabulated surface is a parallel polygon mesh created along a path, the user defining:

- a) the **path curve** – the profile of the final model
- b) the **direction vector** – the ‘depth’ of the profile

The following are important points to note when creating a tabulated surface:

- 1 The path curve can be created from lines, arcs, circles, ellipses, splines or 2D/3D polylines
- 2 The direction vector **MUST** be a line or an open 2D/3D polyline
- 3 The system variable SURFTAB1 determines the ‘appearance’ of curved tabulated surfaces.

Example

- 1 Open your MV3DSTD template file with MVLAY1 tab and layer MODEL current, lower left viewport active and UCS BASE. Display toolbars to suit.
- 2 Refer to Fig. 17.1 (which only displays the 3D viewport) and draw two lines:
 - a) start point: 0,0,0 next point: @0,0,120
 - b) start point: 0,0,0 next point: @-150,0,0

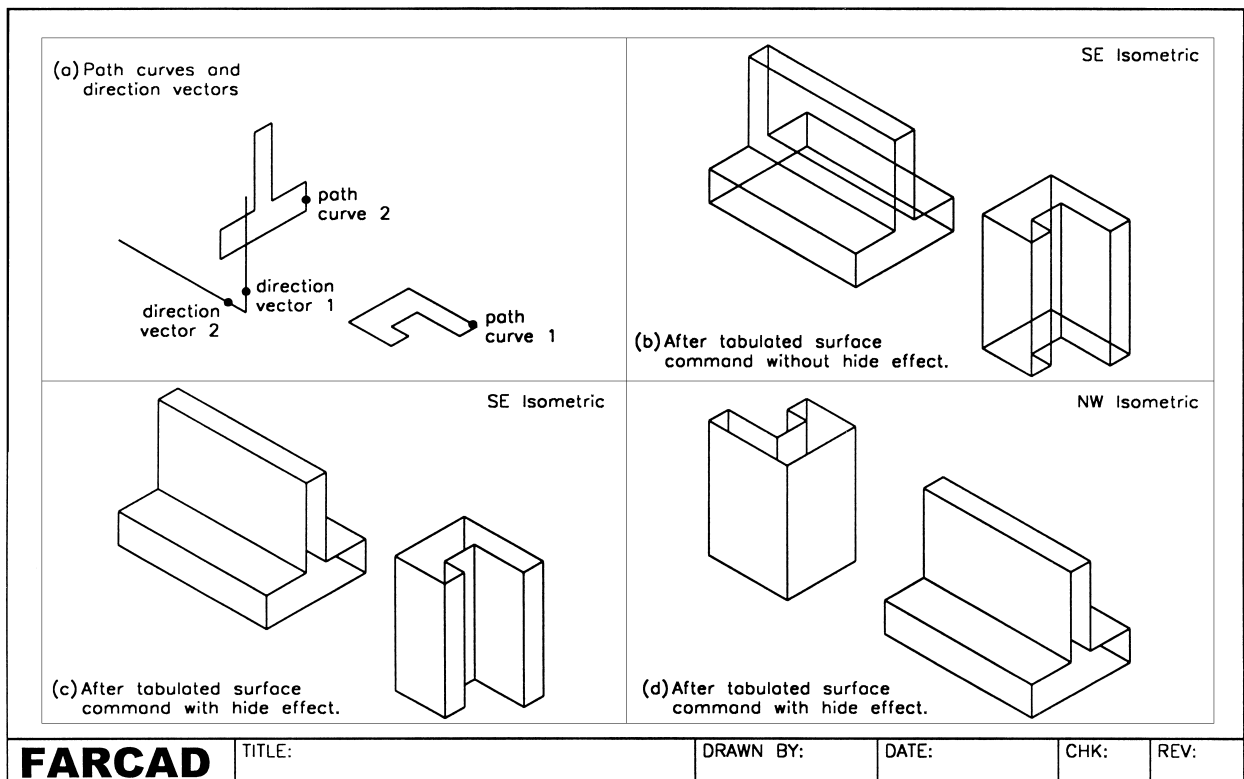


Figure 17.1 Tabulated surface example.

- 3 Restore the appropriate UCS and draw two closed polylines with the following coordinate data:

<i>UCS BASE</i>	<i>UCS RIGHT</i>
Start: 70,50,0	Start: 20,20,-50
Next: @50,0	Next: @100,0
Next: @0,20	Next: @0,30
Next: @-20,0	Next: @-40,0
Next: @0,30	Next: @0,80
Next: @50,0	Next: @-20,0
Next: @0,20	Next: @0,-80
Next: @-80,0	Next: @-40,0
Next: close	Next: close

- 4 Restore UCS BASE and zoom centre about -35,70,60 at 250 mag in all viewports

- 5 Select the TABULATED SURFACE icon from the Surfaces toolbar and:

prompt Select object for path curve
respond **pick polyline 1** as fig(a)
prompt Select object for direction vector
respond **pick line 1 at the end indicated**
and a tabulated surface is added to the path curve



- 6 The added tabulated surface has a 'depth' equal to the length of the direction vector, i.e. 120

- 7 Menu bar with **Draw-Surfaces-Tabulated Surface** and:

prompt Select object for path curve
respond **pick polyline 2** as fig(a)
prompt Select object for direction vector
respond **pick line 2 at the end indicated**

- 8 Figure 17.1 displays (in 3D) the results of the tabulated surface operations:

- reference information
- tabulated surfaces without hide at SE Isometric viewpoint
- tabulated surfaces with hide at SE Isometric viewpoint
- at a NW Isometric viewpoint with hide

- 9 *Task*

- Erase the tabulated surfaces to display the original path curves
- Repeat the tabulated surface commands, but pick the direction vector lines at the 'opposite ends' from the exercise. The path curve will be 'extruded' in the opposite sense.

Summary

- A tabulated surface is a parallel polygon mesh
- The command requires:
 - a path curve – a single object
 - a direction vector – generally a line
- The command can be used in 2D or 3D
- The final surface orientation is dependent on the direction vector 'pick point'
- SURFTAB1 determines the surface appearance with curved objects
- The command can be activated:
 - in icon form from the Surfaces toolbar
 - from the menu bar with Draw-Surfaces
 - by entering **TABSURF <R>** at the command line.

Revolved surface

A revolved surface is a polygon mesh generated by rotating a path curve (profile) about an axis, the user selecting:

- a) **the path curve** – a single object, e.g. a line, arc, circle or 2D/3D polyline
- b) **the axis of revolution** – generally a line, but can be an open or closed polyline.

The generated mesh is controlled by two system variables:

- a) SURFTAB1: controls the mesh in the direction of the revolution
- b) SURFTAB2: defines any curved elements in the profile
- c) the default value for both variables is 6

Example 1

- 1 Open the MV3DSTD template file, MVLAY1 tab and layer MODEL current, UCS BASE and refer to Fig. 18.1
- 2 Make the lower right viewport active and display toolbars
- 3 Draw two lines:
 - a) start point: 0,0 next point: @100,0
 - b) start point: 0,0 next point: @0,100
- 4 Set SURFTAB1 to 16 and SURFTAB2 to 6 – command line entry

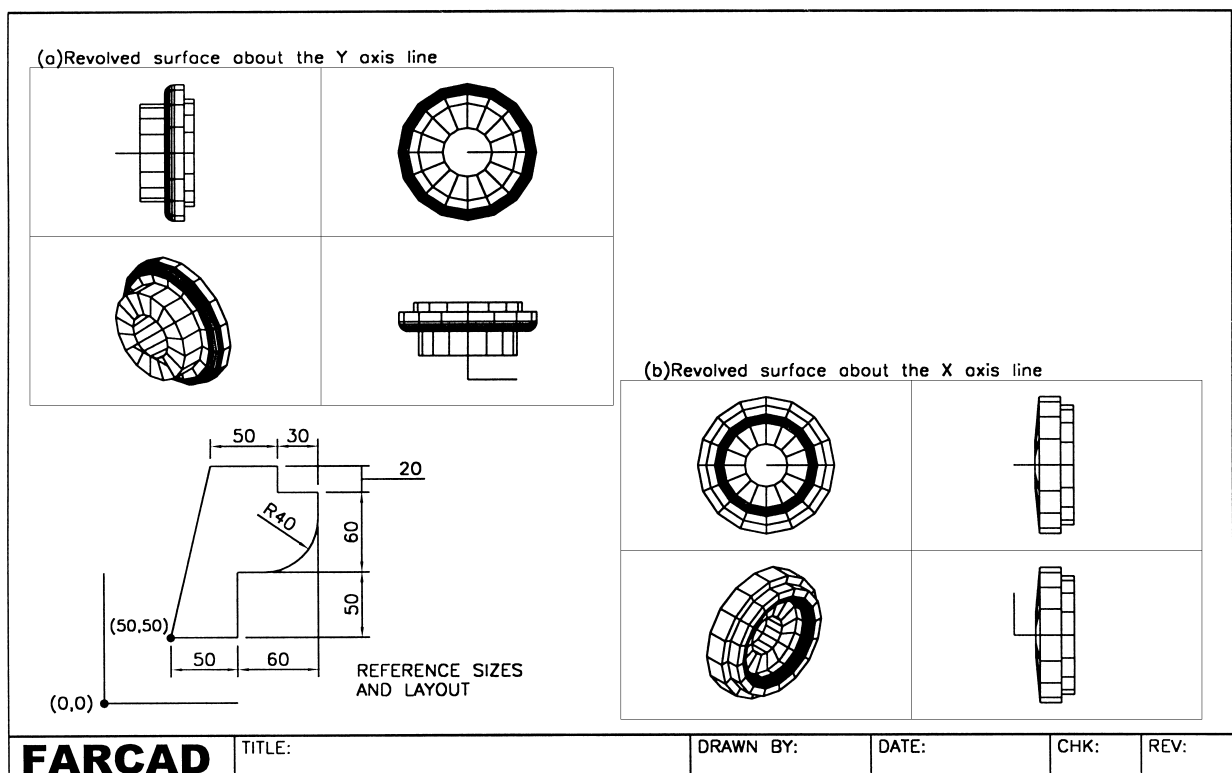


Figure 18.1 Revolved surface example 1.

- 5 Using the polyline icon from the Draw toolbar, create a **CLOSED** polyline shape using the reference sizes given in Fig. 18.1. The start point is to be 50,50

Note

The actual polyline shape is not that important. Use your discretion/own design, but try and keep to the overall reference sizes given.

- 6 Select the REVOLVED SURFACE icon from the Surfaces toolbar and:

prompt Select object to revolve

respond **pick any point on the polyline**

prompt Select object that defines the axis of revolution

respond **pick the Y axis line**

prompt Specify start angle<0> and enter: **0 <R>**

prompt Specify included angle (+ = ccw, - = cw)<360>

enter **360 <R>**



- 7 A revolved surface model will be displayed in each viewport
- 8 In all viewport, zoom centre about 0,120,0 at 350 magnification
- 9 Hide each viewport – fig(a)
- 10 Erase the revolved surface (regen needed?) to display the original polyline shape and from the menu bar select **Draw-Surfaces-Revolved Surface** and:
- a) object to revolve: pick the polyline shape
- b) object to define axis of revolution: pick the X axis line
- c) start angle: 0
- d) included angle: 360
- 11 Zoom centre about 100,0,0 at 400 magnification
- 12 Hide the viewports – fig(b)
- 13 Save if required, as this first exercise is complete.

Example 2

- 1 Open the MV3DSTD template file, MVLAY1 tab, layer MODEL, UCS BASE with the lower right viewport active
- 2 Refer to Fig. 18.2
- 3 Draw a line from 0,0 to @0,250
- 4 With the polyline icon, draw an **OPEN** polyline shape using the sizes in fig(a) as a reference. The start point is to be (0,50) but the final polyline shape is at your discretion as it is your wine glass design.
- 5 Set SURFTAB1 to 18 and SURFTAB2 to 6
- 6 At the command line enter **REVSURF <R>** and:
- a) object to revolve: pick the polyline shape
- b) object to define axis of revolution: pick the line
- c) start angle: enter 0
- d) included angle: enter 270
- 7 Set the following 3D viewpoints in the named viewports:
 Top left: NE Isometric Top right: NW Isometric
 Lower left: SE Isometric Lower right: SW Isometric
- 8 Zoom centre about 0,120,0 at 200 magnification
- 9 Hide the viewports – fig(b)
- 10 Save if required

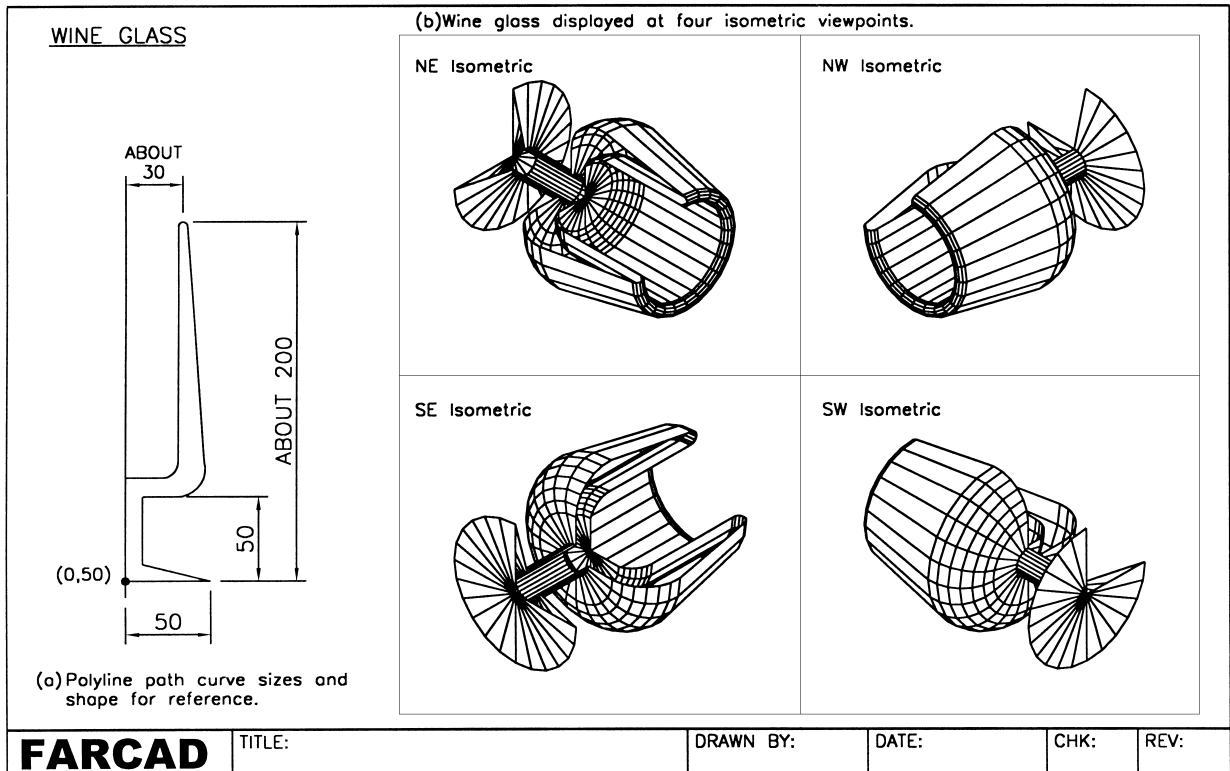


Figure 18.2 Revolved surface example 2.

Summary

- 1 The revolved surface command can be used to produce very complex surface models from relatively simple profiles
- 2 The resultant polygon mesh is controlled by the two system variables, SURFTAB1 and SURFTAB2:
 - a) SURFTAB1: controls the mesh in the direction of rotation
 - b) SURFTAB2: controls the display of curved elements in the profile
- 3 The start angle can vary between 0 and 360. A start angle of 0 means that the surface is to begin on the current drawing plane. This is generally what is required.
- 4 The included angle allows the user to define the angle the path curve is to be revolved through. The 360 default value gives a complete revolution, but 'cut-away' models can be obtained with angles less than 360
- 5 The direction of the revolved surface is controlled by the sign of the included angle and:
 - a) +ve for anti-clockwise revolved surfaces
 - b) -ve for a clockwise revolution
- 6 The command can be activated by icon, from the menu bar or by command line entry.

Assignment

MACFARAMUS designed a garden furniture arrangement for the gardens in CADOPOLIS. This garden furniture set complemented the ornamental flower bed created as a ruled surface. You need to create two profiles and revolve them about two different axes. Adding colour to the revolved surfaces greatly enhances the model appearance with shading and rendering.

Activity 12: Garden furniture set of MACFARAMUS.

- 1 Use your MV3DSTD template file – MVLAY1 tab
- 2 Make the top right viewport active and restore UCS FRONT
- 3 Draw two polyline profiles using the reference data given. Use your discretion for sizes not given, or design your own table and chair. Also draw two vertical lines for the axes of revolution.
- 4 Set SURFTAB1 to 18 and SURFTAB2 to 6
- 5 Revolve the profiles about vertical lines
- 6 Change the colour of the revolved chair to green and the table to blue
- 7 Restore UCS BASE and make the lower left viewport active
- 8 Polar array the chair for five items about the point (0,0) with rotation
- 9 Hide the viewports, then REGENALL and save as MODR2002\GARDEN
- 10 Try the following with the Model tab active:
 - a) Gouraud shade
 - b) Use the 3D orbit command
- 11 Make sure this model has been saved.

Edge surface

An edge surface is a 3D polygon mesh stretched between four **touching** edges. The edges can be combinations of lines, arcs, polylines or splines but **must form a closed loop**.

The edge surface mesh is controlled by the system variables:

- a) SURFTAB1: the M facets in the direction of the first edge selected
- b) SURFTAB2: the N facets in the direction of the edges adjacent to the first selected edge

Three examples will be used to demonstrate the command, the first being in 2D, the second to allow us to use the editing features of a polygon mesh, and the third will use splines as the four touching edges.

Example 1 (2D edge surfaces)

- 1 Open any 2D drawing and make two layers, EDGE colour red and MESH colour blue
- 2 Refer to Fig. 19.1 and display toolbars as required. Erase all objects

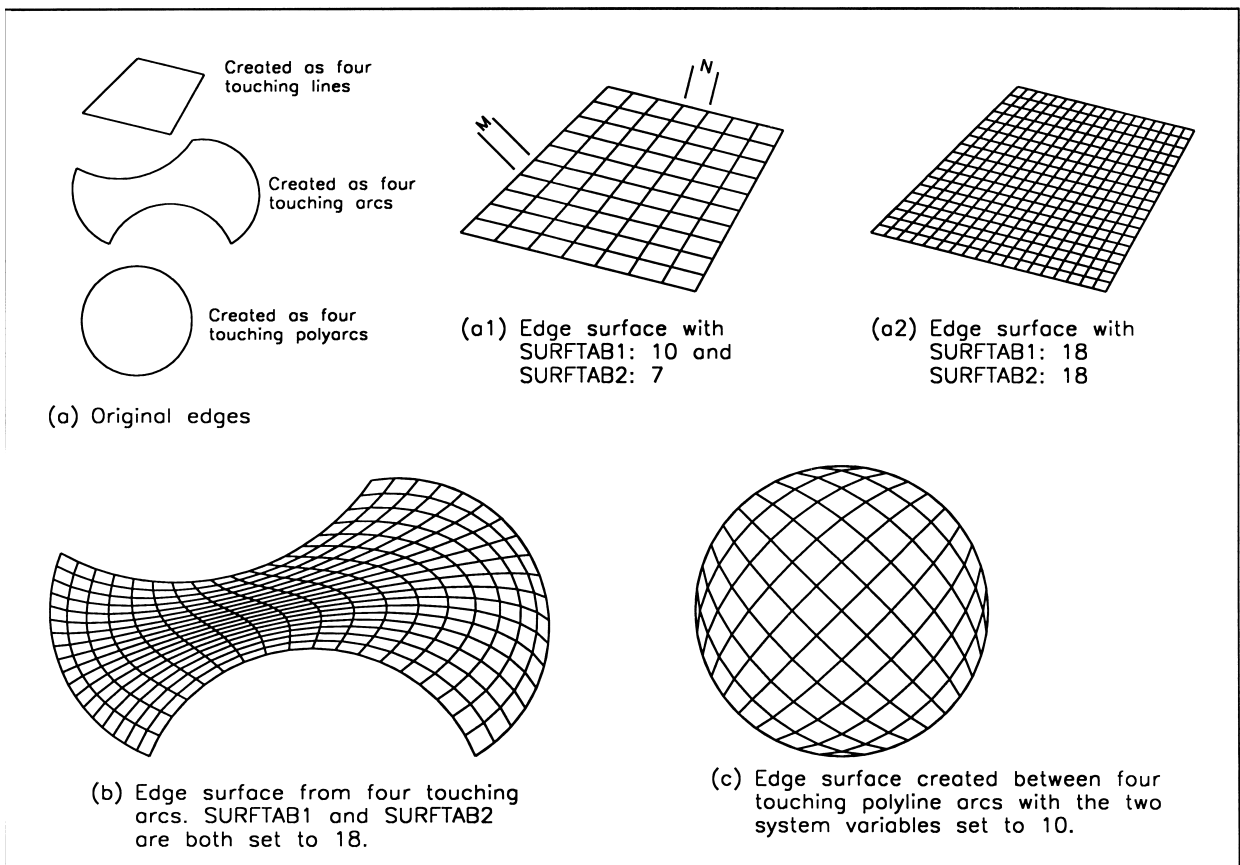



Figure 19.1 Edge surface example 1 – 2D application.

- 3 With layer EDGE current, create the following touching edges similar in layout to fig(a):
 - a) four lines
 - b) four three point arcs
 - c) four single 90deg polyline arcs (use CE option)
- 4 Set SURFTAB1 to 10 and SURFTAB2 to 7
- 5 Select the EDGE SURFACE icon from the Surfaces toolbar and:



```

prompt Select object 1 for surface edge
respond pick a point on any line
prompt Select object 2 for surface edge
respond pick a point on another line
prompt Select object 3 for surface edge
respond pick a point on a third line
prompt Select object 4 for surface edge
respond pick a point on the fourth line
      
```
- 6 A 10x7 surface mesh is stretched between the four touching lines as fig(a1)
- 7
 - a) Erase the added edge surface
 - b) Set SURFTAB1 and SURFTAB2 to 18
 - c) Menu bar with **Draw-Surfaces-Edge Surface** and pick the four touching lines in any order
 - d) The edge surface mesh is displayed as fig(a2)
- 8 At the command line enter **EDGESURF <R>** and pick the four arcs to display the edge surface mesh as fig(b)
- 9 Set both SURFTAB1 and SURFTAB2 to 10 and add an edge surface mesh between to four touching polyarcs – fig(c). The result of this mesh is quite interesting?

This completes the first exercise and it need not be saved.

Example 2 (a 3D edge surface mesh)

- 1 Open your MV3DSTD template file (MVLAY1 tab)
- 2 Refer to Fig. 19.2
- 3 With layer MODEL, UCS BASE and the lower left viewport active, use the LINE icon to draw the four touching lines:


```

Start point: 0,0,0
Next point: 150,0,-20
Next point: 180,200,30
Next point: 40,120,50
Next point: close
      
```
- 4 The four lines will be displayed as fig(a)
- 5 Centre each viewport about the point 90,100,25 at 250 mag.
- 6 Make a new layer, MESH colour blue and current
- 7 Set both SURFTAB1 and SURFTAB2 to 10
- 8 Using the edge surface icon, pick the four lines in the order indicated 1–2–3–4 as fig(a)
- 9 An edge surface mesh will be stretched between the four lines as fig(b)
- 10 In paper space, zoom the 3D viewport and return to model space

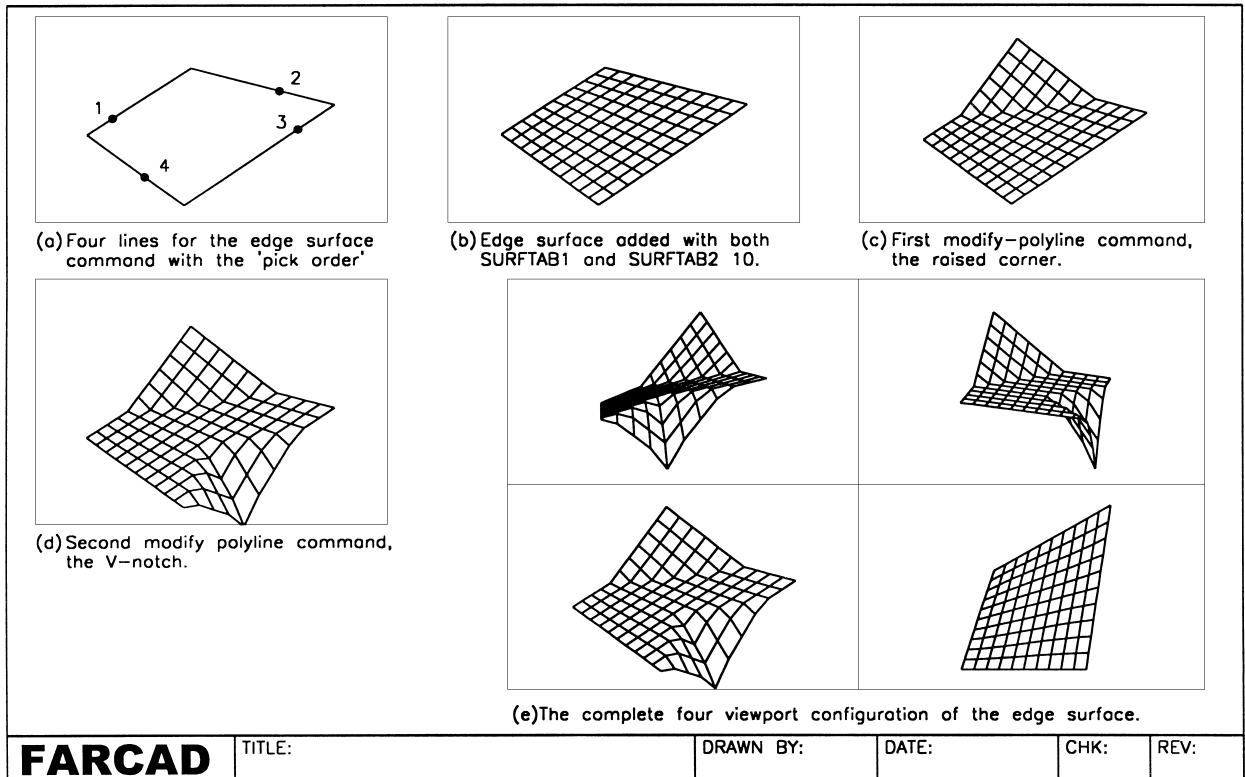


Figure 19.2 Edge surface example 2 – 3D with edit vertices.

11 Menu bar with **Modify-Object-Polyline** and:

prompt Select polyline
respond **pick any point on mesh**
prompt Enter an option [Edit vertex/Smooth surface/..
enter **E <R>** – the edit vertex option
prompt Current vertex (0,0)
then Enter an option [Next/Previous/..
and an X is displayed at the 0,0 vertex – leftmost?
enter **U <R>** until X is at vertex (10,0) and:
prompt Current vertex (10,0)
then Enter an option [Next/Previous/..
enter **M <R>** – the move option
prompt Specify new location for marked vertex
enter **@0,0,60 <R>**
and **DO NOT EXIT COMMAND**

12 We now want to alter other vertices of the mesh to create a raised effect. This will be achieved by:

- a) moving the X to the required vertices
- b) using the M option
- c) entering the required relative vertex coordinates

- 13 Use the N/D/L/R/U options and enter the following new locations for the named vertices:

<i>relative movement</i>	<i>vertices</i>				
@0,0,50	9,0	10,1			
@0,0,40	8,0	9,1	10,2		
@0,0,30	7,0	8,1	9,2	10,3	
@0,0,20	6,0	7,1	8,2	9,3	10,4
@0,0,10	5,0	6,1	7,2	8,3	9,4 10,5

- 14 When all the new vertex locations have been entered:

- enter X <R> to exit the edit vertex option
- then enter X <R> to end the command

- 15 The mesh will be displayed with a 'raised corner' as fig(c)

- 16 At the command line enter **PEDIT <R>** then:

- pick any point on the mesh
- enter E <R> for the edit vertex option
- use the N/U/R/L/D entries to move the X to the following named vertices and with the M (move option), enter the following new locations:

<i>relative movement</i>	<i>vertices</i>					
@0,0,-80	4,10					
@0,0,-50	3,10	4,9	5,10			
@0,0,-30	2,10	3,9	4,8	5,9	6,10	
@0,0,-10	1,10	2,9	3,8	4,7	5,8	6,9 7,10

- exit the vertex option with X <R>

- exit the polyline edit command with X <R>

- 17 These vertex modifications have produced a v-type notch in the mesh as fig(d)

- 18 Paper space and zoom-previous then model space. Freeze the MODEL layer

- 19 The complete four viewport configuration of the edge surface mesh is displayed in fig(e)

- 20 The exercise is now complete – save if required.

- 21 Now investigate the effect of the smooth surface option (S) on the mesh

Example 3 (an edge surface mesh created from splines)

This example will demonstrate how an edge surface can be stretched between four spline curves to simulate a car body panel.

- Open your MV3DSTD template file as usual, i.e. MVLAY1 tab, layer MODEL and UCS BASE
- Zoom centre about the point (-100,50,50) at 225 magnification in the 3D viewport and 200 in the other three
- With the lower left viewport active, refer to Fig. 19.3 and by selecting the SPLINE icon from the Draw toolbar, draw four spline curves using the following coordinate information. Use the start and end coordinates as the start and end tangents, indicated by (S) and (E) in the following data table:

spline 1	spline 2	spline 3	spline 4
0,0,0 (S)	-200,0,0 (S)	-200,80,105 (S)	0,0,0 (S)
-200,0,0 (E)	-200,0,100	0,100,105 (E)	0,0,110
<RETURN>	-200,80,105 (E)	<RETURN>	0,100,105 (E)
	<RETURN>		<RETURN>

- The four spline curves will be displayed as fig(a)

- Make a new layer, MESH colour blue and current

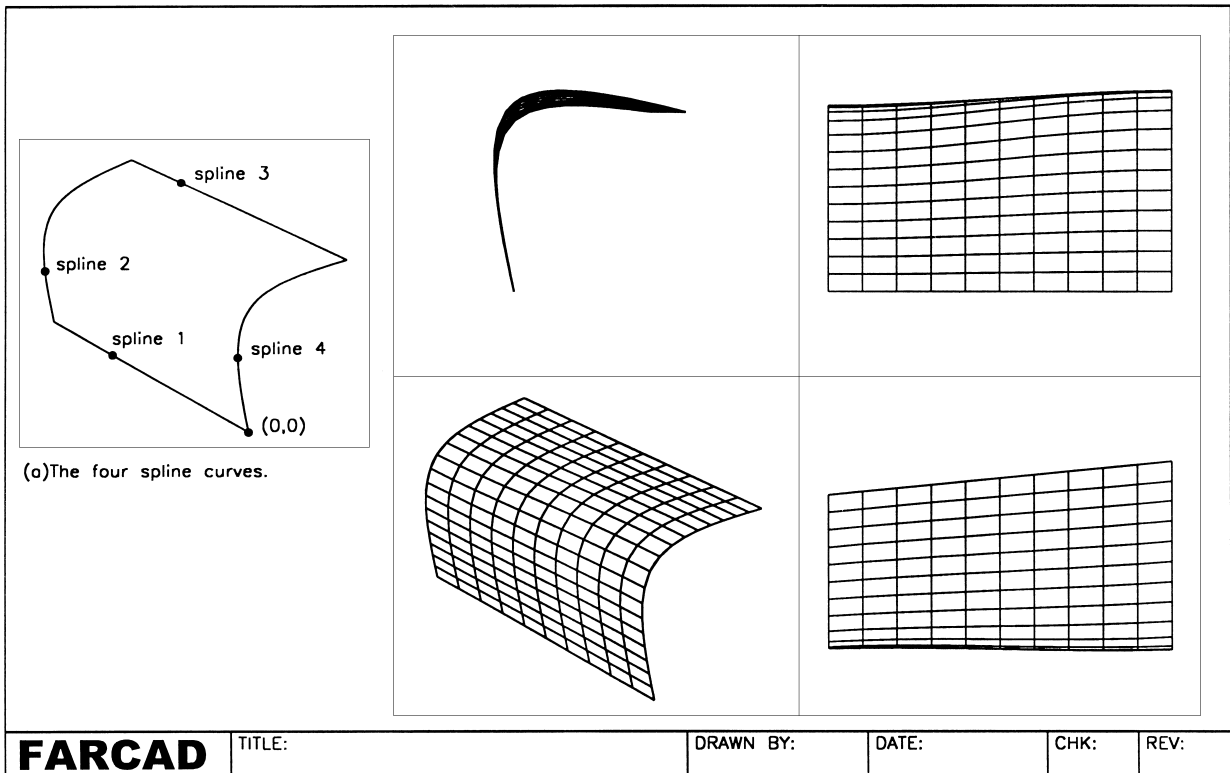


Figure 19.3 Edge surface example 3 – using splines.

- 6 Set both SURFTAB1 and SURFTAB2 to 18
- 7 Using the Edge Surface icon, pick the four spline curves
- 8 *Task*
 - a) Use the View-Hide sequence in the four viewports, and you will probably not notice any difference
 - b) Gouraud shade the four viewports and the effect is impressive
 - c) Use the 3D orbit command in the 3D viewport, and real-time rotate the shaded model. This is really impressive.
- 9 Save if required, as this completes the edge surface exercises.

Summary

- 1 An edge surface is a polygon mesh stretched between four **touching** objects – lines, arcs, splines or polylines
- 2 An edge surface mesh can be edited with the polyline edit command
- 3 The added surface is a **COONS** patch and is **bicubic**, i.e. one curve is defined in the mesh M direction and the other is defined in the mesh N direction
- 4 The first curve (edge) selected determines the mesh M direction and the adjoining curves define the mesh N direction
- 5 The mesh density is controlled by the system variables:
 - a) SURFTAB1: in the mesh M direction
 - b) SURFTAB2: in the mesh N direction

- 6 The default value for SURFTAB1 and SURFTAB2 is 6
- 7 The type of mesh stretched between the four curves is controlled by the **SURFTYPE** system variable and:
 - a) SURFTYPE 5 – Quadratic B-spline
 - b) SURFTYPE 6 – Cubic B-spline (default)
 - c) SURFTYPE 8 – Bezier curve
- 8 The SURFTYPE variable controls the appearance of all mesh curves.

Assignment

MACFARAMUS was contracted by the emperor TOOTENCADUM to create a flat topped hill for a future project.

The activity is very similar to the second example, i.e. an edge surface has to have several of its vertices modified to give a 'flat-top hill' effect. The process is quite tedious, but persevere with it as it is needed for another activity in a later chapter.

Activity 13: The flat topped hill made by MACFARAMUS.

- 1 Use your MV3DSTD template file with UCS BASE as usual
- 2 Zoom centre about 0,0,50 at 400 magnification originally
- 3 With layer MODEL, create four touching polyline arcs of radius 200 with 0,0 as the arc centre point. If you are unsure of this, use the Centre, Start, End ARC option.
- 4 Make a new layer called HILL, colour green and current
- 5 Set SURFTAB1 and SURFTAB2 to 20
- 6 Add an edge surface to the four touching arcs
- 7 In paper space, zoom the 3D viewport then return to model space
- 8 Use the Modify-Object-Polyline (PEDIT) command with the Edit Vertex option to move the following vertices by **@0,0,100**:

a)			6,9	6,10	6,11			
b)	7,7	7,8	7,9	7,10	7,11	7,12	7,13	
c)	8,7	8,8	8,9	8,10	8,11	8,12	8,13	
d)	9,6	9,7	9,8	9,9	9,10	9,11	9,12	9,13
e)	10,6	10,7	10,8	10,9	10,10	10,11	10,12	10,13
f)	11,6	11,7	11,8	11,9	11,10	11,11	11,12	11,13
g)	12,7	12,8	12,9	12,10	12,11	12,12	12,13	
h)	13,7	13,8	13,9	13,10	13,11	13,12	13,13	
i)			14,9	14,10	14,11			
- 9 *Note*
 - a) The named vertices all lie within a circle of radius 100
 - b) Use the N/U/D/L/R entries of the edit vertex option until the named vertex is displayed then use the M option with an entry of @0,0,100
- 10 When all the vertices have been modified, optimise your viewpoints. I used four different VPOINT-ROTATE values and the effect with hide was quite 'pleasing'
- 11 Save this drawing as **MODR2002\HILL** as it will be used with a later activity.
- 12 The View-Shade-Gouraud Shaded effect in any 3D viewport with the 3D orbit command is very interesting. You can rotate the model to 'see inside' the raised part of the edge surface.

3D polyline

A 3D polyline is a continuous object created in 3D space. It is similar to a 2D polyline, but does not possess the 2D versatility, i.e. there are no variable width or arc options available with a 3D polyline. The benefit of a 3D polyline is that it allows x,y,z coordinates to be used. A 2D polyline can only be created in the plane of the current UCS.

Example

This exercise will create a series of hill contours from splined 3D polylines, so:

- 1 Open your MV3DSTD template file, MVLAY1 tab with the lower left viewport active and UCS BASE.
- 2 Refer to Fig. 20.1 and make the following new layers:
LEVEL1, LEVEL2, LEVEL3, LEVEL4 – all colour green
PATH1: red; PATH2: blue; PATH3: magenta
- 3 With layer LEVEL1 current, menu bar with **Draw-3Dpolyline** and:
 - prompt* Specify start point of polyline and enter: **0,50,0 <R>**
 - prompt* Specify endpoint of line and enter: **50,0,0 <R>**
 - prompt* Specify endpoint of line and enter: **150,0,0 <R>**
 - prompt* Specify endpoint of line and enter: **220,60,0 <R>**
 - prompt* Specify endpoint of line and enter: **260,140,0 <R>**
 - prompt* Specify endpoint of line and enter: **170,180,0 <R>**
 - prompt* Specify endpoint of line and enter: **20,160,0 <R>**
 - prompt* Specify endpoint of line and enter: **c <R>**
- 4 Make layer LEVEL2 current and at the command line enter **3DPOLY <R>** and:
 - prompt* Specify start point of polyline and enter: **40,60,50 <R>**
 - prompt* Specify endpoint of line and enter: **80,20,50 <R>**
 - prompt* Specify endpoint of line and enter: **140,35,50 <R>**
 - prompt* Specify endpoint of line and enter: **200,100,50 <R>**
 - prompt* Specify endpoint of line and enter: **130,140,50 <R>**
 - prompt* Specify endpoint of line and enter: **60,130,50 <R>**
 - prompt* Specify endpoint of line and enter: **c <R>**
- 5 With layers LEVEL3 and LEVEL4 current, use the 3D polyline command with the following coordinate values:

	<i>Level3</i>	<i>Level4</i>
Start point	70,70,100	85,70,125
Endpoint	80,35,100	90,50,125
Endpoint	130,45,100	130,60,125
Endpoint	170,90,100	130,80,125
Endpoint	100,120,100	close
Endpoint	close	

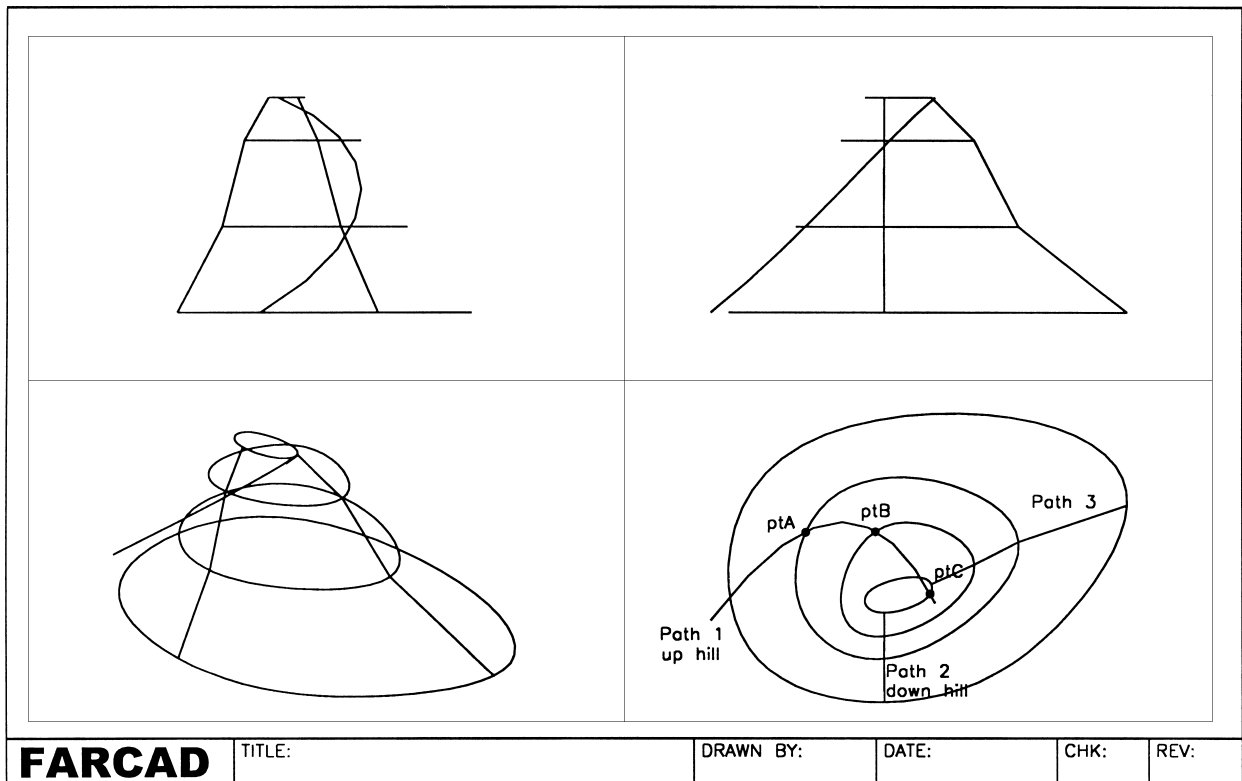


Figure 20.1 3D polyline example.

- 6 Menu bar with **Modify-Object-Polyline** and:
 - prompt* Select polyline
 - respond* **pick the 3D polyline created at level 1**
 - prompt* Enter an option
 - enter* **S <R>** – the spline option
 - prompt* Enter an option
 - enter* **X <R>** – the exit command option
- 7 The selected polyline will be displayed as a splined curve
- 8 Use the S option of the Edit Polyline command to spline the other three 3D polylines
- 9 Zoom centre about 120,90,60 at 200 magnification in all viewports
- 10 With layer PATH1 current, use the 3D polyline command with the following entries:
 - Start: 0,50,0 – level 1 point
 - Endpoint: 60,130,50 – level 2 point
 - Endpoint: 100,120,100 – level 3 point
 - Endpoint: 130,60,125 – level 4 point
 - Endpoint: <R>
- 11 This 3D polyline is a path 'up the hill' and each entered coordinate value is a point on the level 1,2,3,4 contours
- 12 Spline this polyline, and it does not pass through the entered coordinates

13 *Task*

- a) Using the ID command, identify the coordinates of the points A, B and C where PATH1 'crosses' the level 2, 3 and 4 contours. My values were:
ptA: 55.06, 101.57, 50
ptB: 95.42, 101.91, 100
ptC: 127.07, 65.65, 125
- b) With layer PATH2 current, create a 3D polyline as a 'path down the hill', this path to be drawn as a vertical line in the top (lower right) viewport. It has to 'touch' each contour, and if possible, be the shortest distance (ortho on helps, as does OSNAP nearest). Do not spline this path. Find the distance from top to bottom for this path. My value was 135.51.
- c) With layer PATH3 current, create another un-splined 3D polyline which is to 'join the four level right-ends' in the top right viewport. Find the distance between each level for this path.
My three values were:
1-2: 82.81
2-3: 57.69
3-4: 38.26

14 The exercise is now complete. Save if required.

Summary


- 1 A 3D polyline is a single object and can be used with x,y,z coordinate entry
- 2 A 3D polyline can be edited with options of Edit vertex, Spline and decurve
- 3 3D polylines cannot be displayed with varying width or with arc segments.
- 4 The command is activated from the menu bar or by command line entry.

3D objects

AutoCAD has nine pre-defined 3D objects, these being box, pyramid, wedge, dome, sphere, cone (and cylinder), torus, dish and mesh. They are considered as 'meshes' and can be displayed with hide, shade and render.

To demonstrate some of these objects:

- 1 Open your MV3DSTD template file, with MVLAY1 tab, layer MODEL, UCS BASE and the lower left viewport active.
- 2 Refer to Fig. 21.1 and display the Surfaces toolbar
- 3 In the steps which follow, reason out the coordinate entry values
- 4 Select the BOX icon from the Surfaces toolbar and:



 - prompt* Specify corner point of box and enter: **0,0,0 <R>**
 - prompt* Specify length of box and enter: **150 <R>**
 - prompt* Specify width of box and enter: **120 <R>**
 - prompt* Specify height of box and enter: **100 <R>**
 - prompt* Specify rotation angle of box about Z axis and enter: **0 <R>**
- 5 A red box will be displayed at the 0,0,0 origin point
- 6 Menu bar with **Draw-Surfaces-3D Surfaces** and:
 - prompt* 3D Objects dialogue box
 - respond* **pick Wedge then OK**
 - prompt* Specify corner point of wedge and enter: **150,0,0 <R>**
 - prompt* Specify length of wedge and enter: **80 <R>**
 - prompt* Specify width of wedge and enter: **70 <R>**
 - prompt* Specify height of wedge and enter: **150 <R>**
 - prompt* Specify rotation angle of wedge about Z axis and enter: **-10 <R>**
- 7 At the command line enter **CHANGE <R>** and:
 - prompt* Select objects and pick the wedge then right-click
 - prompt* Specify change point or [Properties] and enter: **P <R>**
 - prompt* Enter property to change and enter: **C <R>** – colour option
 - prompt* Enter new colour and enter: **14 <R>**
 - prompt* Enter property to change and **<RETURN>**
- 8 Using the icons from the Surfaces toolbar, or the 3D Objects dialogue box, create the following two 3D objects:

<i>Cone</i>	<i>Cylinder</i> (using cone object)
Base centre: 50,70,100	Base centre: 75,0,50
Radius for base: 50	Radius for base: 50
Radius for top: 0	Radius for top: 50
Height: 85	Height: 90
Number of segments: 16	Number of segments: 16
Colour: green	Colour: blue
- 9 Restore UCS RIGHT and with the ROTATE icon from the Modify toolbar:
 - a) pick the blue cylinder then right-click
 - b) base point: 0,50
 - c) rotation angle: 90

- 10 Restore UCS BASE
- 11 Create the following two 3D objects:
- | | |
|---------------------------|----------------------------|
| <i>Dish</i> | <i>Torus</i> |
| Centre of dish: 75,60,0 | Centre of torus: 75,-90,50 |
| Radius: 60 | Radius of torus: 100 |
| Longitudinal segments: 16 | Radius of tube: 20 |
| Latitudinal segments: 16 | Tube segments: 16 |
| Colour: magenta | Torus segments: 16 |
| | Colour: cyan |
- 12 With the ROTATE icon:
- pick the cyan torus then right-click
 - base point: -90,50
 - angle: 90
- 13 Restore UCS BASE
- 14 Zoom centre about 75,0,50 at 300 magnification
- 15 Hide and shade to display the model with 'bright' colours
- 16 *Task*
- Still with the MVLAY1 tab active:
- enter paper space and make layer VP current
 - menu bar with View-Viewports-1 Viewport and make a new viewport selecting one of the points 'outwith' the original four as shown in Fig. 21.1
 - display the model in this new viewport from below and zoom centre about the same point as before
 - Question:* why did we pick one of the new viewport points outwith the original viewports?
 - Gouraud shade the model in each viewport, then convert back to 2D wireframe.
- 17 Save the layout, although we will not use it again

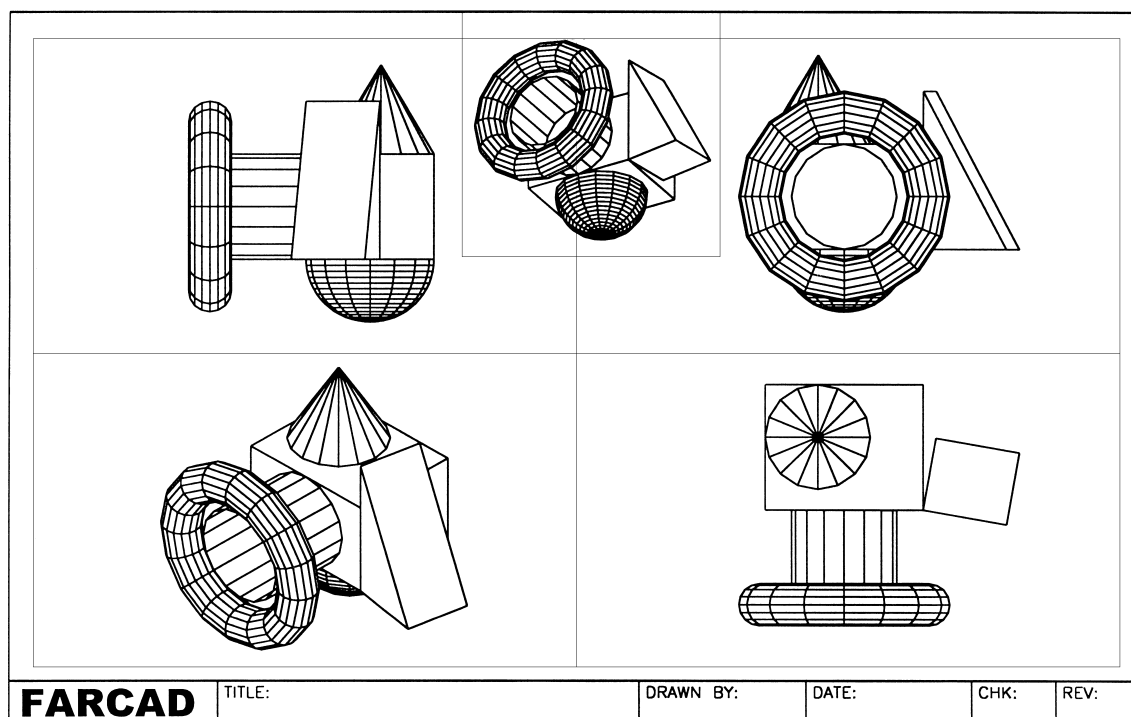


Figure 21.1 3D objects example.

Summary

- 1 The nine 3D objects are displayed as either faced or meshed surface models
- 2 The hide and shade (and render) commands can be used with 3D objects
- 3 3D objects are created from a reference point (corner/centre etc) and certain model geometry, e.g. length, width and height
- 4 A cylinder is created from a cone with the base and top radii having the same value
- 5 Activating 3D objects is available from the Surfaces toolbar or from the 3D Objects dialogue box. The various objects can also be created by command line entry, although this is **NOT** recommended.
- 6 *Note*
 In the 3D objects exercise, I used the command line CHANGE to alter the colour of the added objects. Changing object colour is an essential user requirement, especially when dealing with solid models. With AutoCAD 2002, there are two ways in which an objects colour can be altered, these being governed by the PICKFIRST system variable as follows:
 - a) PICKFIRST set to 0: CHANGE at the command line then pick the objects to be modified
 - b) PICKFIRST set to 1: select the objects to be modified then pick the Properties icon
 - c) the user must decide which method is to be used. Either method is perfectly valid
 - d) I will generally refer to this type of operation as:
 1. change the colour of the box to blue
 2. make the added cylinder green

Assignment

MACFARAMUS was commissioned by the emperor TOOTENCADUM to design and build a palace for queen NEFERSAYDY in the city of CADOPOLIS. With his knowledge of CAD, MACFARAMUS decided to build the palace from 3D objects, and this is your assignment. You have to use your imagination and initiative when designing the palace from 3D objects. Remember that the polar array command is very useful.

Activity 14: Palace of queen NEFERSAYDY by MACFARAMUS.

- 1 Use your MV3DSTD template file with UCS BASE and any layout (or the model) tab
- 2 The 3D objects have to be positioned in a circle with an 85 **maximum** radius. This circle has to be:
 - a) created from four touching polyarcs – as a previous exercise
 - b) edge surfaced with both SURFTAB1 and SURFTAB2 set to 16. The edge surface mesh should be on its own layer with a colour number of 42
- 3 The 3D objects have to be created on layer MODEL and can be to your own specification and layout. Some of my 3D objects were:

<i>box</i>	<i>wedge</i>	<i>dome</i>
corner: -40,-40,0	corner: 40,-40,0	centre: 0,0,60
length: 80	length: 20	radius: 25
width: 80	width: 10	colour: magenta
height: 60	height: 60	
colour: red	colour: blue	
<i>cylinder</i>	<i>cone</i>	
centre: 50,-50,0	centre: 50,-50,70	
radius: 8	radius: 12	
height: 70	height: 20	
colour: green	colour: green	
- 4 When the palace layout is complete, hide and shade
- 5 Save the complete model as **MODR2002\PALACE**. It will be used in a later activity.

3D geometry commands

All AutoCAD commands can be used in 3D but there are three commands which are specific to 3D models, these being 3D Array, Mirror 3D and Rotate 3D. In this chapter we will investigate these three commands and how to use the Align, Extend and Trim commands with 3D models.

Getting started

To investigate the 3D commands, we will create a new model from 3D Objects, so:

- 1 Begin a new drawing with your MV3DSTD template file
- 2 For this exercise it is recommended that the user works with the Model tab active so:
 - a) select the Model tab
 - b) layer MODEL and UCS BASE current
 - c) pan so that the UCS BASE icon is at the lower centre of the screen and alter the viewpoint with the command line entry **VPOINT <R>** then **R <R>** (rotate option) and enter angles of 300 and 30
 - d) refer to Fig. 22.1 which displays the 3D viewport only
- 3 Select the BOX icon from the Surfaces toolbar and:
 - a) corner: 0,0,0
 - b) length: 100; width: 100; height: 40; Z rotation: 0
 - c) colour: red
- 4 Select the WEDGE icon from the Surfaces toolbar and:
 - a) corner: 0,0,40
 - b) length: 30; width: 30; height: 30; Z rotation: 0
 - c) colour: blue
- 5 Zoom to a scale factor of 1
- 6 The two 3D objects will be displayed as fig(a)

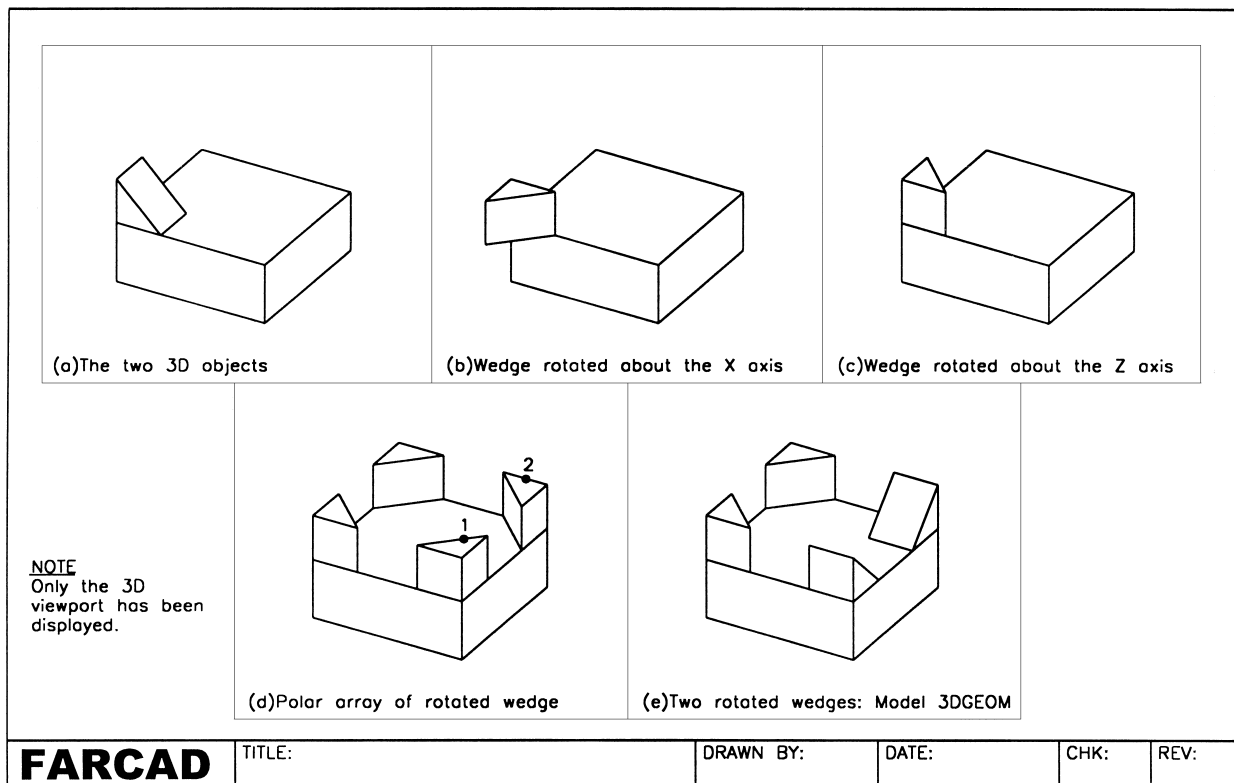


Figure 22.1 The 3DGEOM model for the 3D commands.

Rotate 3D

Using the menu bar sequence Modify-Rotate or selecting the ROTATE icon from the Modify toolbar results in a 2D command, i.e. the selected objects are rotated in the current XY plane. Objects can be rotated in 3D relative to the X, Y and Z axes with the Rotate 3D command. The command will be demonstrated by rotating the blue wedge and then the complete model.

Rotating the wedge

- From the menu bar select **Modify-3D Operation-Rotate 3D** and:
 - prompt* Select objects
 - respond* **pick the blue wedge then right-click**
 - prompt* Specify first point on axis or define axis by [Object/Last/..
 - enter* **X <R>** – the X axis option
 - prompt* Specify a point on the X axis<0,0,0>
 - enter* **0,0,40 <R>** – why these coordinates?
 - prompt* Specify rotation angle
 - enter* **90 <R>**
- The blue wedge is rotated about the x-axis as Fig. 22.1(b)

- 3 Activate the Rotate 3D command and:
 - prompt* Select objects
 - enter* **pick the blue wedge the right-click**
 - prompt* Rotate 3D options
 - enter* **Z <R>** – the Z axis option
 - prompt* Specify a point on Z axis<0,0,0>
 - enter* **0,0,0 <R>**
 - prompt* Specify rotation angle
 - enter* **90 <R>**
- 4 The blue wedge is now aligned as required – fig(c)
- 5 Select the ARRAY icon from the Modify toolbar and with the Array dialogue box select:
 - a) Type: Polar Array
 - b) Objects: the blue wedge
 - c) Center point: X: 50 and Y: 50
 - d) Method: Total number of items & Angle to fill
 - e) Total items: 4
 - f) Angle to fill: 360
 - g) Rotate items as copied: active
- 6 The blue wedge is arrayed to the four corners of the box as fig(d)
- 7 With the Rotate 3D command:
 - a) pick wedges 1 and 2 then right-click
 - b) enter Y <R> as the defined axis
 - c) enter 100,0,40 <R> as a point on axis – why this entry?
 - d) enter 90 <R> as the rotation angle
- 8 Now move these two rotated wedges from 0,0 by @-30,0
- 9 The final result is fig(e) and can be saved as **MODR2002\3DGEOM** for future recall.

Rotating the model

- 1 Model 3DGEOM on the screen in model tab with UCS BASE and layer MODEL current. Refer to Fig. 22.2 (3D only)
- 2 Menu bar with Modify-3D Operation-Rotate 3D and:
 - prompt* Select objects
 - respond* **window the complete model then right-click**
 - prompt* Specify first point on axis or define axis
 - enter* **X <R>** – the X axis option
 - prompt* Specify a point on the X axis
 - respond* **right-click, i.e.** accept the 0,0,0 default point
 - prompt* Specify rotation angle and enter: **45 <R>**
 - then* PAN if required then HIDE – fig(b)
- 3 At the command line enter **ROTATE3D <R>** and:
 - prompt* Select objects
 - respond* **window the model then right-click**
 - prompt* Specify first point on axis or define axis
 - enter* **Z <R>** – the Z axis option
 - prompt* Specify a point on Z axis<0,0,0> and: right-click
 - prompt* Specify rotation angle and enter: **60 <R>**
 - then* PAN to suit and HIDE – fig(c)

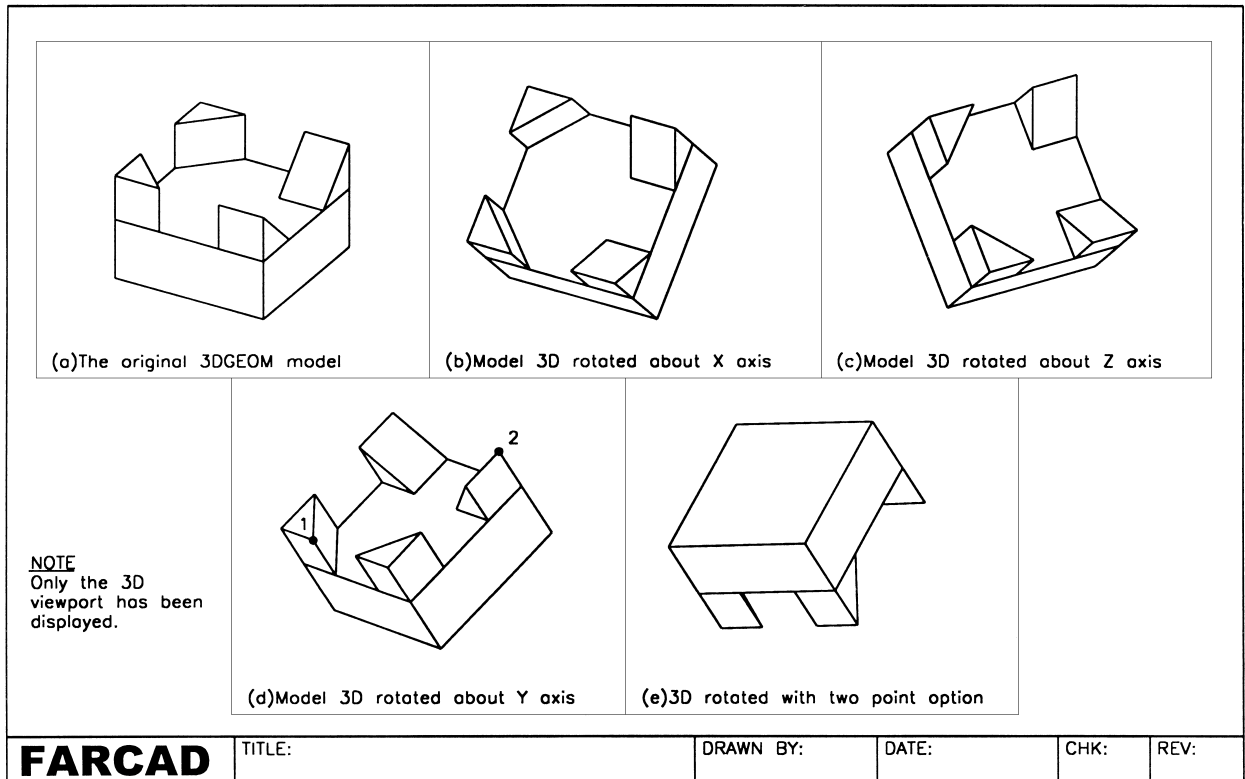


Figure 22.2 The ROTATE 3D command using 3DGEOM model.

- 4 Activate the Rotate 3D command and:
 - a) window the model
 - b) select the Y axis option
 - c) accept the default 0,0,0 point on Y axis
 - d) enter -60 as the rotation angle to give fig(d) with hide
- 5 Rotate 3D again, window the model and:

prompt Specify first point on axis
respond **Intersection icon and pick pt1**
prompt Specify second point on axis
respond **Intersection icon and pick pt2**
prompt Specify rotation angle and enter: **180 <R>**
then PAN if needed, then HIDE – fig(e)
- 6 Shade the model then:
 - a) investigate the 3D orbit effect in model space
 - b) investigate the layout tabs
- 7 Save layout if required, but we will not use it again.

Mirror 3D

This command allows objects to be mirrored about selected points or about any of the three X–Y–Z planes.

- 1 Open model 3DGEOM in model tab with UCS BASE and at the 300–30 angle viewpoint

- 2 Refer to Fig. 22.3 which again only displays the 3D viewport
- 3 Menu bar with **Modify-3D Operation-Mirror 3D** and:
- prompt* Select objects
respond **window the model then right-click**
prompt Specify first point on mirror plane (3 points) or [Object/Last/..
enter **XY <R>** – the XY plane option
prompt Specify point on XY plane<0,0,0> and: **right click**
prompt Delete source objects<Yes/No> and enter: **Y <R>**
then PAN to suit – fig(b)
- 4 The model at this stage has the AMBIGUITY effect of all 3D models, i.e. are you looking down or looking up? Hence HIDE!
- 5 At the command line enter **MIRROR3D <R>** and:
- prompt* Select objects
respond **window the model then right-click**
prompt Specify first point on mirror plane
respond **Intersection icon and pick pt1**
prompt Specify second point on mirror plane
respond **Intersection icon and pick pt2**
prompt Specify third point on mirror plane
respond **Intersection icon and pick pt3**
prompt Delete source object<Yes/No> and enter: **Y <R>**
then PAN to suit, then HIDE – fig(c)

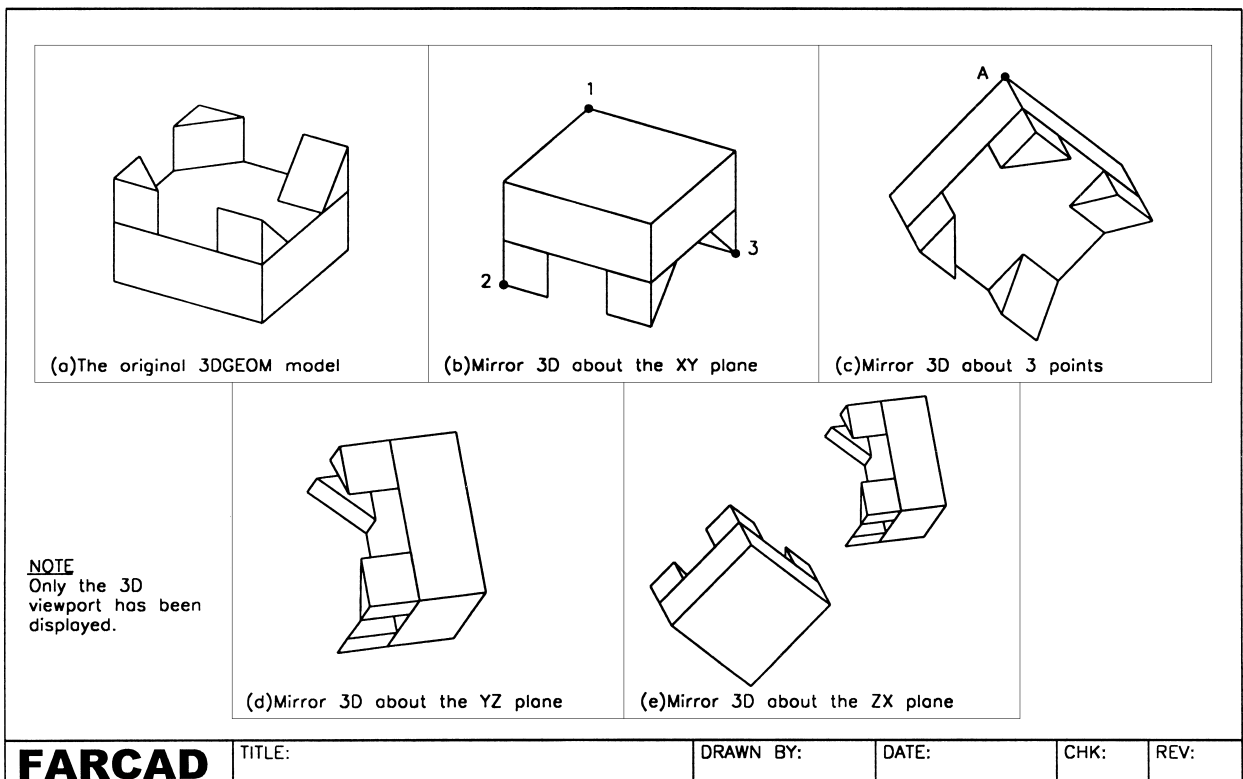


Figure 22.3 The MIRROR 3D command using 3DGEOM model.

- 6 Activate the Mirror 3D command and:
 - a) window the model the right-click
 - b) select the YZ plane option
 - c) pick intersection of point A as a point on the plane
 - d) enter Y to delete source objects prompt
 - e) pan and hide – fig(d)
- 7 Using the Mirror 3D command:
 - a) window the model the right-click
 - b) select the ZX option
 - c) enter –50,–50 as a point on the ZX plane
 - d) accept the N default delete source objects option
 - e) pan and hide – fig(e)
- 8 Shade and 3D orbit, then investigate the layout tabs
- 9 Save if required. We will not refer to this drawing again.

3D Array

The 3D array command is similar in operation to the 2D array. Both rectangular and polar arrays are possible, the rectangular array having rows and columns as well as levels in the Z direction. The result of a 3D polar array requires some thought!

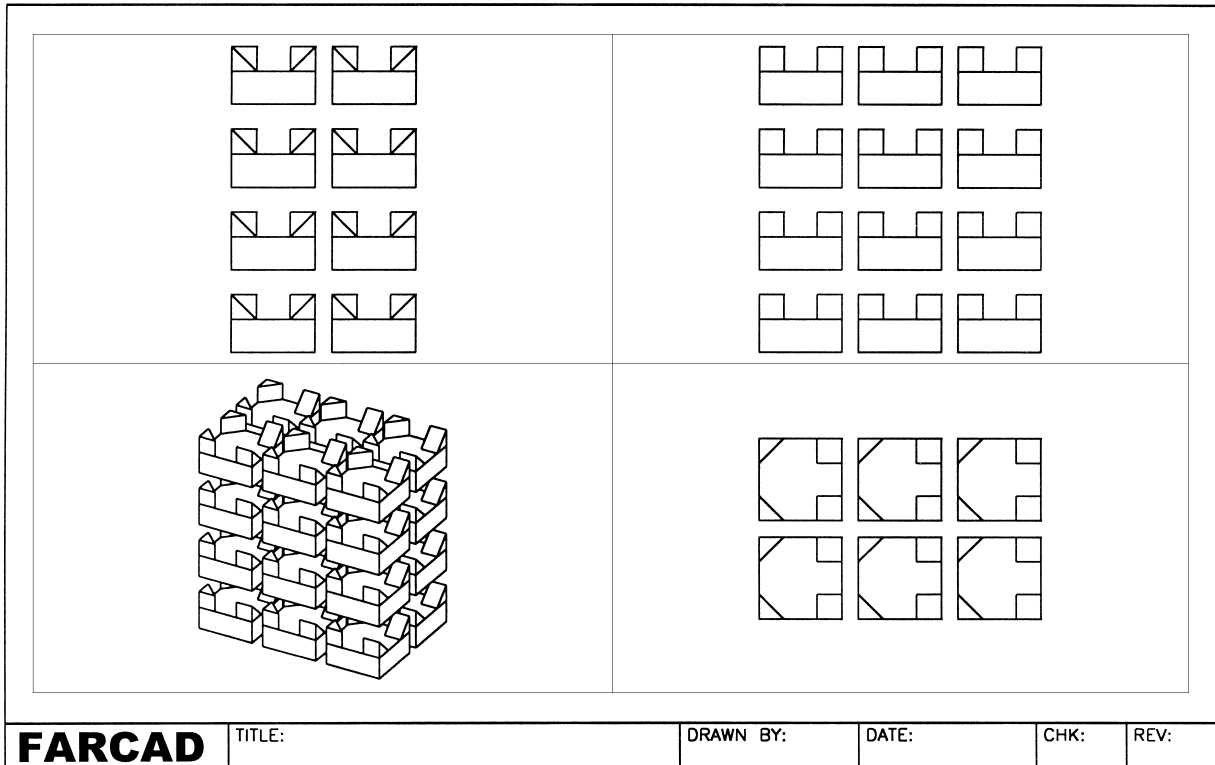
Rectangular

- 1 Open the 3DGEOM drawing, UCS BASE, Model tab active and refer to Fig. 22.4 which displays the MVLAY1 tab
- 2 At the command line enter **3DARRAY <R>** and:


```

prompt  Select objects
respond window the model then right-click
prompt  Enter the type of array [Rectangular/Polar]
enter    R <R> – rectangular option
prompt  Enter the number of rows(---)<1> and enter: 2 <R>
prompt  Enter the number of columns(|||)<1> and enter: 3 <R>
prompt  Enter the number of levels(...)<1> and enter: 4 <R>
prompt  Specify the distance between rows(---) and enter: 120 <R>
prompt  Specify the distance between columns(|||) and enter: 120 <R>
prompt  Specify the distance between levels(...) and enter: 100 <R>

```
- 3 The model will be displayed in a 2×3×4 rectangular matrix pattern but will 'be off the screen'.
- 4 Zoom-extents then hide to display the complete array
- 5 With MVLAY1 tab active, zoom centre about the point 170,110,185 (why these coordinates) at 400 magnification (550 in 3D view)
- 6 Hide the model – Fig. 22.4 then Gouraud shade the 3D viewport.
- 7 Try the 3D orbit command with the shaded model.
- 8 This exercise does not need to be saved.

**FARCAD**

TITLE:

DRAWN BY:

DATE:

CHK:

REV:

Figure 22.4 The 3D ARRAY (rectangular) command using 3DGEOM model.

Polar

- 1 Open 3DGEOM, UCS BASE in Model tab
- 2 Menu bar with **Modify-3D Operation-3D Array** and:
 - prompt* Select objects
 - respond* **window the model then right-click**
 - prompt* Enter the type of array and enter: **P <R>**
 - prompt* Enter the number of items and enter: **5 <R>**
 - prompt* Specify the angle to fill and enter: **360 <R>**
 - prompt* Rotate arrayed objects and enter: **Y <R>**
 - prompt* Specify center point of array
 - enter* **150,150,0 <R>**
 - prompt* Specify second point on axis if rotation
 - enter* **@0,0,100 <R>** i.e. a vertical line
- 3 Zoom-all then investigate the MVLAY1 tab – Fig. 22.5(a)
- 4 Undo the 3D polar array
- 5 Activate the 3D array command and:
 - a) window the original model then right-click
 - b) enter a polar array type
 - c) number of items: 5
 - d) angle to fill: 360
 - e) rotate as copied: Y
 - f) centre point of array and enter: **-50,0,0 <R>**
 - g) second point on axis and enter: **@0,100,0 <R>**
- 6 Zoom-all then hide and shade
- 7 Investigate the MVLAY1 tab which will need a zoom-extents and then a zoom scale factor – Fig. 22.5(b)

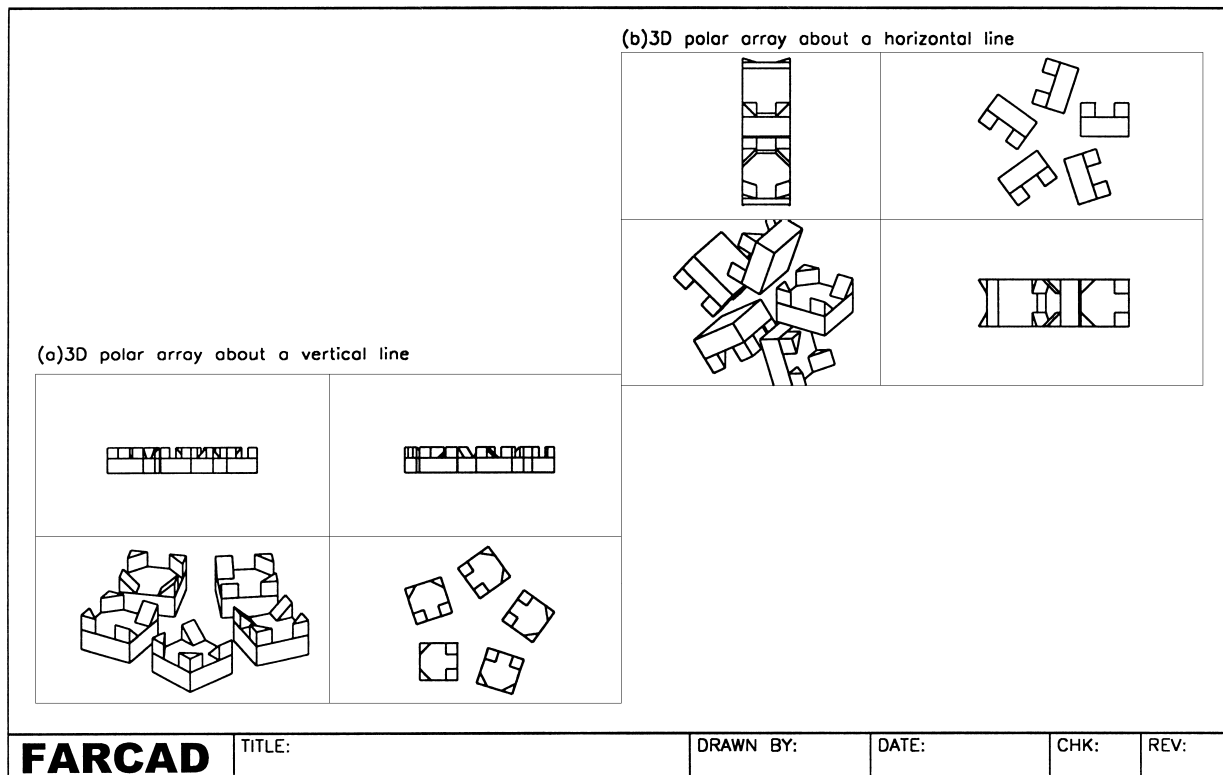


Figure 22.5 The 3D ARRAY (polar) using the 3DGEOM model.

- 8 *Note:*
 - a) The first polar array was about a vertical line, the second about a horizontal line. Think about the entered coordinates
 - b) The ARRAY icon/dialogue box is for 2D arrays only
- 9 This exercise is complete and need not be saved.

Align

The align commands can be used in 2D or 3D and allows objects (models) to be aligned with each other.

- 1 Open your MV3DSTD template file, MVLAY1 tab with UCS BASE and layer MODEL current. Refer to Fig. 22.6
- 2 Using the Surfaces toolbar create the following objects:

	<i>Box</i>	<i>Wedge</i>
corner	0,0,0	160,0,0
length	100	100
wedge	80	80
height	50	50
rotation	0	0
colour	red	blue
- 3 Copy the box and wedge:
 - a) base point: 0,0,0
 - b) second point: @150,150
- 4 Set the running object snap to Intersection

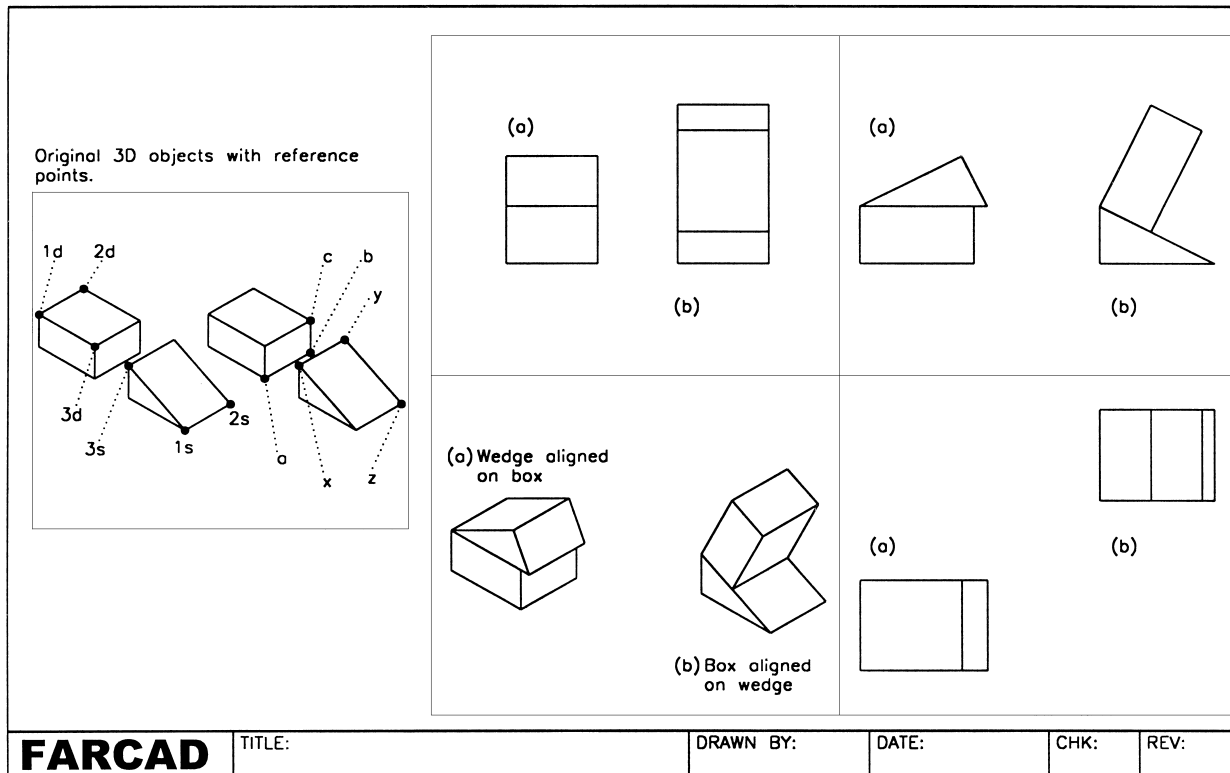


Figure 22.6 The ALIGN command with 3D objects.

- 5 Menu bar with **Modify-3D Operation-Align** and:
 - prompt* Select objects
 - respond* **pick a blue wedge then right-click**
 - prompt* Specify first source point and pick point 1s
 - prompt* Specify first destination point and pick point 1d
 - prompt* Specify second source point and pick point 2s
 - prompt* Specify second destination point and pick point 2d
 - prompt* Specify third source point and pick point 3s
 - prompt* Specify third destination point and pick point 3d
- 6 The blue wedge will be aligned with its sloped surface on the top of the box as (a)
- 7 At the command line enter **ALIGN <R>** and:
 - a) pick the copied red box then right-click
 - b) pick first source/destination points: pick a and x
 - c) pick second source/destination points: pick b and y
 - d) pick third source/destination points: pick c and z
 - e) the red box will be aligned onto the sloped surface of the wedge as (b)
- 8 This completes the align exercise.

3D extend and trim

Objects can be trimmed and extended in 3D irrespective of the objects' alignment. The two commands are very dependent on the UCS position.

- 1 Open your MV3DSTD template file and refer to Fig. 22.7
- 2 Using the LINE icon draw a square with:
start point: 0,0 next point: @100,0 next point: @0,100
next point: @-100,0 next point: close
- 3 Multiple copy the square with:
 a) base point: 0,0
 b) second point: @0,0,120 and: @0,0,180
- 4 Scale the top square about the point 50,50,180 by 0.5
- 5 Draw in the four vertical lines between the two large squares, then draw the following four lines:

line	start point	next point	colour
1	25,25,180	@0,-5,-30	blue
2	75,25,180	@5,0,-30	green
3	75,75,180	@0,5,-30	cyan
4	25,75,180	@0,0,-60	magenta
- 6 Change the viewpoint in the 3D viewport with VPOINT-ROTATE and angles of 300 and 30. The model will be displayed as fig(a).

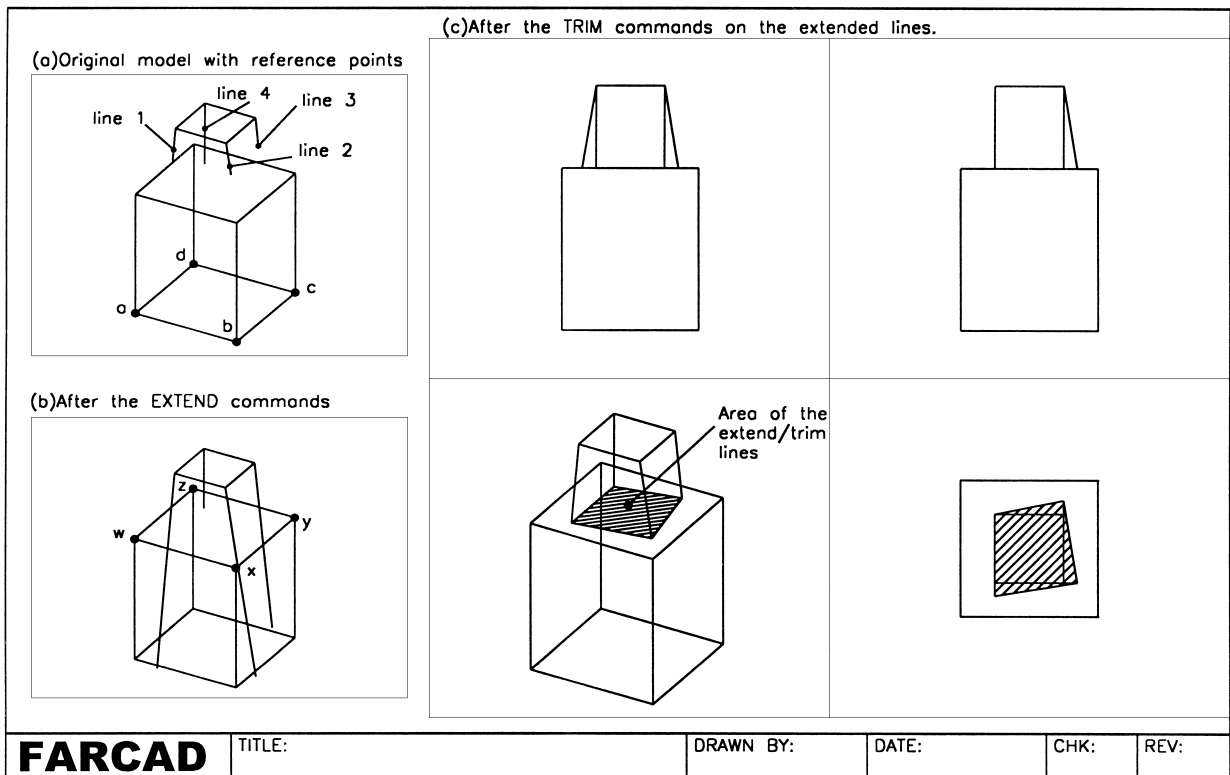


Figure 22.7 The EXTEND and TRIM commands.

Extend

- 1 With the 3D viewport active, restore UCS RIGHT and select the EXTEND icon from the Modify toolbar and:
 - prompt* Select objects
 - respond* **pick line ab then right-click**
 - prompt* Select object to extend or [Project/Edge/Undo]
 - enter* **E <R>** – the edge option
 - prompt* Enter an implied edge extension mode [Extend/No extend]
 - enter* **E <R>** – the extend option
 - prompt* Select object to extend or [Project/Edge/Undo]
 - enter* **P <R>** – the project option
 - prompt* Enter a projection mode [None/Ucs/View]
 - enter* **U <R>** – the current UCS option
 - prompt* Select object to extend
 - respond* **pick blue line 1 then right-click/enter**
- 2 Repeat the EXTEND command using the entries E,E,P,U as step 1 and:
 - a) extend the green line 2 to edge bc with UCS RIGHT
 - b) extend the cyan line 3 to edge cd with UCS FRONT
- 3 The extended lines are displayed as fig(b)

Trim

- 1 Still with the 3D viewport active, restore UCS BASE and select the TRIM icon from the modify toolbar and:
 - prompt* Select objects
 - respond* **pick line wx then right-click**
 - prompt* Select object to trim or [Project/Edge/Undo]
 - enter* **P <R>** – the project option
 - prompt* Enter a projection option [None/Ucs/View]
 - enter* **V <R>** – the view option
 - prompt* Select object to trim
 - respond* a) make the top right viewport active
b) pick the blue line then right-click/enter
- 2 Repeat the TRIM command with P and V entries as step 1 and:
 - a) trim the green line to edge xy, picking the green line in the top left viewport at the select object to trim prompt
 - b) trim the cyan line to edge yz, picking the cyan line in the top right viewport at the select object to trim prompt
- 3 The coloured lines have now been extended and trimmed ‘to the top surface’ of the large red box – fig(c)

Task

- 1 Draw four lines connecting the ‘bottom ends’ of the blue, green, cyan and magenta lines
- 2 Hatch this area
- 3 Find the hatched area and perimeter. My values were:
Area: 3350 and Perimeter: 232.65

Summary

- 1 The commands Rotate3D, Mirror3D and 3DArray are specific to 3D models.
- 2 Rotate 3D allows models to be rotated about the X,Y and Z axes as well as two specified points and about objects
- 3 Mirror 3D allows models to be mirrored about the XY, YZ and ZX axes as well as three specified points and objects
- 3 A rectangular 3D Array is similar to the 2D command but has 'levels' in the Z direction. The result of the polar 3D array can be difficult to 'visualise'
- 4 3D models can be aligned with each other.
- 5 The trim and extend commands can be used in 3D, the result being dependent on the UCS position. Generally objects are trimmed or extended 'to a plane'.

Blocks and Wblocks in 3D

3D blocks and wblocks are created and inserted into a drawing in a similar manner as 2D blocks and wblocks. The UCS position and orientation is critical. In this chapter we will:

- a) create a chess set using blocks
- b) create a wall-clock as wblocks
- c) save both the models for rendering exercises

Creating the models for the blocks

- 1 Open your MV3DSTD template file with MVLAY1 tab, layer MODEL and UCS BASE current, and the lower left viewport active.
- 2 In each viewport, zoom centre about the point 120,90,50 at 275 magnification. Refer to Fig. 23.1

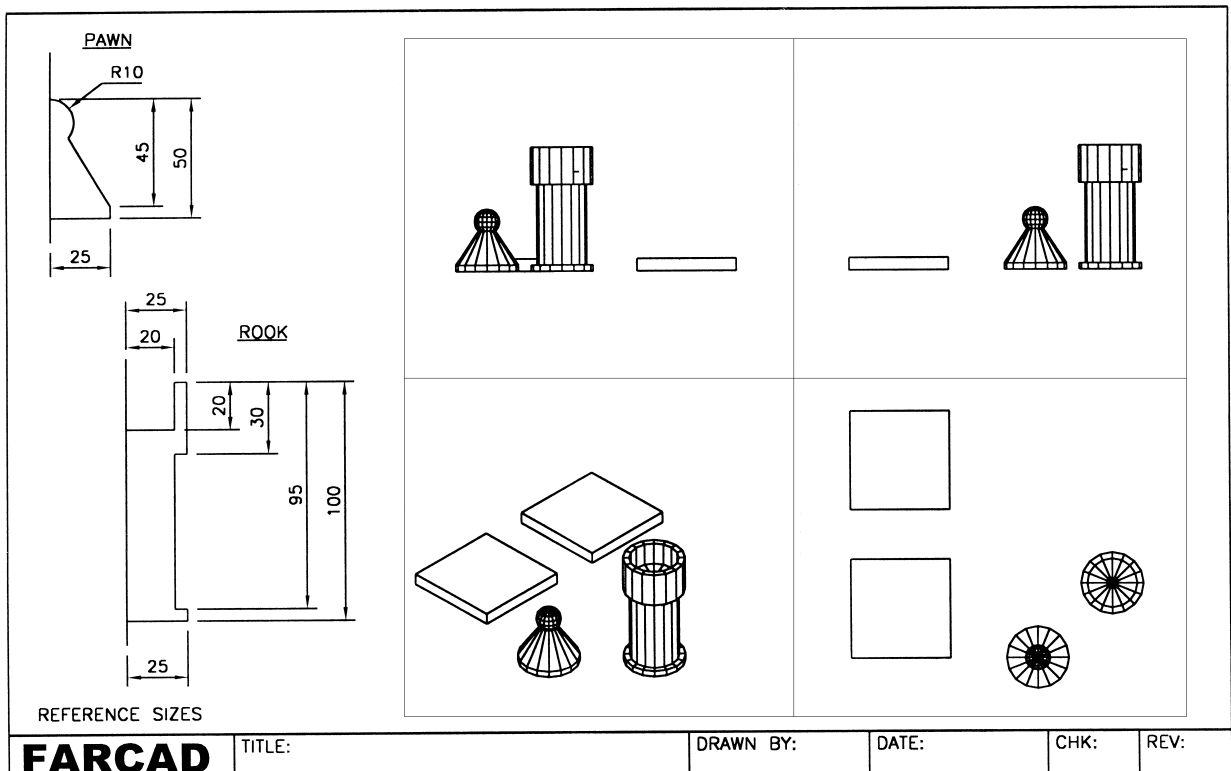


Figure 23.1 The four 3D block details.

- 3 With the lower left viewport active, create the following two 3D box objects:
- | | <i>box1</i> | <i>box2</i> |
|----------|-------------|-------------|
| corner | 0,0,0 | 0,120,0 |
| length | 80 | 80 |
| width | 80 | 80 |
| height | 10 | 10 |
| rotation | 0 | 0 |
| colour | number 126 | number 220 |
- 4 Restore UCS FRONT and make the upper right viewport active.
- 5 Draw a line with start point: 150,-10 and next point: @0,80
- 6 In paper space, zoom-window the top right viewport, then return to model space
- 7 a) Draw the pawn outline as a polyline from: 150,0 using your own design but with the 'overall' sizes given as reference.
 b) Set SURFTAB1 to 16
 c) With the REVOLVED SURFACE icon from the Surfaces toolbar, revolve the pawn outline about the vertical line – full circle
 d) The pawn colour is to be red
- 8 Still with UCS FRONT, draw a line with start point: 210,-10,-60 and next point: @0,120
- 9 a) Draw the rook outline as a polyline from: 210,0,-60 using the reference sizes given or create your own design
 b) Revolve the rook polyline about the vertical line (360 degs)
 c) The rook colour is to be red
- 10 Erase the two vertical lines
- 11 Restore UCS BASE and models displayed as Fig. 23.1.

Making the blocks

- 1 With the lower left viewport active and UCS BASE, menu bar with **Draw-Block-Make** and:
- prompt* Block Definition dialogue box
- respond* 1. enter name: SQ1
 2. enter base point as X: 0; Y:0; Z: 0
 3. pick Select objects and pick the green coloured box then right-click
 4. Objects: ensure Delete active
 5. Preview: Create icon from block geometry active
 6. enter description: FIRST SQUARE
 7. ensure insert units: millimeters (dialogue box as Fig. 23.2)
 8. pick OK
- 2 The green coloured square will be 'made into a block' and should disappear from the screen due to respond 4
- 3 Using the Block Definition dialogue box, make the other three blocks using the same method as step 1 with the following information:
- | | <i>first</i> | <i>second</i> | <i>third</i> |
|--------------|---------------|---------------|--------------|
| name | SQ2 | PAWN | ROOK |
| insertion pt | 0,120,0 | 150,0,0 | 210,60,0 |
| object | purple box | 3D pawn | 3D rook |
| description | SECOND SQUARE | PAWN PIECE | ROOK PIECE |
- 4 Now erase the original polyline outlines which should be displayed
- 5 Activate the block definition dialogue box and scroll at name and the four created blocks should be listed (PAWN, ROOK, SQ1, SQ2) in alphabetical order.

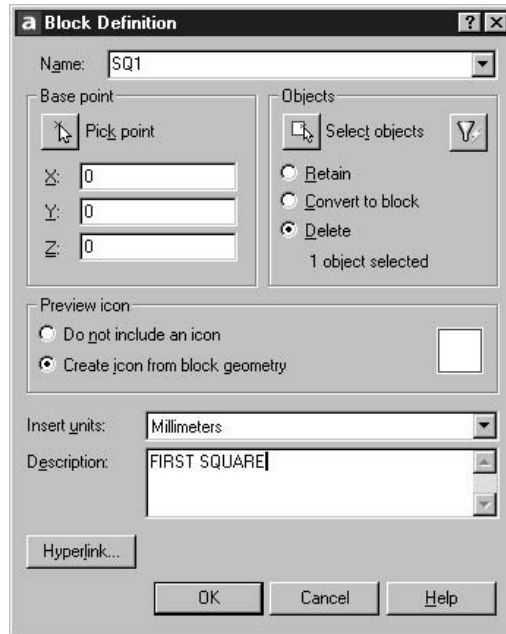


Figure 23.2 Block Definition dialog box for SQ1.

6 *Note:*

- a) The command line BLOCK <R> entry will display the Block Definition dialog box
- b) The command line entry -BLOCK <R> will allow block creation from the command line
- c) At the command line enter -BLOCK <R> and:
 - prompt* Enter block name or [?] and enter: ? <R> – query option
 - prompt* Enter block(s) to list and enter: * <R>
 - prompt* Text window with information about the four defined blocks
 - respond* cancel the text window

Inserting the blocks

- 1 Four 'blank' viewports should be displayed with MVLAY1 tab, UCS BASE, layer MODEL and the 3D viewport active
- 2 Menu bar with **Insert-Block** and:
 - prompt* Insert dialogue box
 - respond*
 1. scroll at name and pick SQ1
 2. deactivate all on-screen prompts, i.e. no tick
 3. enter insertion point as X: 0; Y: 0; Z: 0
 4. enter scale as X: 1; Y: 1; Z: 1
 5. enter rotation as 0
 6. dialogue box as Fig. 23.3
 7. pick OK

and the green 3D box block will be displayed at the 0,0,0 insertion point
- 3 Repeat the Insert-Block sequence with:
 - a) scroll and pick SQ2
 - b) ensure on-screen prompts are off
 - c) insertion point of X: 80; Y: 0; Z: 0
 - d) scale of X: 1; Y: 1; Z: 1 and rotation: 0
 - e) pick OK and the purple 3D box block displayed adjacent to the green 3D box

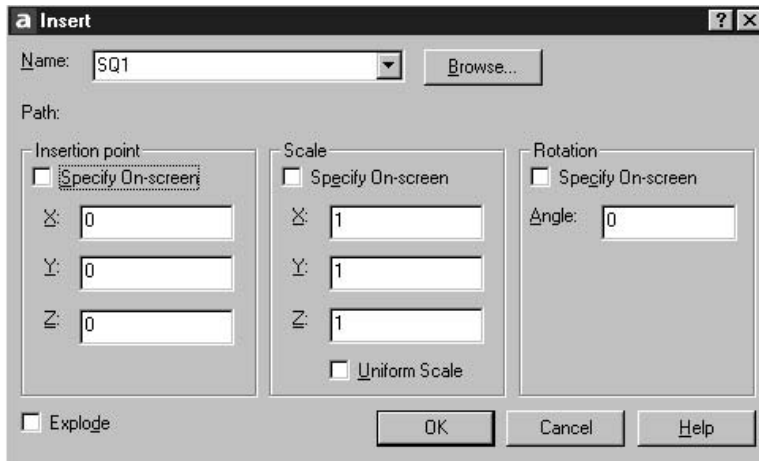


Figure 23.3 The Insert dialogue box for SQ1.

- 4 Zoom centre about 320,320,100 at 700 in all viewports
- 5 With the 3D viewport active, complete the 64 square chess board using one of the following methods:
 - a) inserting each block SQ1 and SQ2
 - b) multiple copy the two inserted boxes
 - c) rectangular array
 - d) note: I used multiple copy
- 6 At the command line enter `-INSERT <R>` and:

prompt Enter block name and enter: **PAWN <R>**

prompt Specify insertion point and enter: **40,120,10 <R>**

prompt Enter X scale factor and enter: **1 <R>**

prompt Enter Y scale factor and enter: **1 <R>**

prompt Specify rotation angle and enter: **0 <R>**

and the red pawn piece will be displayed on top of the left second row square
- 7 Now insert the ROOK full-size with 0 rotation at the insertion point of 40,40,10 using either the Insert dialogue box or the `-INSERT` command line entry.
- 8
 - a) Rectangular array the red pawn for 1 row and 8 columns, the column distance being 80. Row distance is 0?
 - b) Copy the eight pawns from: 40,80,10 to: @0,400,0
 - c) Multiple copy the rook from a base point of 40,40,10 to second points of: @560,0,0; @0,560,0; @560,560,0
- 9 *Task*
 - a) Alter the layout to display a three viewport configuration, remembering to make layer VP current
 - b) set the viewports to display a 3D, top and front view
 - c) Change the colour of one set of pawns and rooks to blue. This may not be as simple as you think. The inserted blocks must be exploded before the colour can be changed. Decide whether to use the PICKFIRST variable set to:
 - i) 0: CHANGE at command line then pick the objects
 - ii) 1: pick the objects then the Properties icon
 - d) re-centre the model in each viewport about 320,320,100 at 1100 magnification in the 3D viewport and 700 in the other two
- 10 When complete, the layout should resemble Fig. 23.4 and should be saved as **MODR2002\CHESS** for the activity part of the chapter.

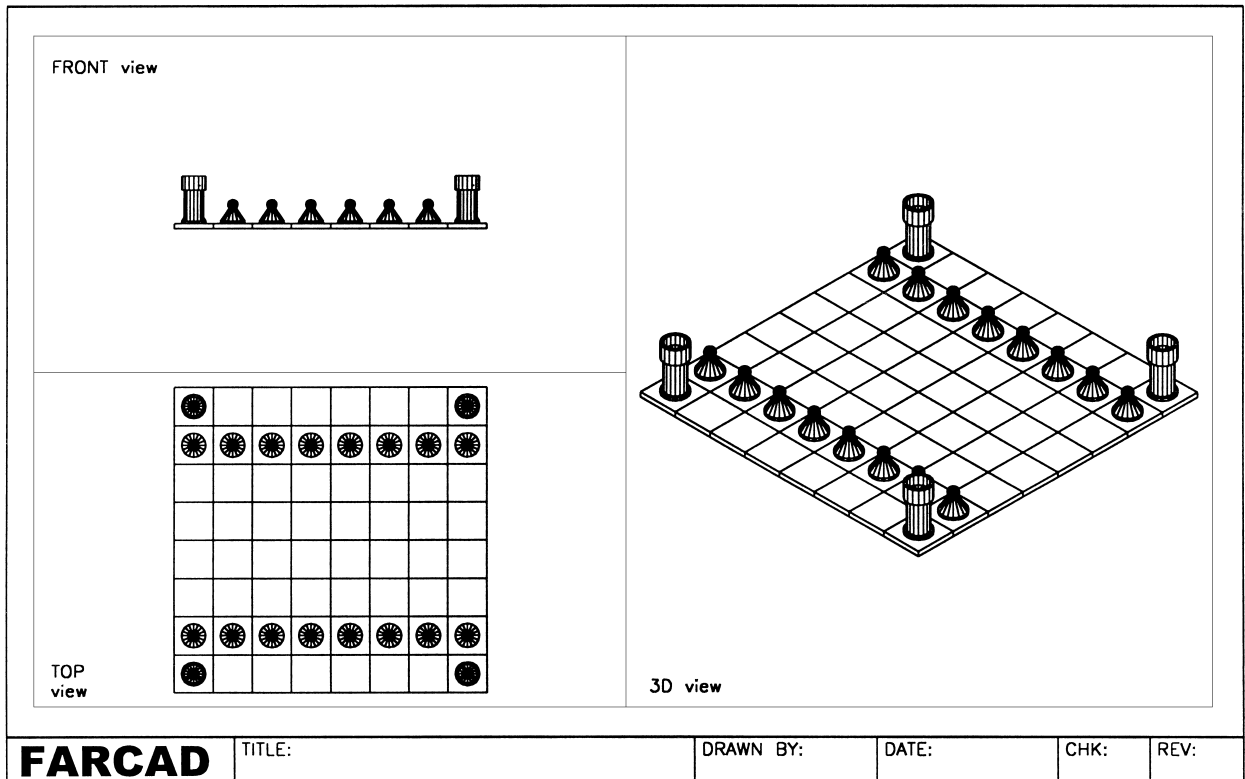


Figure 23.4 The chess layout after the block insertions.

Creating the models for the Wblocks

- 1 Open the MV3DSTD template file with MVLAY1 tab, UCS BASE and zoom centre about 125,0,100 at 300 magnification in all viewports. Refer to Fig. 23.5
- 2 The wblocks will be created using the WCS, so menu bar with **Tools-New UCS-World**
- 3 With the top right viewport active, layer MODEL current rotate the WCS about the X axis with menu bar sequence **Tools-New UCS-X** and enter 90 as the rotation angle
- 4 Create the three outlines for parts of a wall clock with:
 - a) draw the body as lines using the 0,0,0 start point and the reference sizes given. Use the Modify-Object-Polyline command to 'convert' the five lines into a single polyline object
 - b) draw the face as an octagon circumscribed in a circle with centre at 90,90,0 and a radius of 40
 - c) draw the dial as a 30 radius circle, centre at 90,10,0
- 5 Draw the following three lines:
 - a) line 1, start point: 150,0 next point: @0,0,40
 - b) line 2, start point: 200,0 next point: @0,0,15
 - c) line 3, start point: 250,0 next point: @0,0,8
- 6 The three wall clock components will be created as tabulated surface models, so set SURFTAB1 to 16

- 7 With the TABULATED SURFACE icon from the Surfaces toolbar:
 - prompt* Select object for path curve
 - respond* **pick the BODY polyline**
 - prompt* Select object for direction vector
 - respond* **pick line 1 at end indicated by the donut**
 - and* extruded red tabulated surface model of wall clock body
- 8 Repeat the tabulated surface command and:
 - a) select the FACE octagon as the path curve and line 2 as the direction vector at end indicated
 - b) select the DIAL circle as the path curve and line 3 as the direction vector at end indicated
- 9 Change the colour of the tabulated surface models:
 - FACE: blue
 - DIAL: green
- 10 The layout at this stage should resemble Fig. 23.5

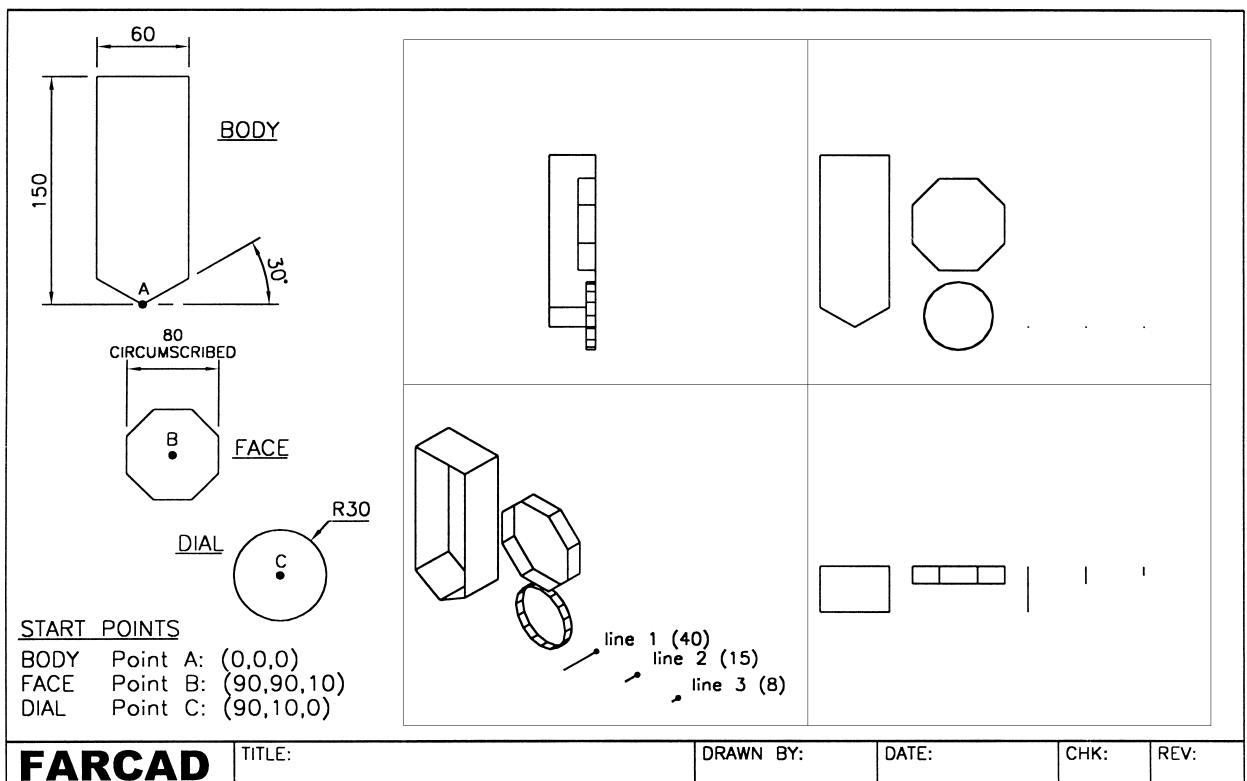


Figure 23.5 Information for creating the three wblocks.

Making the Wblocks

- 1 Menu bar with **Tools-New UCS-World** to restore the WCS and make the lower left viewport active
- 2 At the command line enter **WBLOCK <R>** and:
 - prompt* Write Block dialogue box
 - respond* 1. Source: Objects
 - 2. Base point: X: 0; Y: 0; Z: 0
 - 3. Objects: Delete from drawing
 - 4. pick Select objects and:
 - prompt* Select objects at command line
 - respond* **pick the red body then right-click**
 - prompt* Write Block dialogue box
 - respond* 1. Destination file name: BODY
 - 2. Location: C:\modr2000 (or your named folder)
 - 3. Insert units: Millimeters – Fig. 23.6
 - 4. pick OK

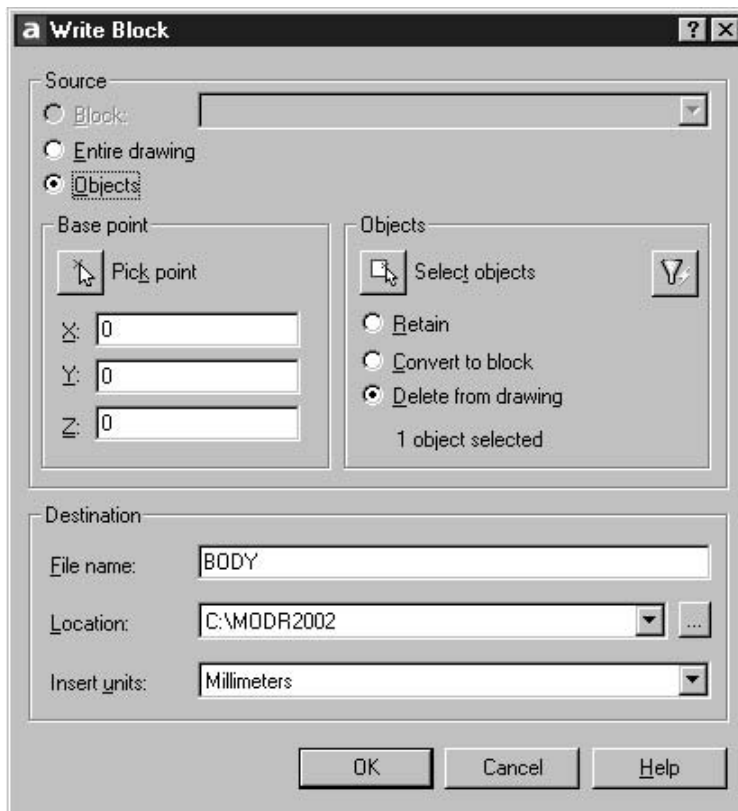


Figure 23.6 The Write Block dialogue box for BODY.

- 3 Create another two wblocks using the same procedure as step 2 with the following information:

	<i>wblock2</i>	<i>wblock3</i>
Insertion base point	90,0,90	90,0,10
Objects	blue object	green object
File name	FACE	DIAL
Location	C:\modr2002	C:\modr2002

Inserting the three wblocks

- 1 Menu bar with **File-Close** and pick No to save changes
- 2 Menu bar with **File-New** and 'open' your MV3DSTD template file with UCS BASE, layer MODEL, MVLAY1 tab and lower left viewport active
- 3 Restore the WCS
- 4 Menu bar with **Insert-Block** and:
 - prompt* Insert dialogue box
 - respond* **pick Browse**
 - prompt* Select Drawing File dialogue box
 - respond*
 1. scroll and pick the C: drive
 2. scroll and pick C:\modr2000 (or your named folder)
 3. pick BODY and note the preview
 4. pick Open
 - prompt* Insert dialogue box
 - respond*
 1. cancel the on-screen prompts
 2. enter insertion point as X: 0; Y: 0; Z: 0
 3. all scale factors 1 and rotation angle 0
 4. pick OK

and the red body will be displayed at the 0,0,0 insertion point
- 5 Activate the Insert dialogue, select the Browse option and insert the other wblocks with the following information:

File name:	FACE	DIAL
Insertion point:	0,-40,130	0,-55,130
X and Y scale:	1	1
Rotation angle:	0	0
- 6 Zoom centre about the point 0,-30,80 at 200 magnification in all viewports – Fig. 23.7
- 7 Investigate the other layout tabs, centring the model using the following suggestion: Zoom-Extents then Zoom to scale 1.5
- 8 Hide and shade from the model tab and try 3D orbit
- 9 Save the model as **MODR2002\WALL**

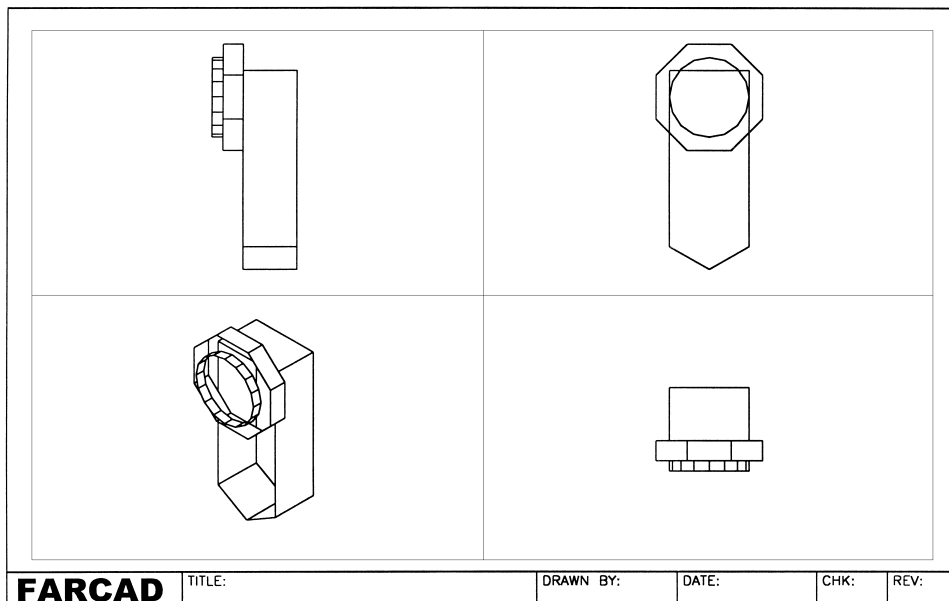


Figure 23.7 The completed wall clock created from inserted wblocks.

Summary

- 1 3D blocks and wblocks are created and inserted in a similar manner to 2D blocks and wblocks
- 2 With 3D blocks, the position of the UCS is important
- 3 With 3D wblocks it is **strongly recommended** that the WCS be restore when creating and inserting the wblocks
- 4 It is also recommended that wblocks are 'stored' in the same folder as drawing file from they were created and into which they will be inserted
- 5 Wblocks which are inserted 'unexploded' into a drawing become blocks within that drawing. It is therefore recommended that wblocks are exploded as they are inserted.

The AutoCAD Design Centre

Before leaving this chapter and attempting the activities, we will investigate using the Design Centre. All AutoCAD users should now be aware that blocks created in a drawing can be inserted into any other drawing using the Design Centre.

- 1 Close all existing drawings then open the 3DSTDA3 standard sheet created before model/paper space was discussed
- 2 Menu bar with **Tools-AutoCAD Design Center** to display the Design Center dialogue box and position to suit
- 3 From the Design Centre menu bar select Preview and Description
- 4 In the hierarchy side (left) of the dialogue box:
 - a) navigate the your named folder
 - b) scroll until CHESS.dwg is displayed
 - c) expand CHESS – pick the (+) at drawing icon
 - d) explore Blocks, i.e. right-click on Blocks
 - e) dialogue box displayed as Fig. 23.8

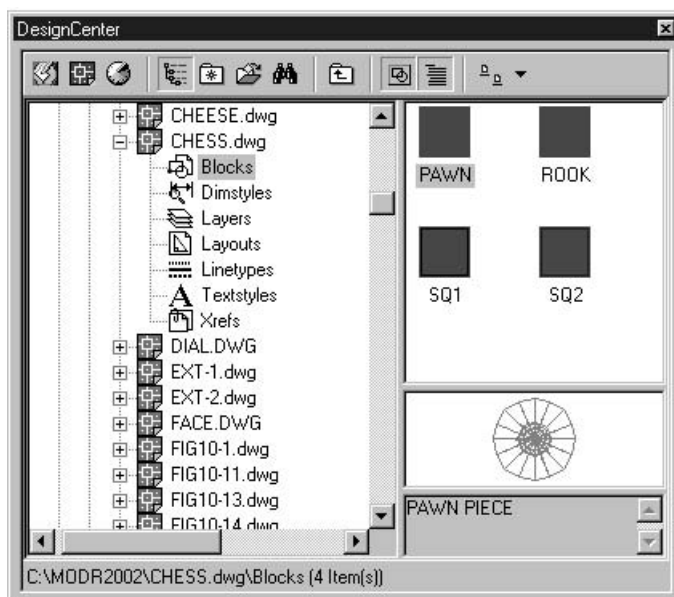


Figure 23.8 The Design Centre dialogue box with blocks explored for C:\MODR2002\CHESS.

- 5 Right-click the SQL icon from the Design Center palette and:
 - prompt* Shortcut menu
 - respond* **pick Insert block**
 - prompt* Insert dialogue box
 - with* SQL named and dialogue box as before
 - respond* cancel the dialogue box and the Design Center
- 6 This exercise was to demonstrate that the Design Center can be used to insert **ANY block from any drawing into any other drawing**

Assignments

Two activities have been included for you to attempt, one involving the partially completed chess set, and the other using two previously saved drawings, one of which will be inserted as a wblock into the other.

ACTIVITY 15: CHESS SET

- 1 Recall the drawing CHESS saved earlier in this chapter to display the 64 square chess board with the two sets of red and blue pawns and rooks.
- 2 Design the other chess pieces – KNIGHT, BISHOP, KING and QUEEN using the same method as the worked example:
 - a) draw the outline as a polyline
 - b) use the revolved surface command to create the piece as a 3D surface model
 - c) the actual shape of the pieces is at your discretion.
 - d) the information given in the Activity 15 drawing is for reference purposes only
 - e) ensure that your start point for the outline is known – it will be useful as the block insertion point
- 3 Create a block of each created piece
- 4 Insert the created blocks onto the chess board
- 5 Complete the chess set layout, remembering to change the colours of the pieces to red and blue as appropriate.
- 6 Save as MODR2002\CHESS.
- 7 Investigate the various shade options with the completed model
- 8 *Note:* the model layout displayed in activity 15 was produced by Craig Matthewman. Craig was a full-time student on the HNC CADD course at Motherwell College and his design was interesting. He gave his permission for the layout to be included as an activity.

ACTIVITY 16: Palace of Queen NEFERSAYDY built by MACFARAMUS.

MACFARAMUS was last encountered building the palace for queen NEFERSAYDY. Unfortunately this palace was to be built on a flat topped hill and you have to create the layout using an existing drawing and inserting another drawing into it as a wblock.

- 1 Open the drawing MODR2002\HILL of the edge surface model created as activity 13
- 2 Insert the wblock drawing file MODR2002\PALACE of the 3D objects created as activity 14
- 3 The palace has to be positioned at the centre point of the hill top, and the coordinates of this point as 0,0,100. This is the only help given.
- 4 Optimise all the layout tabs for maximum effect.
- 5 When complete save the layout as MODR2002\HILLPAL

Dynamic viewing

Dynamic viewing is a powerful (yet underused) command which is very useful with 3D modelling as it allows models to be viewed from a perspective viewpoint. The command also allows objects to be 'cut-away' enabling the user to 'see inside' models. Dynamic viewing has its own terminology which is obvious when you are familiar with the command, but can be confusing to new users.

The basic concept of dynamic viewing is that the user has a **CAMERA** which is positioned at a certain **DISTANCE** from the model – called the **TARGET**. The user is looking through the camera lens at the model and can **ZOOM** in/out as required. The viewing direction is from the camera lens to a **TARGET POINT** on the model. The camera can be moved relative to the stationary target, and both the camera and target can be turned relative to each other. The target can also be **TWISTED** relative to the camera. Two other concepts which the user will encounter with the dynamic view command are the **slider bar** and the **perspective icon**. The slider bar allows the user to 'scale' the variable which is current, while the perspective icon is displayed when the perspective view is 'on'.

Figure 24.1(A) displays the various dynamic view concepts of:

- a) the basic terminology
- b) the slider bar
- c) the perspective icon

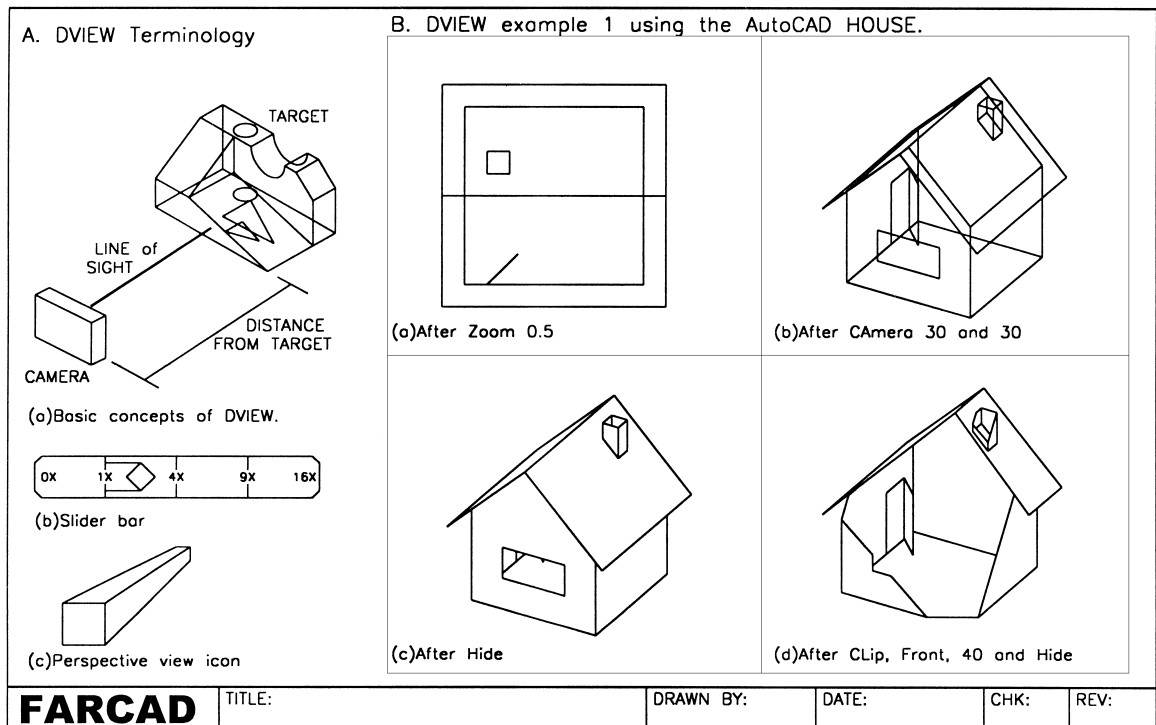


Figure 24.1 Dynamic view terminology and AutoCAD's DVIEWBLOCK 'house'.

The dynamic view command has eleven options, these being:

CAmera, TArget, DIstance, POints, Pan, Zoom, TWist, CLip, HIde, Off, Undo

The option required is activated by entering the CAPITAL letters at the command line, e.g. CA for the camera option, TW for twist etc.

We will investigate the dynamic view command with two examples:

- 1 using AutoCAD's dynamic view 'house'
- 2 with a previously created and saved model

Note

- 1 Dynamic view is a model space concept, and cannot be used in paper space.
- 2 Dynamic view is **viewport independent**, i.e. if the command is used in a specific viewport, the model display in the other viewports will not be affected.
- 3 The command is activated by entering **DVIEW <R>** at the command line

Example 1 – AutoCAD's 'house'

AutoCAD has a 'drawing' – actually a type of block – which can be used as an interactive aid with the dynamic view command. We will use this house block to demonstrate some of the options so:

- 1 Close any existing drawings and start a new metric drawing from scratch and refer to Fig. 24.1(B)
- 2 At the command line enter **DVIEW <R>** and:
 - prompt* Select objects or <use DVIEWBLOCK>
 - respond* **<RETURN>** i.e. accept the DVIEWBLOCK default
 - a) *prompt* Enter option
[Camera/Target/Distance/Points/Pan/Zoom/Twist/Clip/Hide/Off/Undo]
 - and* some coloured lines appear on the screen
 - enter* **Z <R>** – the zoom option
 - prompt* slider bar with scale displayed at top of screen
 - and* Specify zoom scale factor
 - enter* **0.5 <R>**
 - and* full plan view of house – fig(a)
 - b) *prompt* Enter option [CAmera/TArget/etc]
 - enter* **CA <R>** – the camera option
 - prompt* **ghost image of house** which moves as mouse moved
 - and* Specify camera location or enter angle from XY plane
 - enter* **30 <R>**
 - prompt* Specify camera location or enter angle in XY plane from X axis
 - enter* **30 <R>**
 - and* 3D view of house – fig(b)
 - c) *prompt* Enter option [CAmera/TArget/etc]
 - enter* **H <R>** – the hide option
 - and* house displayed with hidden line removal – fig(c)

- d) *prompt* Enter option [CAmera/TARget/etc
enter **CL <R>** – the clip option
prompt Enter clipping option [Back/Front/Off]
enter **F <R>** – the front clip option
prompt Specify distance from target or [set to Eye(camera)/ON/OFF]
enter **40 <R>**
prompt Enter option [CAmera/TARget/etc
enter **H <R>** – the hide option
and house displayed 'cut-away' similar to fig(d)
e) *enter* **U <R>** – undoes the hide effect of (d)
enter **U <R>** – undoes the clip effect of (d)
enter **U <R>** – undoes the hide effect of (c)
and **leave house with Camera option displayed**
and **command prompt line options**
then **read the explanation before proceeding**

Explanation of the dynamic view command

Dynamic view is an **interactive** command and the various options can be used one after the other. The undo (U) option will undo the last option performed, and can be used repeatedly until all the options entered have been 'undone'. Some of the options have been used to demonstrate how the command is used, these options being zoom, camera, clip, hide and undo. The hide option is very useful as it allows the model to be displayed when other options have been entered, and removes the 'ambiguity' effect from the model. The command can be used with all 3D models, i.e. extruded, wire-frame, surface and solid. The command is also **viewport independent**, i.e. it can be used in any viewport without affecting the display in other viewports. The AutoCAD 'house' is a user-reference, and if a model is displayed on the screen, this model will assume the house orientation when the dynamic view command is completed. This will be investigated during the next example.

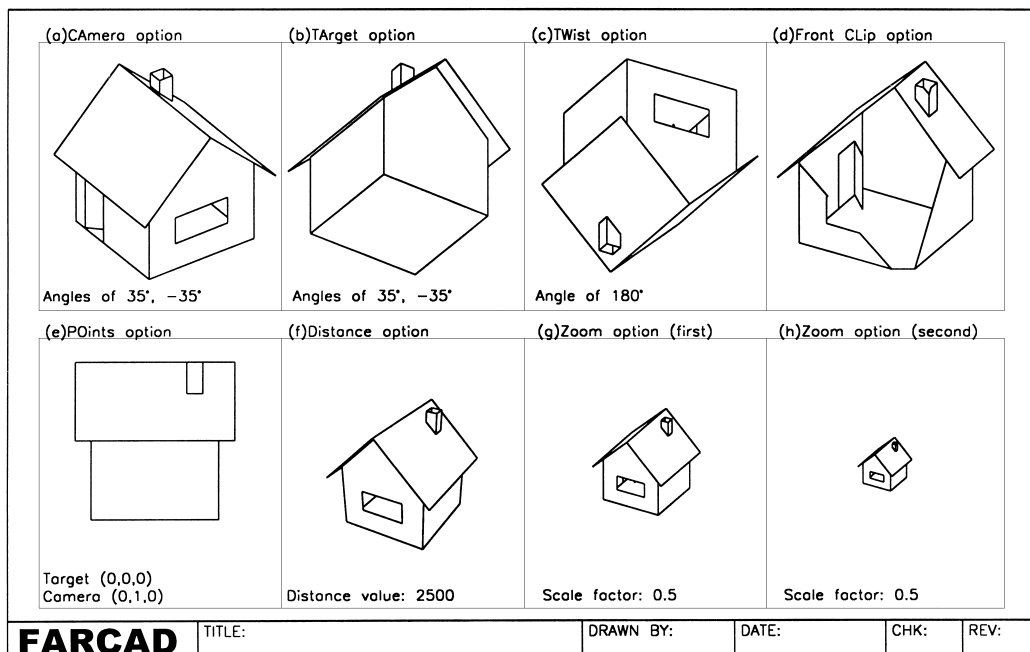


Figure 24.2 The various DVIEW options with DVIEWBLOCK – AutoCAD's house.

The house displayed on the screen has been left with the Camera option with entered angles of 30 and 30. We will continue with the screen display and investigate the other dynamic view options. This means that you have to enter the various options and values as prompted.

Refer to Fig. 24.2 which displays one entry (with hide) from the exercises which follow.

CAmera

- 1 This option is used to direct the camera at the target and the camera can be 'tilted' relative to the target. These are two angle prompts:
prompt 1 angle in the XY plane, between -90degs and +90degs
prompt 2 angle from the XY plane, between -180degs and +180degs
- 2 The angles can be:
 - a) toggled using the ghost image as a guide
 - b) entered directly from the keyboard
- 3 Using the CAmera option enter the following angle values:

<i>angle in XY plane</i>	<i>angle from XY plane</i>
a) 35	35
b) 35	-35
c) -35	35
d) -35	-35
- 4 The option can be considered similar to **VPOINT-ROTATE**
- 5 When all the above entries have been completed, return the camera angles to the original 30 and 30, but do not exit the command.

TArget

- 1 This option allows the target (the model) to be tilted relative to the camera. The two angle prompts are the same as the camera option:
prompt 1 angle in the XY plane
prompt 2 angle from the XY plane
- 2 The angles can be toggled or entered from the keyboard
- 3 Using the TArget option enter the following angle values:

<i>angle in XY plane</i>	<i>angle from XY plane</i>
a) 35	35
b) 35	-35
c) -35	35
d) -35	-35
- 4 The option can be used to give the same effect as the camera option, but it should be remembered that the camera and target are being 'tilted' in the 'opposite sense' to each other
- 5 When all angles have been entered, restore the camera to angles of 30 and 30, but do not exit the command.

Twist

- 1 A very useful option as it allows the 'plane' on which the target is 'resting' to be twisted through an entered angle. This angle can be positive or negative and have values between 0 and 360degs
- 2 The prompt with this option is: Specify new view twist angle
- 3 The result of the option is dependent on the CAmera/Target angles

- 4 Using the TWist option enter angle values of:
a) 35 b) -35 c) 180 d) -90
- 5 This is one of the few AutoCAD commands which allows models to be 'flipped' over by 180degs.
- 6 When the four twist angles have been viewed with the hide effect, restore the original twist angle of 0, with the camera options of 30 and 30. Do not exit the command.

CLip

- 1 The clip option of the dynamic view command is probably of the most useful of all the options, as it allows models to be 'cut-away', thus allowing the user to 'see inside' the model.
- 2 The user selects a (F)ront or (B)ack clip and then decides on the clip distance either:
a) using the slider bar
b) entering a value at the command line
- 3 The result of the clip option is dependent on the CAmera/TArget angles as well as the 'size' of the model on the screen
- 4 With the CAmera angles set to 30 and 30, activate the front clip option and move the slider bar until the ghost image displays a clip effect then right-click
- 5 When the clip option attempts are complete, undo the various effects with **U <R>** until the house is again displayed with the CAmera settings of 30 and 30.

POints

- 1 This option allows the model (the target) to be viewed from a specific 'stand point', the user looking at a specific point on the target.
- 2 Two sets of coordinates need to be specified:
a) the target point coordinates to be looked at
b) the coordinates of the camera – the user
- 3 The coordinate entries can be absolute or relative
- 4 When this option is used, the PAn option is also usually needed
- 5 The result does not depend on the CAmera or TArget options
- 6 Use the Point option with the following entries:

<i>target point</i>	<i>camera point</i>
a) 0,0,0	1,0,0
b) 0,0,0	0,1,0
c) 0,0,0	0,0,1
d) 0,0,0	1,1,0
e) 0,0,0	1,0,1
f) 0,0,0	1,1,1
g) 1,2,3	0,0,0
h) 0,0,0	1,2,3
- 7 The option is similar to the **VPOINT-VECTOR** command.
- 8 When all the points entries have been entered, restore the camera angles of 30 and 30, but do not exit the command.

Distance

- 1 Alters the distance between the camera and the target
- 2 The distance can be:
 - a) entered as a value from the command line
 - b) toggled using the slider bar
- 3 With the distance option, enter some values e.g.:
 - a) 1000 b) 1500 c) 2500 d) 5000
- 4 The distance option introduces **true perspective** to the model
- 5 When the distance option has been used, and the command is exited, the zoom command cannot be used.
- 6 Restore the original CAMera option of 30 and 30

Zoom

- 1 This option does what you would expect – it ‘zooms the model’
- 2 The zoom factor can be:
 - a) entered as a value from the keyboard
 - b) toggled using the slider bar
- 3 Try some zoom entries e.g.:
 - a) 1 b) 0.75 c) 0.5 d) 0.25
- 4 Restore the original CAMera effect (30,30) and use the zoom option with a 0.5 scale factor three times.

Pan

- 1 This option is similar to the AutoCAD PAN command, but the ‘real-time’ pan effect is not available.
- 2 The user selects (or enters) the pan displacement

Hide

- 1 Will display the model with a hide effect
- 2 Removes any ambiguity

Undo

- 1 Entering **U <R>** will undo the last option of the DVIEW command
- 2 Can be used repetitively until all the option entries have been undone

eXit

- 1 Entering **X <R>** will end the dynamic view command and a blank screen will be returned
- 2 The blank screen is because we did not have any model displayed, the AutoCAD ‘house’ being a visual aid indicating what any model would ‘look like’
- 3 With ‘real models’, the model orientation will be similar to the ‘house’ orientation when the DVIEW command has been exited, as will now be investigated.
- 4 If the DVIEW command is still active, enter X <R>
- 5 This first exercise is now complete.

Example 2 – an 3D model

In this example we will use the dynamic view command with a previously created model.

- 1 Open the ruled surface model MODR2002\ARCHES created during chapter 16 with UCS BASE, layer MODEL. Re-centre the model about the point 50,10,75 at 150 mag.
- 2 Make the upper left viewport active and refer to Fig. 24.3
- 3 At the command line enter **DVIEW <R>** and:
 - prompt* Select objects or use <DVIEWBLOCK>
 - respond* <RETURN>
 - prompt* AutoCAD's house as an 'end view' in the active viewport
 - and* dynamic view options
 - enter* **CA <R>**
 - prompt* Angle from XY plane and enter: **-20 <R>**
 - prompt* Angle in XY plane and enter: **30 <R>**
 - prompt* dynamic view options and enter: **X <R>**
- 4 The model will be displayed with house CAmera configuration as fig(b) with hide
- 5 With the top right viewport active, enter **DVIEW <R>** at the command line and:
 - prompt* Select objects or use <DVIEWBLOCK>
 - respond* **window the model then right-click**
 - prompt* dynamic view options
 - enter* **TW <R>**
 - prompt* Specify view twist angle and enter: **-90 <R>**
 - prompt* dynamic view options and enter: **X <R>**
 - and* Model displayed with new twist as fig(c)

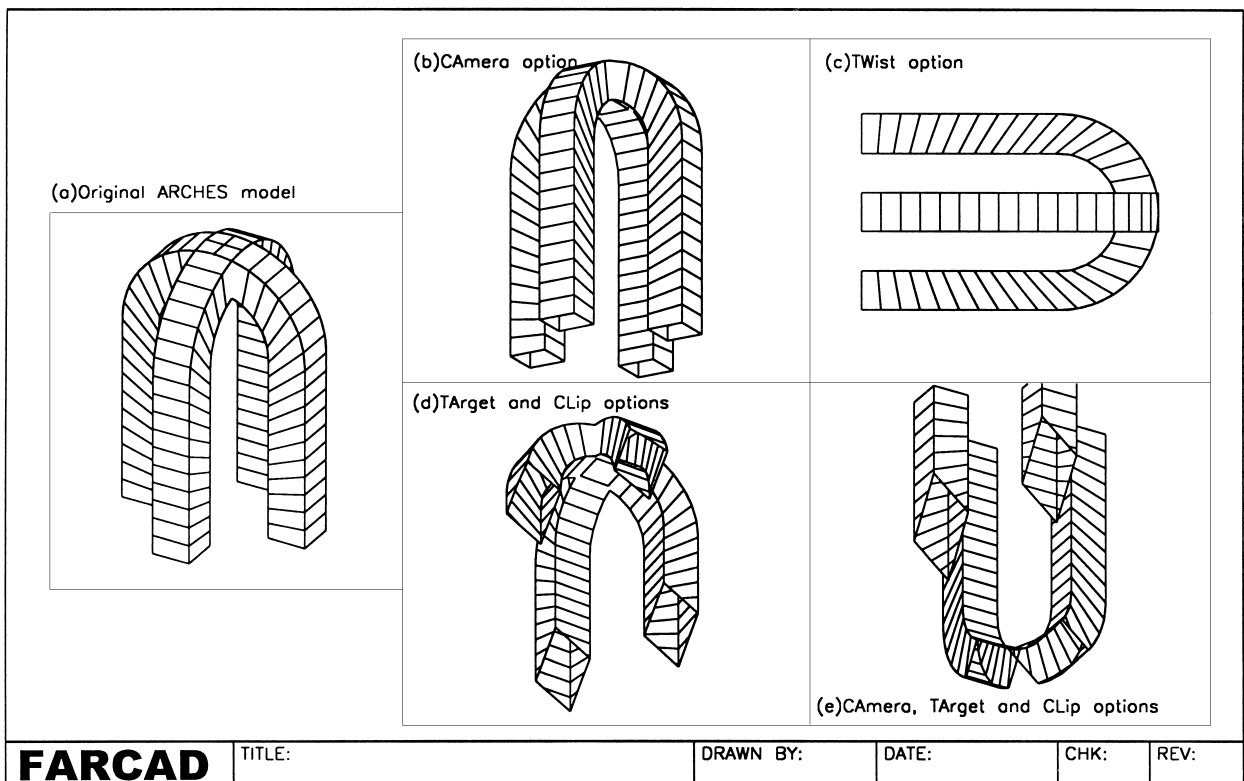


Figure 24.3 Dynamic view example 2 using MODR2002\ARCHES.

- 6 Use the dynamic view command in the lower viewports with the following entries:
- | | |
|----------------------|------------------------------------|
| <i>lower left</i> | <i>lower right</i> |
| options: TArget | options: CAmera |
| angles: 40 and -30 | angles: 30 and 30 |
| options: CLip, Front | options: TWist, angle: 180 |
| distance: 10 | options: CLip, Front, Distance: 15 |
| options: X | options: X |
| fig(d) | fig(e) |
- 7 *Note:* if the clip-front distances do not give the exact same effect as Fig. 24.3 it could be as a result of your model having a different zoom effect from mine.
- 8 Save the drawing if required, but we will not refer to this exercise again.

Summary

- 1 The dynamic view command is viewport specific, i.e. it only affects the active viewport
- 2 The command has several useful options:
 - a) CAmera, TArget: similar to VPOINT rotate
 - b) TWist: allows models to be 'inverted'
 - c) Distance: introduces true perspective
 - d) CLip: useful to 'see inside' models
- 3 The command can only be activated with DVIEW <R> at the command line
- 4 The command can be used:
 - a) directly on models
 - b) interactively using AutoCAD's house
- 5 The command is used relative to the WCS – observe the prompt line when the command is activated.

Assignment

No specific activity, but investigate DVIEW with some previously created models.

Viewport specific layers

When layers are used with multiple viewports they are generally **GLOBAL**, i.e. what is drawn on a layer in one viewport will be displayed in the other viewports. This is quite acceptable for creating models but is unacceptable for certain other concepts, e.g. adding dimensions to a model, sectioning the model, obtaining a true shape etc. If dimensions (for example) have to be added to a model in a multi-view layout, then these dimensions should only be visible in the active viewport. This is also true for true shapes and sections.

In this chapter we will investigate how to create and use viewport specific layers by adding dimensions to an existing model.

Note

- 1 Remember that dimensioning is a 2D concept, the result depending on the position and orientation of the UCS. Think back to when we dimensioned the 3D wire-frame model.
- 2 In later chapters we will use viewport specific layers to extract sections and true shapes from solid models
- 3 Although dimensioning is being used to demonstrate using viewport specific layers, later chapters will introduce the user to other methods of dimensioning models.

Global layers

- 1 Open the 3D faced model MODR2002\CHEESE from chapter 14 and refer to Fig. 25.1
- 2 With the MVLAY1 tab current, UCS BASE, lower right viewport active make layer DIM current and display the Dimension toolbar
- 3 Select the LINEAR DIMENSION icon from the Dimension toolbar and dimension lines 1-2 and 1-3.
- 4 Make the upper right viewport active and restore UCS FRONT
- 5 Linear dimension line a-b and baseline dimension lines a-c and a-d.
- 6 The five dimensions will be displayed in all four viewports due to the GLOBAL nature of layer DIM. Fig. 25.1(a) displays the five dimensions as displayed in the 3D viewport.
- 7 Now erase the five dimensions and restore UCS BASE

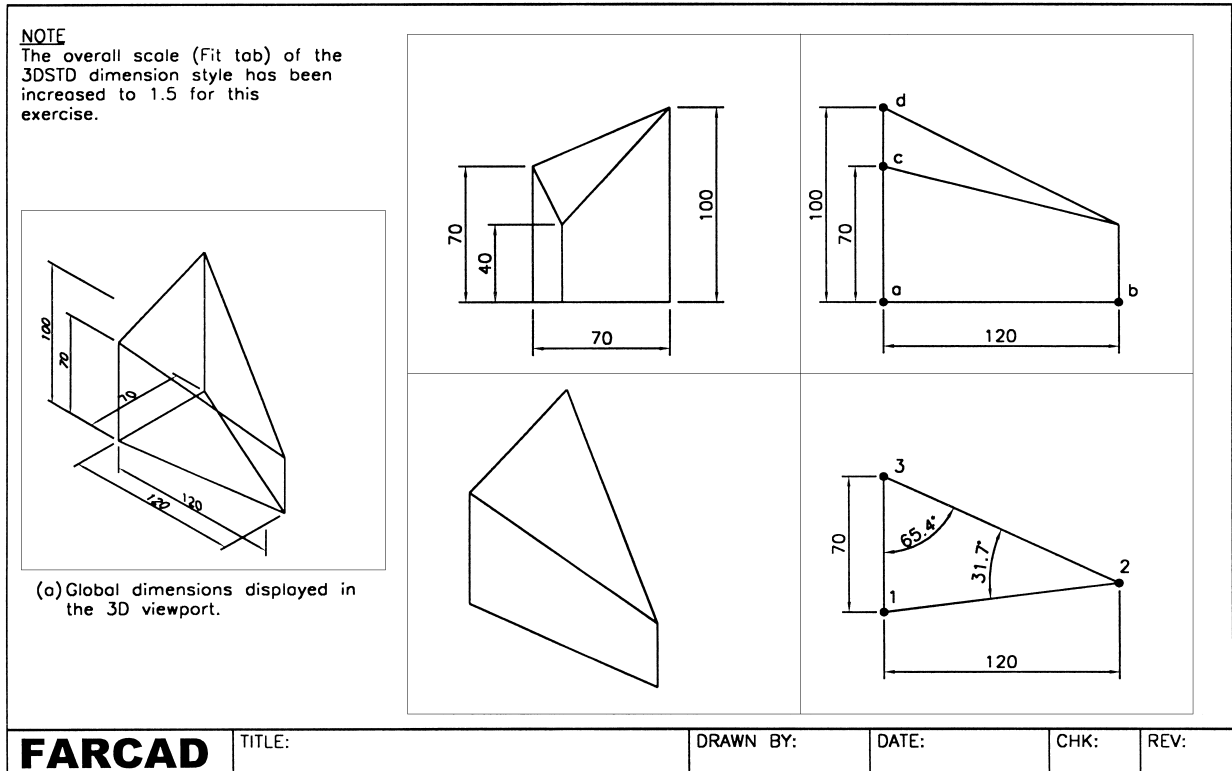


Figure 25.1 Viewport specific layer exercise using MODR2002\CHEESE

Viewport specific layers

- 1 Still with the 3D faced model displayed on screen?
- 2 Menu bar with **Format-Layer** and:
 - prompt* Layer Properties Manager dialogue box
 - respond*
 1. pick the DIM layer line – becomes highlighted
 2. pick New three times to make three new magenta layers: Layer1, Layer2 and Layer3
 3. alter the new layer names to:
Layer1: DIMTL; Layer2: DIMTR; Layer3: DIMBR
 4. The nomenclature for these new layer names is for individual viewports, i.e. TL: top left; BR: bottom right; TR: top right
 5. pick OK
- 3 In model space make the top left viewport active and:
 - a) activate the Layer Properties manager dialogue box
 - b) select the Show Details option
 - c) hold down the control key (Ctrl) and pick the DIMTR and DIMBR layer lines
 - d) pick the Freeze in Current viewport option (tick)
 - e) note the Current VP icon changes from yellow to blue
 - f) pick OK
- 4 With the top right viewport active:
 - a) activate the Layer Properties Manager dialogue box
 - b) by selecting the Current VP Freeze icon, freeze in the current viewports, the new layers DIMTL and DIMBR
 - c) pick OK

- 5 With the lower right viewport active use the Layer Properties Manager dialogue box to freeze layers DIMTL and DIMTR in the current viewport
- 6 In the lower left viewport freeze in the current viewport, the three new layers DIMTL, DIMTR and DIMBR
- 7 Re-centre the model in top left, top right and lower right viewports about the point 60,35,50 at 175 magnification. This will allow additional 'space' for the dimensions
- 8 With UCS BASE and the lower right viewport active:
 - a) make layer DIMBR current
 - b) linear dimension lines 1–2 and 1–3
 - c) angular dimension any two angles using the specify vertex option (snap to endpoint helps)
 - d) the four dimensions should only be displayed in the lower right viewport?
- 9 Restore UCS FRONT and make the upper right viewport active and:
 - a) make layer DIMTR current
 - b) linear dimension line a–b and baseline dimension lines a–c and a–d
 - c) the three dimensions are only displayed in the top right viewport
- 10
 - a) restore UCS RIGHT
 - b) make the upper left viewport active
 - c) make layer DIMTL current
 - d) add the four dimensions as Fig. 25.1
- 11 *Notes*
 - a) All dimensions should have been added to the viewport which was active when the command was used
 - b) The dimensions have been added to a viewport specific layer which was made current in the viewport where the dimensions had to be added
 - c) Generally the 3D viewport does not require a dimension layer as dimensions are not usually added to a 3D viewport
 - d) In the 3D viewport the three viewport specific layers were all currently frozen
 - e) The Overall Dimension Scale (Fit tab) of the 3DSTD dimension style was set to 1.5 for this exercise.

Layer states

- 1 Layers can have different **states** depending on whether they are global or viewport specific, these states being easily controlled:
 - a) from the Layer Properties Manager dialogue box
 - b) from the icons displayed in the Object Properties toolbar
- 2 The different states are:

<i>Global</i>	<i>Viewport specific</i>
On/Off	On/Off
Freeze/Thaw	Freeze/Thaw in all viewports
Lock/Unlock	Freeze/Thaw in active viewports
Plot/non-plot	Freeze/Thaw in new viewports
	Lock/Unlock
	Plot/non-plot
- 3 Layers can have more than one state active at a time , e.g. they can be:
 - a) on and locked
 - b) on and currently frozen etc
- 4 Figure 25.2 displays the Layer Properties Manager dialogue box layers icons and names and:
 - a) an icon displayed in yellow is on or thawed
 - b) an icon displayed in blue is off or frozen

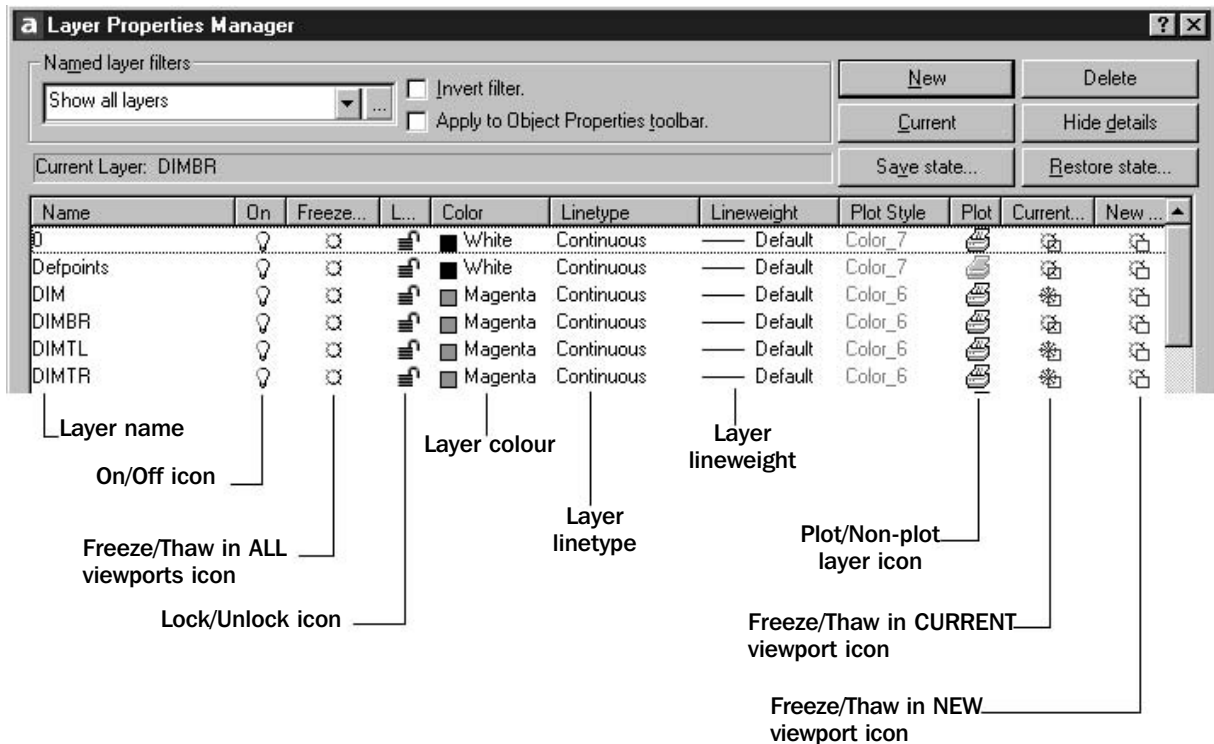


Figure 25.2 The Layer Properties Manager dialogue box with layer state descriptions.

Summary

- 1 Viewport specific layers are layers which are specific to a named viewport
- 2 Viewport specific layers are used with multi-view drawings and are essential for such concepts as dimensioning and extracting details from solid models
- 3 Viewport specific layers can **ONLY BE CREATED** if the TILEMODE variable is set to 0, i.e. if paper space is active
- 4 Viewport specific layers can be created using the:
 - a) Layer Properties Manager dialogue box – recommended
 - b) VPLAYER command – not considered in this exercise
- 5 The most commonly used state with viewports specific layers is ‘Freeze in current viewport’

Assignment

Activity 17: Dimensioning a 3D model.

- 1 Recall the 3D faced hexagonal prism from activity 10
- 2 Make three new dimension layers using the same procedure as the exercise in this chapter, e.g. DIMTL etc
- 3 Freeze in current viewports layers which have not to be ‘displayed’ using the same method as the exercise.
- 4 Making the appropriate viewport active:
 - a) restore the required UCS
 - b) make the correct DIM layer current
 - c) add the dimensions as shown – or your own
- 5 Save the completed activity with a different name.

Shading and 3D orbit

Shading and 3D orbit have been used in previous chapters without any discussion. In this chapter we will investigate both topics in greater detail.

Shading

Shading allows certain models to be displayed on the screen as a more realistic coloured image. The models which can be shaded (and rendered) are 2½D extruded models, 3D objects, 3D surface models and solid models. We will use some previously created 3D surface models to investigate the topic.

- 1 Open the ruled surface model MODR2002\ARCHES from chapter 14
- 2 Make the model tab active and restore UCS BASE. Any layer can be active
- 3 *a)* Menu bar with **View-Hide** to display the model with hidden line removal. Note the 2D icon is still displayed
b) Menu bar with **View-Regen** to restore original model
- 4 *a)* Menu bar with **View-Shade-Hidden** and the model will be displayed with hidden line removal, but note the coloured 3D icon with X-axis red, Y-axis green and Z-axis blue
b) Menu bar with **View-Regen** and model is unchanged
c) Menu bar with **View-Shade-3D Wireframe** and the model will be displayed without hidden line removal, but the coloured 3D icon is still displayed
d) Menu bar with **View-Shade-2D Wireframe** to restore the original model with the 2D icon
- 5 Using the menu bar sequence View-Shade, activate the four shade options returning the display to 3D Wireframe before activating the next option. Note the difference in the shading between the Flat and Gouraud options, evident at the arch curved 'shoulders'.

Explanation of the shade options

Activating SHADE from the View pull-down menu allows the user access to seven options. These options allow models to be displayed as shaded/wireframe images as follows:

- 1 2D Wireframe:
The model is displayed with the boundaries as lines and curves with the 'normal' 2D icon. This option is generally used to restore shaded models to their original appearance.
- 2 3D Wireframe:
Models are displayed as lines and curves for their boundaries but with a coloured 3D icon. When used, this option restores shaded models to their original appearance but retains the coloured 3D icon.

- 3 **Hidden:**
Displays models with hidden line removal and displays the coloured 3D icon. The REGEN command will not work when this option has been used, and models are restored to their original appearance with either the 2D Wireframe or 3D Wireframe options.
- 4 **Flat Shaded:**
Models are shaded between their polygon mesh faces and appear flatter and less smooth than the Gouraud shaded models. Any materials (later chapter) which have been applied are also displayed flat shaded
- 5 **Gouraud Shaded:**
Models are shaded with the edges between the polygon mesh faces smoothed. This option gives models a realistic appearance. Added materials are also Gouraud shaded.
- 6 **Flat Shaded, Edges On:**
Models are flat shaded with the wireframe showing through the shade effect
- 7 **Gouraud Shaded, Edges On:**
Models displayed with the Gouraud shading effect and the wireframe shows through
- 8 **General:**
 - a) Both the Flat and Gouraud shading options display the coloured 3D icon
 - b) The Flat and Gouraud shading use the 2D wireframe or the 3D Wireframe options to restore the model to its original appearance
 - c) The REDRAW/REGEN/REGENALL commands cannot be used if the View-Shade sequence is activated
 - d) If HIDE <R> is entered from the command line to display any model with hidden line removal, then REGEN can be used.

This completes the shading part of the chapter. Ensure that the ARCHES drawing is still displayed in the model tab for the 3D orbit exercise.

3D orbit

The 3D orbit command (introduced in AutoCAD Release 2000) allows real-time 3D rotation, the user controlling this rotation with the pointing device. The basic concept is that you are viewing the model (the target) with a camera (the user), the target remaining stationary while the camera moves around the target. The opposite effect is displayed on the screen, i.e. the models appears to 'rotate'.

The 3D orbit concept has its own terminology, the most common being the arcball and icons – Fig. 26.1.

- 1 *Arcball*: is a circle with smaller circles at the quadrants
- 2 *Icons*: which alter in appearance dependent on where the pointing device is positioned relative to the arcball
- 3 *Click and drag*: the term for holding down the left button of the mouse and moving the mouse to give rotation of the model
- 4 *Roll*: a type of rotation when the icon is outside the arcball. It is a rotation about an axis through the centre of the arcball perpendicular to the screen.

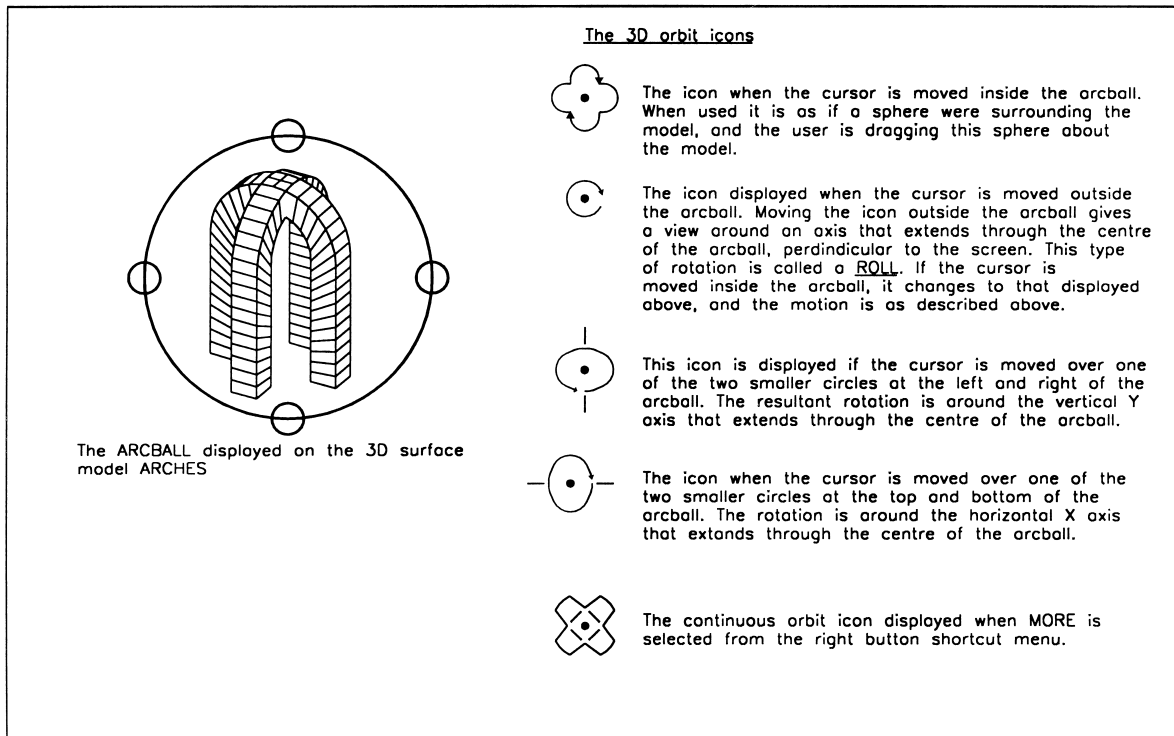


Figure 26.1 The basic 3D orbit terminology.

Using 3D orbit

- 1 The ARCHES model should still be displayed in Model tab
- 2 Pick the 3D Orbit icon from the Standard toolbar and:

prompt Arcball displayed


respond a) hold down the left mouse button

b) move the mouse about the screen

c) release the mouse button

d) practice this hold down, move, release and note the movement of the model

e) press ESC to end command


- 3 Restore the model to its original viewpoint with U <R>
- 4 Menu bar with **View-Shade-Gouraud Shaded**
- 5 At the command line enter **3DORBIT <R>** and:

prompt Arcball displayed

respond **right-click**

prompt Shortcut menu

respond **pick More**

prompt cascade shortcut menu – Fig. 26.2

respond **pick Continuous Orbit**

prompt continuous icon displayed

respond 1. move mouse slightly then leave it alone

2. model displayed shaded with continuous rotation

3. use the mouse left button to alter the rotation

4. practice the continuous rotation

5. right-click and pick Exit
- 6 Restore the model to the original orientation with U <R>

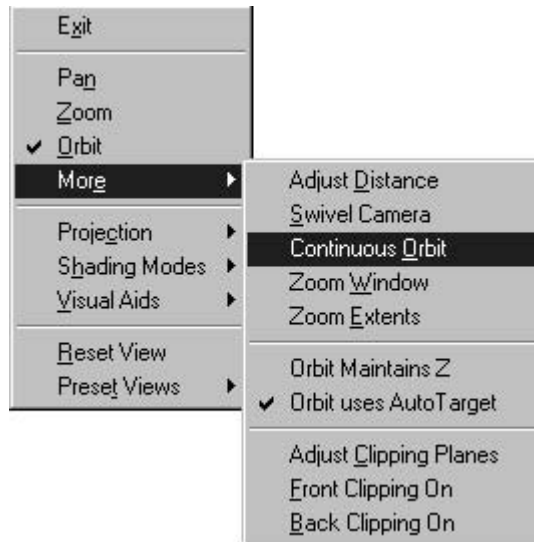


Figure 26.2 The 3D Orbit shortcut menu.

The 3D orbit shortcut menu

When the 3D orbit command is active, a right-click will display the shortcut menu which has been used to select the More and Exit options. The shortcut menu allows the user access to several useful and powerful options, the complete list being:

- a) Exit: does what it says, it exits the command
- b) Pan: the real-time AutoCAD pan command
- c) Zoom: the real-time AutoCAD zoom command, i.e. movement upwards gives magnification of model, movement downwards gives a reduction
- d) Orbit: indicates that the command is active/inactive
- e) More: allows the user access to an additional ten options which include Adjust Distance, Swivel Camera, Continuous Orbit, Clipping Planes
- f) Projection: allows parallel or perspective selections
- g) Shading Modes: allows selection of the normal AutoCAD shading
- h) Visual Aids: compass, grid and UCS icon
- i) Reset View: a useful selection as it restores the model to its original orientation prior to using the orbit command and should be used instead of the U <R> used previously
- j) Preset Views: allows the normal 3D views to be activated

Using the shortcut menu is straightforward so:

- 1 Activate the 3D orbit command and start rotating the model
- 2 Press the right button
- 3 Select the required option, e.g. Shading Modes-Flat Shaded
- 4 Display restored and the 3D rotation can continue

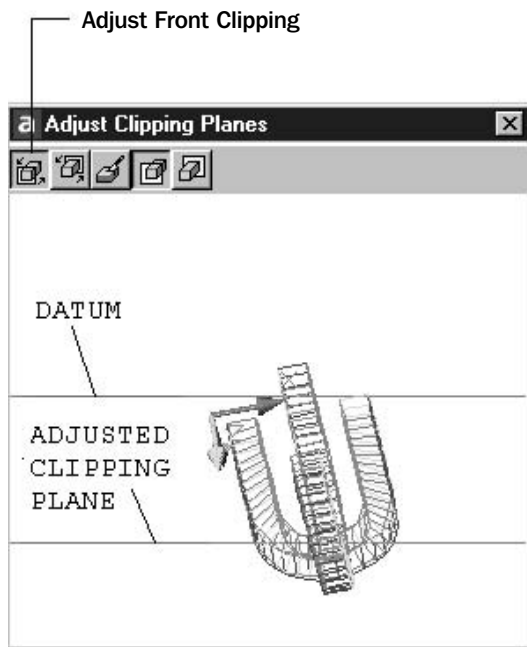
The visual aids selection allows the user to display:

- a) a compass: a sphere is displayed inside the arcball consisting of three circles representing the X, Y and Z axes. This sphere rotates with the model
- b) A grid: draws an array of lines on a plane parallel to the current X and Y axes, perpendicular to the Z axis.

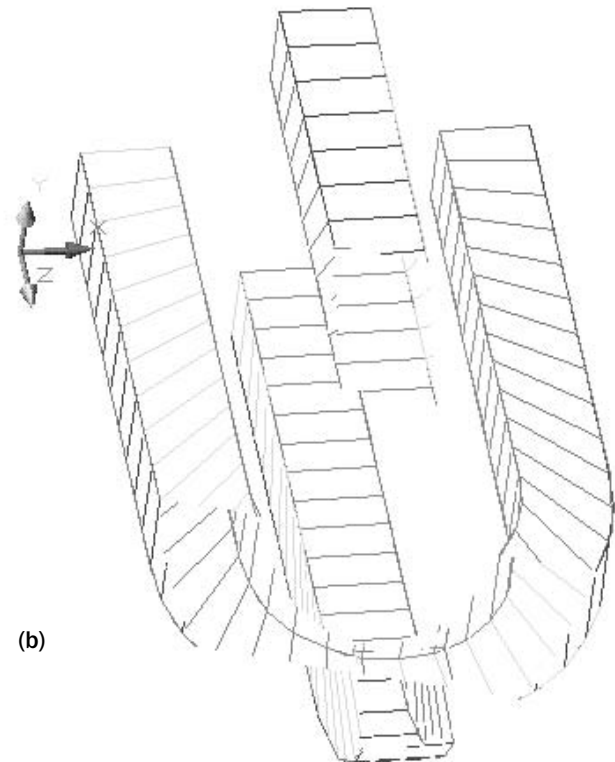
The clipping planes

The clipping plane options in 3D orbit are similar to the clip option of the DVIEW command, i.e. the clipping plane is an invisible plane set by the user. Parts of the model can be clipped, i.e. 'cut-away' relative to this clipping plane. To demonstrate the option:

- 1 Restore the model to its original orientation
- 2 Start the model rotating with the 3D orbit command
- 3 Activate the shortcut menu and select the Hidden shading mode
- 4 Activate the shortcut menu and select More-Adjust Clipping Plane and:
 - prompt* Adjust Clipping Plane dialogue box
 - respond* 1. pick the Adjust Front Clipping icon
 - 2. drag the datum plane downwards – Fig. 26.3(a)
 - 3. right-click and close
- 5 Continue with the 3D orbit and the model will be rotated with a front clip effect – Fig. 26.3(b)
- 6 When you are satisfied with the display:
 - a) right-click and reset view
 - b) right-click and exit
- 7 The model should be restored to its original orientation
- 8 Try this clipping plane option a few times. It is relatively easy to use.



(a)



(b)

Figure 26.3 Using the More-Adjust Clipping Planes option of 3D orbit.

The 3D Orbit toolbar

The toolbar for the 3D orbit command is displayed in Fig. 26.4 and allows the user access to:

- a) 3D pan and 3D zoom
- b) the 3D orbit, swivel and continuous rotations
- c) the 3D adjust distance
- d) the clipping plane options: adjust, front and back
- e) selecting the current 3D view

This completes the chapter on shading and 3D orbit.

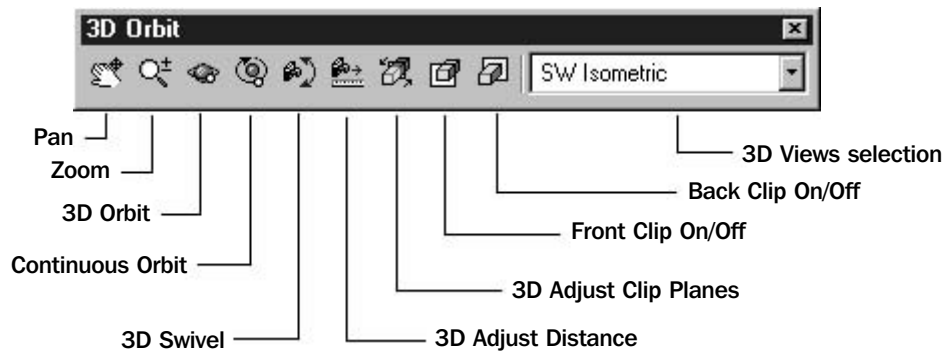


Figure 26.4 The 3D Orbit toolbar.

Summary

- 1 Surface and solid models can be shaded, the two options being Flat and Gouraud
- 2 Gouraud shaded models appear smoother than flat shaded models and this is the suggested mode for all future shading
- 3 The 3D orbit command allows interactive real-time 3D rotation of models – shaded or unshaded, with/without hidden line removal
- 4 The user has access to several useful options when the 3D orbit command is being used. These include parallel and perspective views of the model, front and back clipping and continuous 3D orbit
- 5 The user should become familiar with using the 3D orbit command as it will be used with solid modelling.

Introduction to solid modelling

Three dimensional modelling with computer-aided draughting and design (CADD) can be considered as three categories:

- wire-frame modelling
- surface modelling
- solid modelling

We have already created wire-frame and surface models and will now concentrated on how solid models are created.

This chapter will summarise the three model types.

Wire-frame modelling

- 1 Wire-frame models are defined by points and lines and are the simplest possible representation of a 3D component. They may be adequate for certain 3D model representation and require less memory than the other two 3D model types, but wire-frame models have several limitations:
 - a) *Ambiguity*: it is difficult to know how a wire-frame model is being viewed, i.e. from above or from below?
 - b) *No curved surfaces*: while curves can be added to a wire-frame model as arcs or trimmed circles, an actual curved surface cannot. Lines may be added to give a 'curved effect' but the computer does not recognise these as being part of the model.
 - c) *No interference*: as wire-frame models have no surfaces, they cannot detect interference between adjacent components. This makes them unsuitable for kinematic displays, simulations etc.
 - d) *No physical properties*: mass, volume, centre of gravity, moments of inertia, etc. cannot be calculated
 - e) *No shading*: as there are no surfaces, a wire-frame model cannot be shaded or rendered.
 - f) *No hidden line removal*: as there are no surfaces, it is not possible to display the model with hidden line removal.
- 2 AutoCAD 2002 allows wire-frame models to be created.

Surface modelling

- 1 A surface model is defined by points, lines and faces. A wire-frame model can be 'converted' into a surface model by adding these 'faces'. Surface models have several advantages when compared to wire-frame models, some of these being:
 - a) Recognition and display of curved profiles
 - b) Shading, rendering and hidden line removal are all possible, i.e. no ambiguity
 - c) Recognition of holes
- 2 Surface models are suited to many applications but they have some limitations which include:
 - a) *No physical properties*: other than surface area, a surface model does not allow the calculation of mass, volume, centre of gravity, moments of inertia, etc.
 - b) *No detail*: a surface model does not allow section detail to be obtained.
- 3 Several types of surface model can be generated including:
 - a) plane and curved swept surfaces
 - b) swept area surfaces
 - c) rotated or revolved surfaces
 - d) splined curve surfaces
 - e) nets or meshes
- 4 AutoCAD 2002 allows surface models of these types to be created.

Solid modelling

- 1 A solid model is defined by the volume the component occupies and is thus a real 3D representation of the component. Solid modelling has many advantages which include:
 - a) Complete physical properties of mass, volume, centre of gravity, moments of inertia, etc.
 - b) Dynamic properties of momentum, angular momentum, radius of gyration, etc.
 - c) Material properties of stress-strain
 - d) Full shading, rendering and hidden detail removal
 - e) Section views and profile extraction
 - f) Interference between adjacent components can be highlighted
 - g) Simulation for kinematics, robotics, etc.
- 2 Solid models are created using a **solid modeller** and there are several types of solid modeller, the two most common being:
 - a) Constructive solid geometry or constructive representation, i.e. CSG/CREP. The model is created from solid primitives and/or swept surfaces using Boolean operations
 - b) Boundary representation (BREP). The model is represented by the edges and faces making up the surface, i.e. the topology of the component.
- 3 AutoCAD 2002 supports solid models of the CSG/CREP type.
- 4 The AutoCAD 2002 modeller is based on the **ACIS** solid modeller and supports **NURBS** – *nonuniform rational B splined curves*

Comparison of the model types

The three model types are displayed in:

a) Figure 27.1: as models with hidden line removal

b) Figure 27.2: as model cross-sections

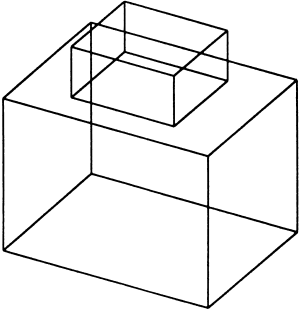
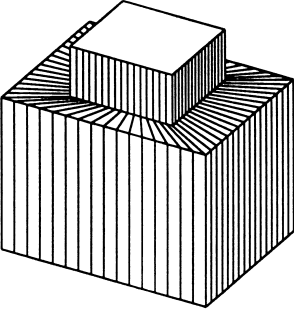
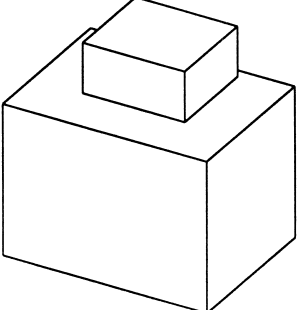
3D WIRE-FRAME MODEL	3D SURFACE MODEL	3D SOLID MODEL
 <ol style="list-style-type: none"> 1. Model has length, width and height 2. There are no surfaces on the model. 3. The model does not have area, volume or mass 4. HIDE has no effect on the appearance of the model 5. The model displays AMBIGUITY ie it is difficult to know if the model is being viewed from above or from below. 	 <ol style="list-style-type: none"> 1. Model has length, width and height 2. Surfaces have been added 3. The model has a surface area but no volume or mass 4. The HIDE command displays the model with hidden line removal 5. There is no ambiguity 6. No details can be extracted from the model. 7. The model can be shaded and rendered. 	 <ol style="list-style-type: none"> 1. The model has length, width and height 2. The model has a surface area, a volume and a mass 3. The HIDE command will display the model with hidden line removal and there is no ambiguity 4. The model has mass properties eg centroid, moment of inertia, radius of gyration etc 5. Details can be extracted from the model eg sections 6. The model can be shaded and rendered

Figure 27.1 Simple comparison between wire-frame, surface and solid models.

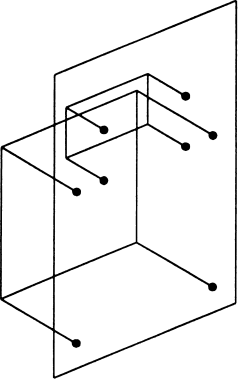
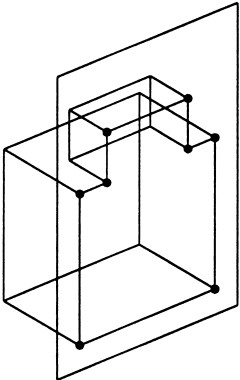
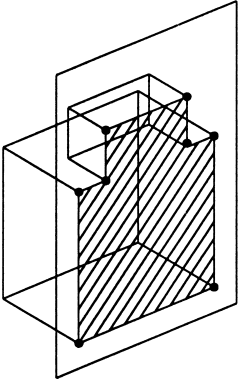
3D WIRE-FRAME MODEL	3D SURFACE MODEL	3D SOLID MODEL
 <p>The model is represented as a series of corner points and edge lines. The cross-section of the model is a series of points which are not connected. The model is useful for general shapes, appearance and position only.</p>	 <p>The model is represented as a series of corner points, edge lines and face surfaces. The cross-section of the model is a series of points and connected face lines. The model is useful for external surface display and can be displayed with hidden line removal and shading. Materials can be applied to the surfaces for rendering.</p>	 <p>The model is represented as a series of corner points, edge lines, face surfaces and interior volume. The cross-section of the model is a series of points, face lines and section planes. The model is useful for mass properties, dynamic properties and material properties with hidden line removal and shading. Materials can be applied for rendering.</p>

Figure 27.2 Further comparison of model types as cross-sections.

The solid model standard sheet

A solid model standard sheet (prototype drawing) will be created as a template and drawing file using the layouts from the surface model exercises, i.e. MV3DSTD. This standard sheet will:

- a) be for A3 paper
- b) have the four tab configurations: Model, Layout1, Layout2 and Layout3

- 1 Close and existing drawings or start AutoCAD
- 2 Menu bar with **File-New** and open your MV3DSTD template or drawing file
- 3 Check the following:
 - a) Tool-Named UCS: BASE, FRONT, RIGHT – set and saved
 - b) Layers: 0,DIM,MODEL,OBJECTS,SECT,SHEET,TEXT,VP
 - c) Sheet: layout to your own specification
 - d) Text style: ST1 (romans.shx) and ST2 (Arial Black)
 - e) Dimension style: 3DSTD with various settings
- 4 At the command line enter **-PURGE <R>** and:
 - prompt* Enter type of unused objects to purge
 - enter* **LA <R>** – layer option
 - prompt* **Enter names to purge<*>** and **<RETURN>**
 - prompt* Verify each name to be purges [Yes/No] and enter: **Y <R>**
 - prompt* Purge layer “DIM” and enter: **N <R>**
 - prompt* Purge layer “OBJECTS” and enter: **Y <R>**
 - prompt* Purge layer “SECT” and enter: **N <R>**
 - prompt* Purge layer “TEXT” and enter: **N <R>**
- 5 Repeat the command line -PURGE command with the following entries:
 - a) B (blocks) and purge all (if any) blocks
 - b) D (dimstyles) and purge all dimension styles except 3DSTD
 - c) ST (text styles) and purge any text styles except ST1 and ST2
 - d) SH (shapes) and purge all (if any) shapes
 - e) M (multilines) and purge all (if any) multilines
 - f) Note: entering PURGE at the command line will allow the user access to the Purge dialogue box. The various unwanted items can then be purged from the standard sheet
- 6 At the command line enter **ISOLINES <R>** and:
 - prompt* Enter new value for ISOLINES<4>
 - enter* **12 <R>**
- 6 At the command line enter **FACETRES <R>** and:
 - prompt* Enter new value for FACETRES<0.5000>
 - enter* **1 <R>**
- 7 Display the Draw, Modify, Object Snap and Solids toolbars
- 8 Make MVLAY1 tab current, model space with the lower left viewport active, restore UCS BASE with layer MODEL current

9 Menu bar with **File-Save As** and:

- prompt* Save Drawing As dialogue box
respond 1. scroll at Files of type
 2. pick **AutoCAD Drawing Template File (*.dwt)**
- prompt* Save in AutoCAD Template folder dialogue box
respond 1. enter file name as: **A3SOL.dwt**
 2. pick **Save**
- prompt* Template Description dialogue box
respond 1. enter: **My solid model prototype created on ???**
 2. measurement: Metric
- then* pick OK

10 Menu bar with **File-Save As** and:

- prompt* Save Drawing As dialogue box
respond 1. scroll at Files of type
 2. pick **AutoCAD 2000 Drawing (*.dwg)**
 3. scroll and pick your named folder
 4. enter file name: **A3SOL.dwg**
 5. pick **Save**

11 Menu bar with **File-Save As** and save you standard sheet as an AutoCAD Drawing Template file (A3SOL) in your named folder

12 You are now ready to start creating solid models.

Notes

- 1 The new A3SOL solid model standard sheet has been saved as both a template file and a drawing file
- 2 The A3SOL drawing file has been saved to your named folder, while the template file has been saved to both the AutoCAD Template folder and your named folder.
- 3 Two new system variables have been introduced in the creation of the A3SOL template/drawing file, these being:
 - ISOLINES*: specifies the number of isolines per surface on objects. It is an integer with values between 0 and 2047. The default value is 4.
 - FACETRES*: adjusts the smoothness of shaded and rendered objects and objects with hidden line. The value can be between 0.01 and 10.00, the default being 0.5
- 4 The *ISOLINES* and *FACETRES* values may be altered when creating some models
- 5 The viewports have not been zoom-centred, as this will depend on the model being created.
- 6 Solid modelling consists of creating 'composites' from 'primitives' and AutoCAD 2002 supports the following types of primitive:
 - a) basic b) swept c) edge
- 7 All three types of primitive will be investigated with examples and the various options for each will be discussed.
- 8 While the A3SOL standard sheet has different layout configurations, I will generally only use the *MVLAY1* or the *MODEL* tabs the, although the other layout tabs will be 'investigated' from time to time

Solid modelling is a fascinating topic, and should give the user a great deal of satisfaction as the various models are created and rendered.

The basic solid primitives

AutoCAD 2002 supports the six basic solid primitives of box, wedge, cylinder, cone, sphere and torus. In this chapter we will create layouts (which I hope are interesting) using each primitive. We will also investigate the various options which are available.

Note: During the exercises do not just accept the coordinate values given. Try and reason out why they are being used.

The BOX primitive – Fig. 28.1

- 1 Open your A3SOL template file with layer MODEL, tab MVLAY1, UCS BASE and the lower left viewport active. Display the Object Snap and Solids toolbars.
- 2 Select the BOX icon from the Solids toolbar and:



```

prompt Specify corner of box or [CEnter]<0,0,0>
enter 0,0,0 <R>
prompt Specify corner or [Cube/Length]
enter C <R> – the cube option
prompt Specify length
enter 100 <R>
and a red cube is displayed in all viewports

```
- 3 Select from the menu bar **Draw-Solids-Box** and:

```

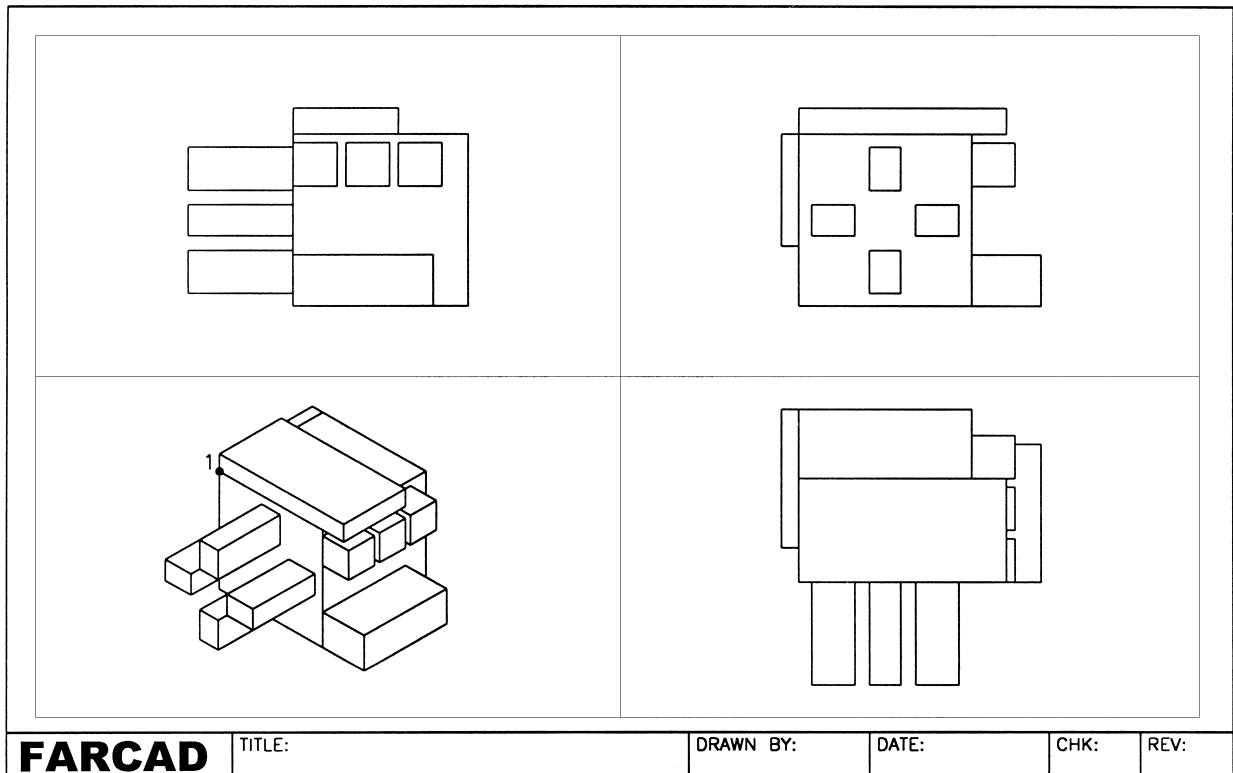
prompt Specify corner of box or [CEnter]<0,0,0>
enter 100,0,0 <R>
prompt Specify corner or [Cube/Length]
enter L <R> – the length option
prompt Specify length and enter: 40 <R>
prompt Specify width and enter: 80 <R>
prompt Specify height and enter: 30 <R>
and another red cuboid is displayed in all viewports

```
- 4 At the command line enter **CHANGE <R>** and:

```

prompt Select objects
respond pick the smaller box then right-click
prompt Specify change point or [Properties] and enter: P <R>
prompt Enter property to change and enter: C <R>
prompt Enter new color and enter: 3 <R> or green <R>
prompt Enter property to change and right-click/enter

```
- 5 Now zoom-extends the zoom at a scale of 1.5 in each viewport then make the lower left viewport again active

**FARCAD**

TITLE:

DRAWN BY:

DATE:

CHK:

REV:

Figure 28.1 The BOX primitive layout (BOXPRIM).

- 6 At the command line enter **BOX <R>** and:
 - prompt* Specify corner of box or [CEnter]<0,0,0>
 - respond* **Endpoint icon and pick pt1** – first corner point
 - prompt* Specify corner or [Cube/Length]
 - enter* **@120,60,15 <R>** – the diagonally opposite corner point
- 7 Change the colour of this box to blue, with **CHANGE <R>** at the command line as step 4
- 8 Restore UCS RIGHT
- 9 Create a solid box with the following information:
 - a) corner: 0,70,100
 - b) cube option with length: 25
- 10 a) At the command line enter **CHANGE <R>** and:
 - prompt* Select objects
 - respond* pick the last box created
 - prompt* **I was not parallel to the UCS**
 - respond* right-click to end the command
 b) At the command line enter **CHPROP <R>** and:
 - prompt* Select objects
 - respond* pick the last box created and right-click
 - prompt* Enter property to change and enter: **C <R>**
 - prompt* Enter new color and enter: **MAGENTA <R>**
 - prompt* Enter property to change and right-click/enter
 c) *Note:* the CHANGE and CHPROP commands will be discussed at the end of the chapter. In future, when an object has to have its colour changed use the CHPROP command from the command line.

- 11 Rectangular array the magenta box:
 - a) for 1 row and 3 columns
 - b) with column distance: 30
- 12 Restore UCS FRONT
- 13 Activate the solid BOX command and:

prompt Specify corner of box or [CEntre]<0,0,0>
enter **CE <R>** – the box center point option
prompt Specify center of box and enter: **50,80,30 <R>**
prompt Specify corner of box or [Cube/Length] and enter: **L <R>**
prompt Specify length and enter: **18 <R>**
prompt Specify width and enter: **25 <R>**
prompt Specify height and enter: **60 <R>**
- 14 With CHPROP, change the colour of this last box to suit yourself. I used colour number 76
- 15 Polar array this last box with:
 - a) center point: 50,50
 - b) items: 4
 - c) 360 angle with rotation
- 16 Finally restore UCS BASE and create another solid box with:
 - a) corner: 0,100,100
 - b) length: -10; width: -80; height: -65
 - c) colour: number 234
- 17 The model is now complete so save as **MODR2002\BOXPRIM**
- 18 *Investigate:*
 - a) The model with hide – menu bar or command line
 - b) Shading the viewports
 - c) The 3D orbit command with the shaded model and the model tab active
 - d) Restoring the model to the original 3D wireframe display
- 19 *Changing properties*
 When solid model layouts are being created the user will be asked to change some properties, especially the colour. All primitives will be originally displayed as red (due to layer MODEL being current) and thus require to be changed to another colour. This is to give a coloured image for shading, rendering and for using the 3D orbit command to maximum effect. Changing the colour of a primitive depends on the value of the PICKFIRST system variable and:
 - a) if PICKFIRST is set to 0, then use CHANGE <R> at the command line as step 4, i.e. activate the command then pick the object which is to have its colour changed
 - b) If PICKFIRST is set to 1 then:
 1. select the object to be changed
 2. pick the Properties icon from the Standard toolbar
 3. pick the Color line
 4. scroll and pick the required colour
 5. cancel the Properties dialogue box and press ESC
- 20 *CHANGE or CHPROP?*
 These two commands are similar but:
 - a) CHANGE can only be used for objects created with the current UCS, hence the step 10(a) message
 - b) CHPROP can be used with **ANY** UCS setting, and should be used for all future change colour operations
- 21 The user must now decide whether to set PICKFIRST to 0 or 1. My personal preference is to have PICKFIRST set to 0 and use the CHPROP command.

The WEDGE primitive – Fig. 28.2

- 1 Open the A3SOL template file with layer MODEL, MVLAY tab, UCS BASE and lower left viewport active
- 2 Menu bar with **Draw-Solids-Wedge** and:
 - prompt* Specify corner of wedge or [CEnter]<0,0,0>
 - enter* **0,0,0 <R>**
 - prompt* Specify corner or [Cube/Length]
 - enter* **C <R>** – the cube option
 - prompt* Specify length
 - enter* **100 <R>**
 - and* red wedge displayed in all viewports
- 3 Select the WEDGE icon from the Solids toolbar and:
 - prompt* Specify corner of wedge or [CEnter]
 - enter* **0,0,0 <R>**
 - prompt* Specify corner or [Cube/Length]
 - enter* **@80,-60 <R>** – the other corner option
 - prompt* Specify height and enter: **50 <R>**
- 4 Change the colour of this wedge to blue
- 5 In each viewport, zoom-extents then zoom to a 1.75 scale then make the lower left viewport again active

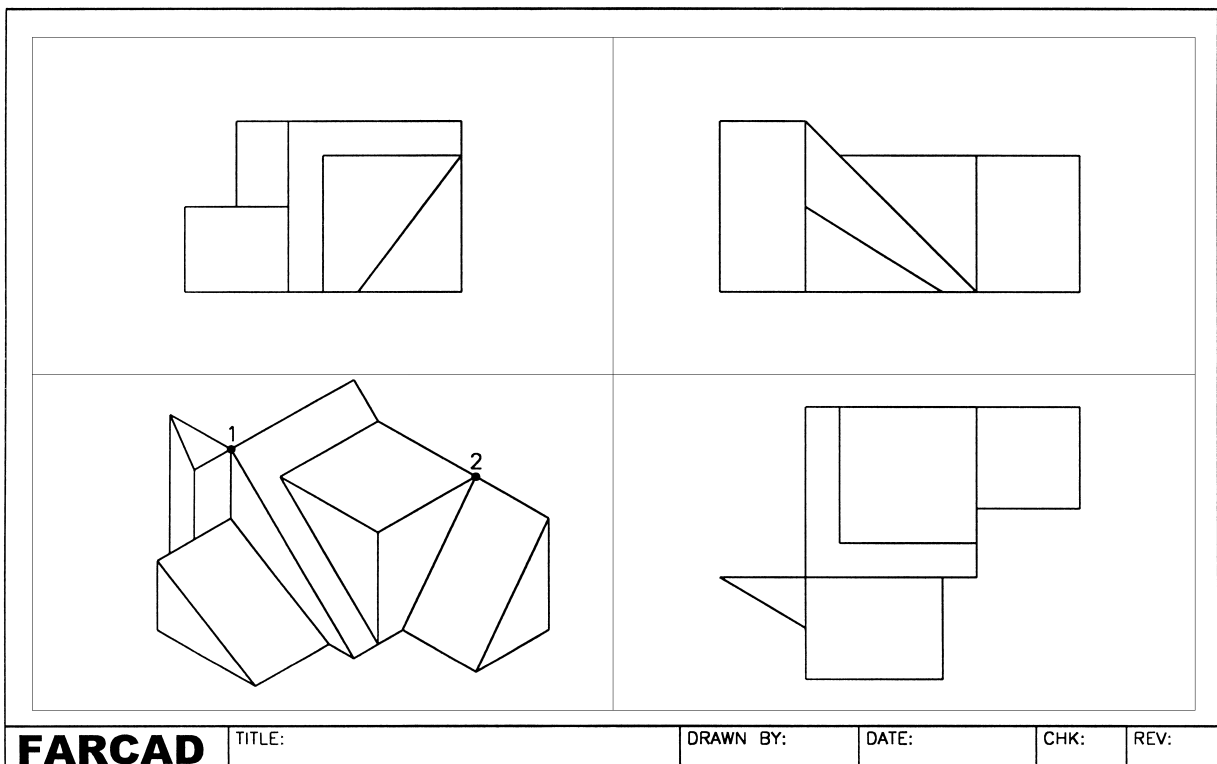


Figure 28.2 The WEDGE primitive layout (WEDPRIM).

- 6 At the command line enter **WEDGE <R>** and create a solid wedge with:
 - a) corner: 100,100,0
 - b) length: 60; width: 60; height: 80
 - c) colour: green
 - d) 2D rotate the green wedge about the point 100,100 by **-90degs**
- 7 Restore UCS FRONT and create a wedge with:
 - a) corner: **endpoint icon and pick pt1**
 - b) length: -50
 - c) width: -100
 - d) height: 30
 - e) colour: magenta (CHPROP command)
- 8 Restore UCS BASE
- 9 The final wedge is to be created with:
 - a) corner: **endpoint of pt2**
 - b) cube option with length: -80
 - c) colour: number 14
- 10 Save the model layout as **MODR2002\WEDPRIM**
- 11 Hide and shade the model in each viewport
- 12 Use 3D orbit with a shaded model in the MODEL tab.

The CYLINDER primitive – Fig. 28.3


- 1 Open A3SOL, UCS BASE, MVLAY1 tab, layer MODEL, lower left viewport active
- 2 Menu bar with **Draw-Solids-Cylinder** and:


```

prompt Specify center point for base of cylinder or [Elliptical]
enter 0,0,0 <R> – the cylinder base centre point
prompt Specify radius for base of cylinder or [Diameter]
enter 60 <R> – the cylinder radius
prompt Specify height of cylinder or [Center of other end]
enter 100 <R> – the cylinder height
and a red cylinder is displayed, centred on 0,0,0
      
```
- 3 Select the CYLINDER icon from the Solids toolbar and:


```

prompt Specify center point for base of cylinder or [Elliptical]
enter E <R> – the elliptical option
prompt Specify axis endpoint of ellipse for base of cylinder or
[Center]
enter C <R> – the centre option
prompt Specify center point of ellipse for base of cylinder
enter 80,0,0 <R>
prompt Specify axis endpoint of ellipse for base of cylinder
enter @20,0,0 <R>
prompt Specify length of other axis for base of cylinder
enter @0,30,0 <R>
prompt Specify height of cylinder or [Center of other end]
enter 50 <R>
      
```


- 4 Change the colour of this cylinder to green, then polar array it:
 - a) about the point 0,0
 - b) for 3 items, full angle with rotation
- 5 Zoom centre about the point 0,0,80 at 300 magnification in all viewports

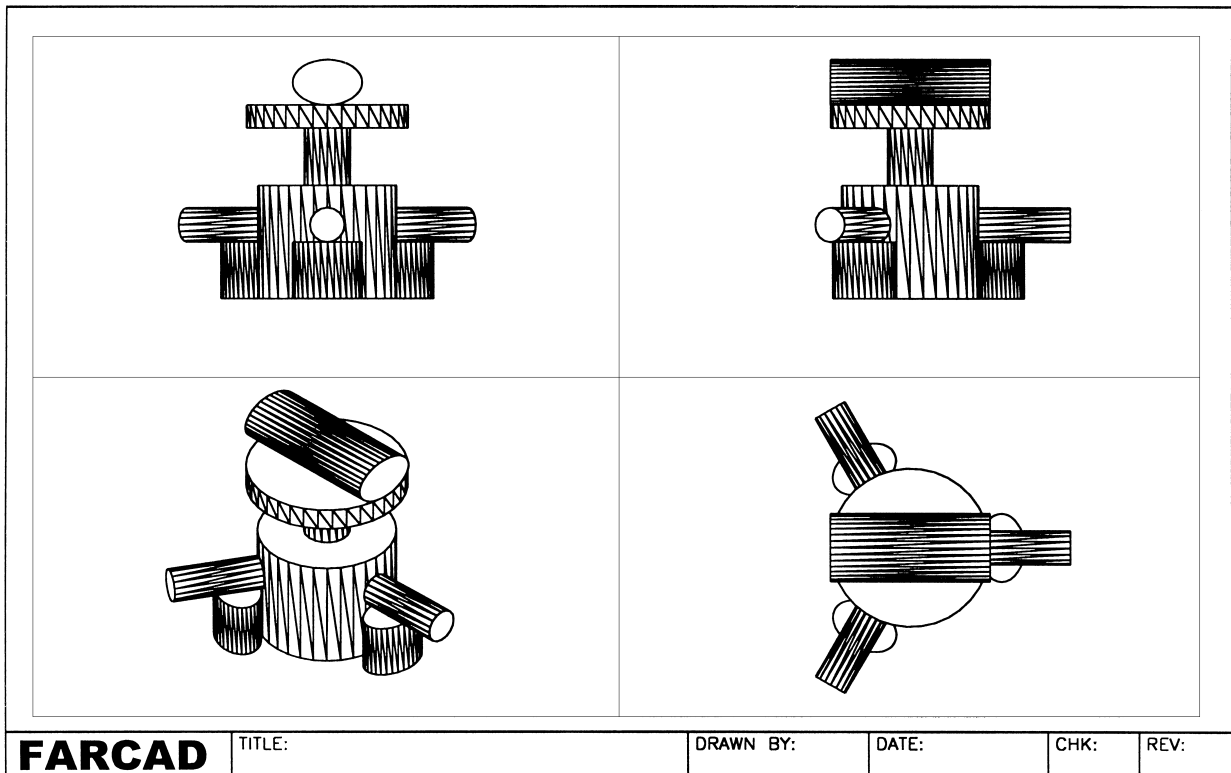


Figure 28.3 The CYLINDER primitive layout (CYLPRIM).

- 6 At the command line enter **CYLINDER <R>** and:

prompt Specify center point for base of cylinder or [Elliptical]

enter **60,0,65 <R>** – the cylinder base centre point

prompt Specify radius for base of cylinder and enter: **15 <R>**

prompt Specify height of cylinder or [Center of other end]

enter **C <R>** – centre of other end option

prompt Specify center of other end of cylinder

enter **@80,0,0 <R>**

- 7 Change the colour of this cylinder to magenta and polar array it with the same entries as step 4

- 8 Create another two cylinders:

<i>centre pt</i>	<i>rad</i>	<i>ht</i>	<i>colour</i>
0,0,100	20	50	number 15
0,0,150	70	20	blue

- 9 Finally create an **elliptical** cylinder:

a) centre: 70,0,190

b) axis endpoint: @30,0

c) length of other axis: @0,20

d) centre of other end: @-140,0,0

e) colour to suit

- 10 Save the model as **MODR2002\CYLPRIM**

- 11 Hide the model and note the triangular facets which are not displayed when the cylinder primitives are created. Shade then return the model to wireframe representation

12 Investigate

- a) With the 3D viewport active, enter ISOLINES at the command line and enter a value of 6. Enter REGEN <R> and note model display
- b) change the ISOLINES value to 48 and regen
- c) return the ISOLINES value to the original 12
- d) enter FACETRES at the command line and alter the value to 2, then enter HIDE <R>. Note the effect, then regen
- e) alter FACETRES to 5, hide, then regen
- f) return FACETRES to 1

13 Note

The ISOLINES system variable controls the appearance of primitive curved surfaces when they are created. The triangulation effect, or *FACETS*, is controlled by the system variable FACETRES. The higher the value of FACETRES (max 10) then the 'better the appearance' of curved surfaces, but the longer it takes for hide and shade. At our level, the values of 12 for ISOLINES and 1 for FACETRES are sufficient.

The CONE primitive – Fig. 28.4

1 Open the A3SOL template file, MVLAY1, UCS BASE etc

2 Menu bar with **Draw-Solids-Cone** and:

```
prompt Specify center point for base of cone or [Elliptical]
enter 0,0,0 <R> – the cone base centre point
prompt Specify radius for base of cone or [Diameter]
enter 50<R>
prompt Specify height of cone or [Apex]
enter 60 <R>
```

3 Create another cone with:

- a) centre: 0,0,0
- b) radius: 90
- c) height: -80
- d) colour: green

4 Centre each viewport about the point 0,0,10 at 200 mag

5 Select the CONE icon from the Solids toolbar and:

```
prompt Specify center point for base of cone or [Elliptical]
enter E <R> – the elliptical option
prompt Specify axis endpoint of ellipse for base of cone or
[Center]
enter C <R> – the centre point option
prompt Specify center point of ellipse for base of cone
enter 70,0,0 <R>
prompt Specify axis endpoint of ellipse for base of cone and enter:
@20,0,0 <R>
prompt Specify length of other axis for base of cone and enter:
@0,25,0 <R>
prompt Specify height of cone or [Apex] and enter: 35 <R>
```



6 Change the colour of this cone to blue, then polar array it with:

- a) centre point: 0,0
- b) items: 5
- c) full angle to fill, with rotation

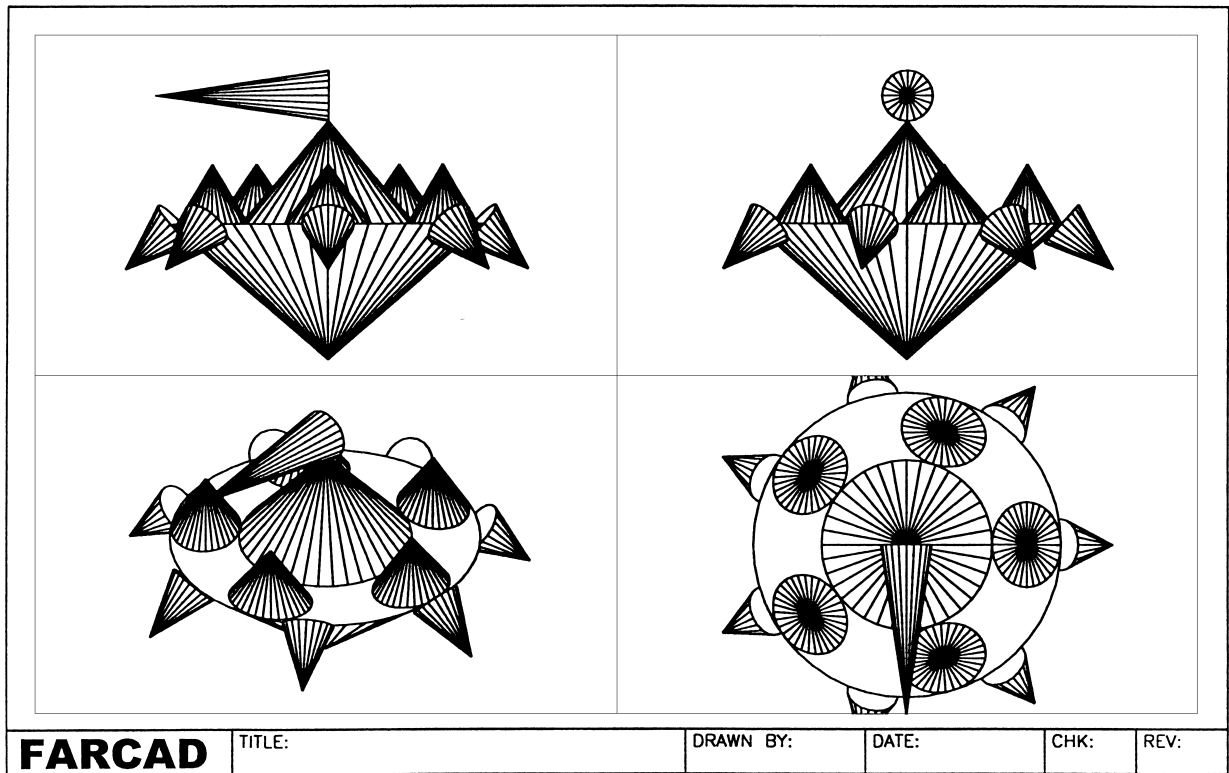


Figure 28.4 The CONE primitive layout (CONPRIM).

- 7 At the command line enter **CONE <R>** and:
 - prompt* Specify center point for base of cone and enter: **90,0,0 <R>**
 - prompt* Specify radius for base of cone and enter: **15 <R>**
 - prompt* Specify height of cone or [Apex]
 - enter* **A <R>** – the apex option
 - prompt* Specify apex point
 - enter* **@40,0,0 <R>**
- 8 Change the colour of this last cone to colour number 235
- 9 Use the ROTATE 3D command with the last cone and:
 - a) enter a Y axis rotation
 - b) enter 90,0,0 as the point on the Y axis
 - c) enter 41.63 as the rotation angle – *why this figure?*
- 10 Polar array the last cone about the point 0,0 for 7 items, full angle with rotation.
- 11 Create the final cone with:
 - a) centre: 0,0,75
 - b) radius: 15
 - c) apex option: @0,-100,0
 - d) colour to suit
- 12 Hide, shade, 3D orbit then save as **MODR2002\CONPRIM**.

The SPHERE primitive – Fig. 28.5

- 1 Open the A3SOL template file, UCS BASE etc
- 2 Menu bar with **Draw-Solids-Sphere** and:
 - prompt* Specify center of sphere<0,0,0>
 - enter* **0,0,0 <R>**
 - prompt* Specify radius of sphere or [Diameter]
 - enter* **60 <R>**
- 3 Centre each viewport about 0,0,20 at 200 magnification
- 4 Select the SPHERE icon from the Solids toolbar and:
 - prompt* Specify center of sphere<0,0,0>
 - enter* **80,0,0 <R>**
 - prompt* Specify radius of sphere or [Diameter]
 - enter* **D <R>** – the diameter option
 - prompt* Specify diameter and enter: **40 <R>**
- 5 Change the colour of this sphere to green, then polar array it:
 - a) about the point: 0,0
 - b) for 5 items
 - c) full angle with rotation
- 6 At the command line enter **SPHERE <R>** and create a sphere with:
 - a) centre: 0,0,75
 - b) radius: 15
 - c) colour: magenta

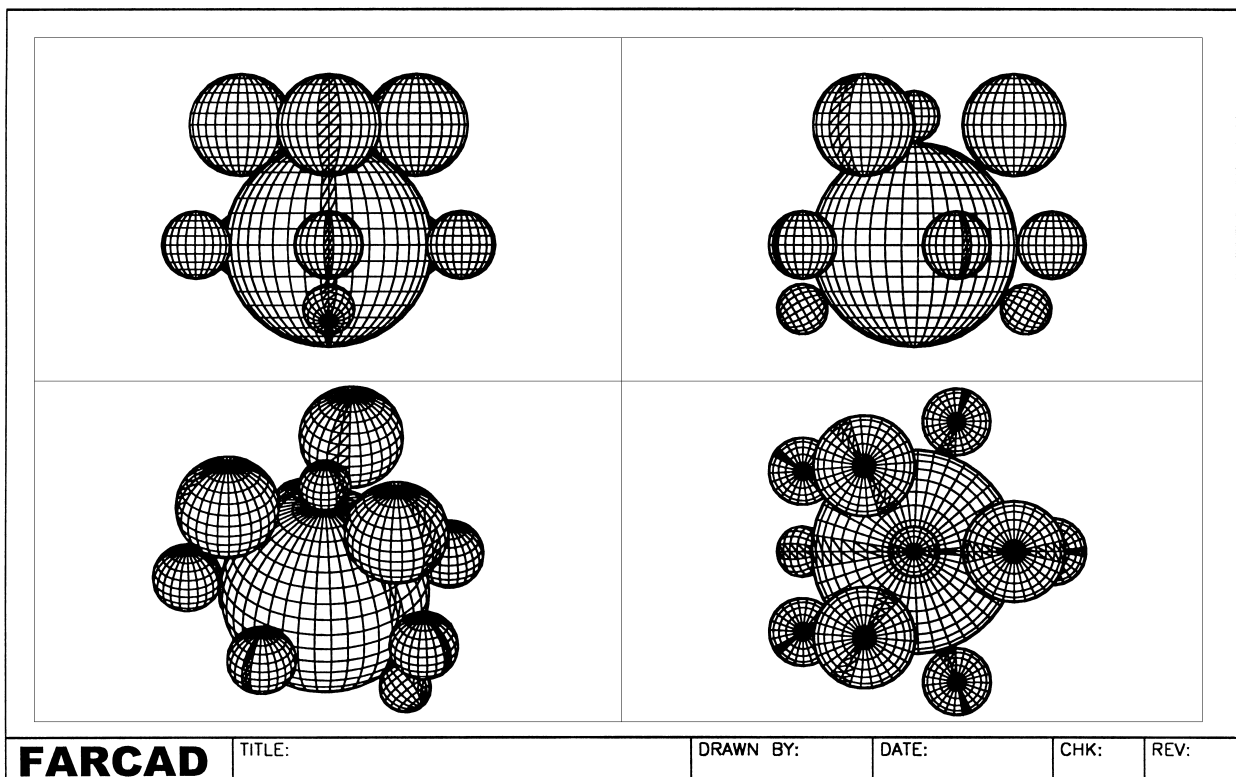


Figure 28.5 The SPHERE primitive layout (SPHPRIM).

- 7 Restore UCS FRONT and polar array the magenta sphere about the point 0,0 for 3 items with full angle rotation
- 8 Restore UCS BASE and create the final sphere with:
 - a) centre: 58,0,70
 - b) radius: 30
 - c) colour blue
 - d) polar array about 0,0 for 3 items, full angle rotation
- 9 Hide, shade, save as **MODR2002\SPHPRIM**
- 10 Investigate
 - a) ISOLINES set to 48 then regen
 - b) ISOLINES set to 5 then regen
 - c) Restore original ISOLINES 18
 - d) FACETRES set to 10 then hide and regen
 - e) Note the appearance of the sphere primitives with these system variable values.

The TORUS primitive – Fig. 28.6

- 1 A3SOL template file with UCS BASE, MVLAY1 layout etc
- 2 Menu bar with **Draw-Solids-Torus** and:


```
prompt Specify center of torus<0,0,0>
enter 0,0,0 <R>
prompt Specify radius of torus or [Diameter]
enter 80 <R>
prompt Specify radius of tube or [Diameter]
enter 15 <R>
```
- 3 Zoom extents then zoom to a scale of 1 in each viewport
- 4 Restore UCS FRONT and select the TORUS icon from the Solids toolbar and:


```
prompt Specify center of torus<0,0,0>
enter 80,0,0 <R>
prompt Specify radius of torus and enter: 50 <R>
prompt Specify radius of tube and enter: 20 <R>
```
- 5 Change the colour of this torus to blue
- 6 Restore UCS BASE and polar array the blue torus about 0,0 for 3 items with full circle rotation
- 7 Restore UCS RIGHT, enter **TORUS <R>** at the command line and create a torus with:
 - a) centre: 0,0,-95
 - b) radius of torus: 80
 - c) radius of tube: 20
 - d) colour: green
- 8 Restore UCS BASE and create the final torus with:
 - a) centre: 80,0,80
 - b) radius of torus: 50
 - c) radius of tube: 20
 - d) colour: to suit
- 9 Save as **MODR2002\TORPRIM**

- 10 a) With the lower left viewport active, HIDE and note the effect then regen
 - b) Enter **DISPSILH <R>** at the command line and:
prompt Enter new value for DISPSILH <0>
enter **1 <R>**
 - c) Hide the model, note the effect and compare this to that obtained in step (a)
 - d) Now restore DISPSILH back to 0
- 11 *Note:* DISPSILH is a system variable which controls the display of silhouette curves of solid objects in wire-frame mode and:
DISPSILH 0 : default value, models displayed 'as normal'
DISPSILH 1 : models displayed with silhouette effect
 - 12 It is your decision as to whether to have DISPSILH set to 0 or 1. I always have it set to 0.

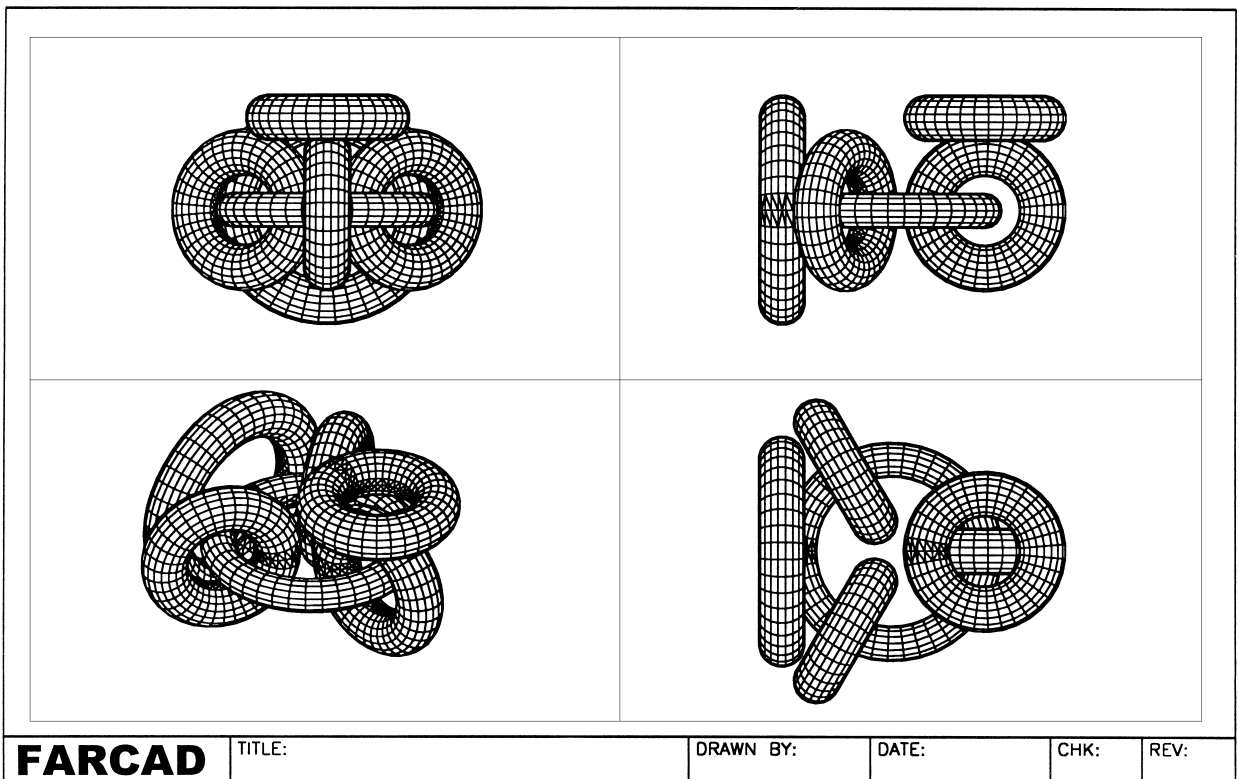


Figure 28.6 The TORUS primitive layout (TORPRIM).

Summary

- 1 The six solid primitives can be activated:
 - a) from the menu bar with Draw-Solids
 - b) by icon selection from the Solids toolbar
 - c) by entering the solid name at the command line
- 2 The corner/centre start points can be:
 - a) entered as coordinates from the keyboard
 - b) referenced to existing objects
 - c) picking suitable points on the screen
- 3 The six primitives have various options:
 - box:
 - a) corner; centre
 - b) cube; length, width, height; other corner
 - wedge:
 - a) corner; centre
 - b) cube; length, width, height; other corner
 - cylinder:
 - a) circular; elliptical
 - b) radius; diameter
 - c) height; centre of other end
 - cone:
 - a) circular; elliptical
 - b) radius; diameter
 - c) height; apex point
 - sphere:
 - a) centre only
 - b) diameter; radius
 - torus:
 - a) centre only
 - b) radius of torus; diameter
 - c) radius of tube; diameter

Assignment

A single activity for your imaginative mind.

Activity 18: A solid primitive layout.

Create a layout **of your own design** with the six basic primitives using the following information and with your own colours:

<i>Box</i>	<i>Wedge</i>	<i>Cylinder</i>	<i>Cone</i>
length: 100	length: 50	radius: 20	radius: 50
width: 40	width: 60	height: 80	height: 120
height: 30	height: 50		
<i>Sphere</i>	<i>Torus</i>		
radius: 40	torus rad: 50		
	tube rad: 20		

The swept solid primitives

Solid models can be generated by extruding or revolving ‘shapes’ and in this chapter we will use several exercises to demonstrate how complex solids can be created from relatively simple shapes.

Extruded solids

Solid models can be created by extruding **CLOSED OBJECTS** such as polyline shapes, polygons, circles, ellipses, splines and regions:

- a) to a specified height and taper angle
- b) along a path.

Extruded example 1: letters

- 1 Open your A3SOL template file, layer MODEL, MVLAY1 and display the Solids toolbar
- 2 Change the viewpoint in lower left viewport with the command line entry **VPOINT <R>** and:
prompt Specify a view point or [Rotate]
enter **R <R>** – the rotate option
then enter angles of 30 and 30
- 3 Make the upper left viewport active and restore UCS RIGHT
- 4 Using the reference sizes given in Fig. 29.1:
 - a) draw the three letters M, T and C as closed shapes using line and arc segments
 - b) use **Modify-Object-Polyline** from the menu bar to ‘convert’ the three outlines into a single polyline with the Join option
 - c) use your discretion for sizes not given (a snap of 5 helps)
 - d) use the start points A, B and C given
- 5 Still with UCS RIGHT, zoom centre about 0,50,0 at 200 in the 3D viewport and 300 in the other three viewports
- 6 Menu bar with **Draw-Solids-Extrude** and:
prompt Select objects
respond **pick the letter M then right-click**
prompt Specify height of extrusion or [Path]
enter **80 <R>**
prompt Specify angle of taper for extrusion
enter **0 <R>**
- 7 The letter M will be extruded for a height of 80 in the positive Z direction.

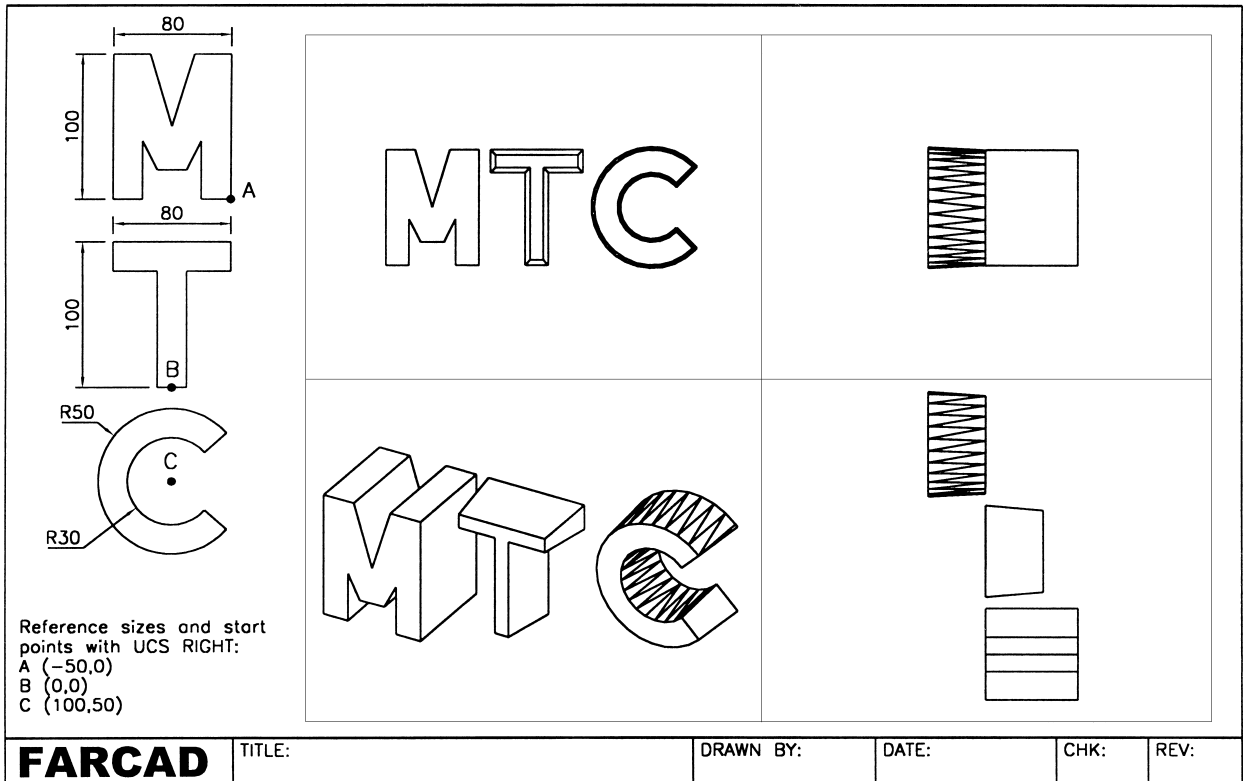


Figure 29.1 Extruded example 1 – letters.

8 Select the EXTRUDE icon from the Solids toolbar and:

prompt Select objects

respond **pick the letter T then right-click**

prompt Specify height of extrusion or [Path]

enter **50 <R>**

prompt Specify angle of taper for extrusion

enter **5 <R>**



9 At the command line enter **EXTRUDE <R>** and:

a) objects: pick the letter C then right-click

b) height: enter **-50**

c) taper angle: enter **-3**

10 Hide and shade etc, then save the model with your own name

Extruded example 2: keyed splined shaft

- 1 Open the A3SOL template file, layer MODEL, MVLAY1, UCS BASE and the lower right viewport active.
Note: in future exercises when I refer to opening A3SOL these are the 'normal settings' I expect, i.e. layer MODEL current, MVLAY1 tab active, UCS BASE current and the lower left (3D) viewport active. I will thus only say: Open A3SOL file.
- 2 Zoom centre about 0,0,-20 at 150 magnification in all viewports
- 3 Refer to Fig. 29.2 and create two profiles:
 - a) an outer tooth profile from two circles and an arrayed line, then trim as required. The circle centres should be at 0,0
 - b) an inner shaft profile to your own specification
- 4 Use the menu bar sequence Modify-Object-Polyline (or PEDIT at the command line) to convert each profile into a single polyline using the Join option.
- 5 Alter the system variables ISOLINES to 6 and FACETRES to 0.5
- 6 Select the EXTRUDE icon from the Solids toolbar and:

prompt Select objects
respond **pick the outer tooth profile then right-click**
prompt Specify height of extrusion or [Path]
enter **-70 <R>**
prompt Specify angle of taper for extrusion
enter **-3 <R>**
- 7 Repeat the Extrude icon selection and:
 - a) objects: pick the inner shaft profile then right-click
 - b) height: enter 30
 - c) taper: enter 0
 - d) note the 'denseness' of the model sides
- 8 Gouraud shade the model and use 3D orbit in the 3D viewport
- 9 Restore the original display with the menu bar sequence **View-Shade-2D Wireframe**
- 10 Save the model layout

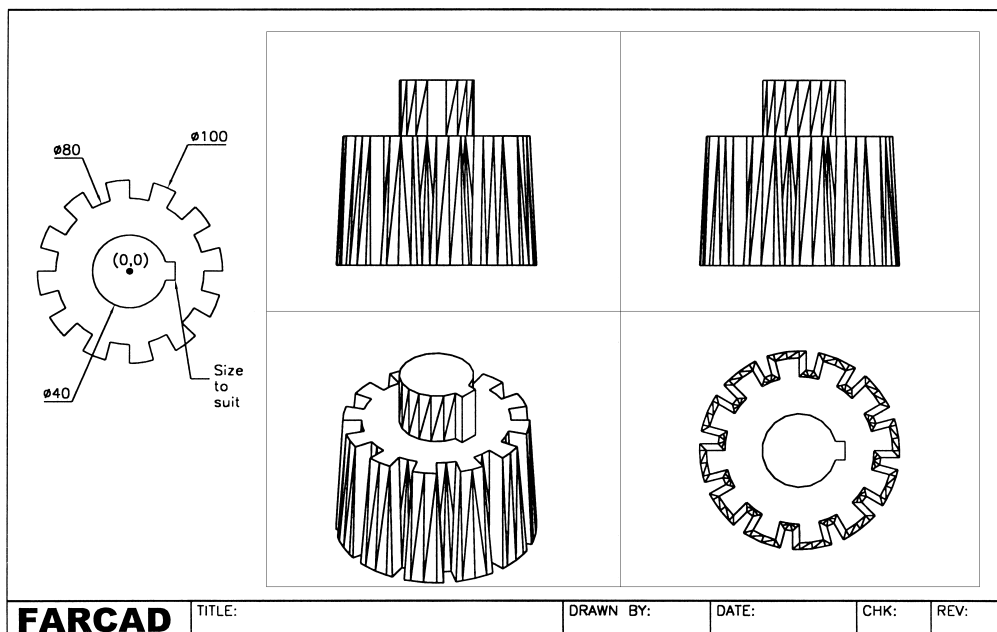


Figure 29.2 Extruded example 2 – splined shaft.

Extruded example 3: a moulding

- 1 Open the A3SOL template file with 'normal settings'
- 2 Centre the viewports about the point 120,75,35 at 300 mag
- 3 Draw a polyline with:
 - start point: 0,0
 - next point: @0,100
 - arc option with endpoint: @100,0
 - arc endpoint: @100,-100
 - line option with endpoint: @100,0 then right-click/enter
- 4 Change the colour of the polyline to blue
- 5 Restore UCS FRONT and make the top right viewport active
- 6 Use the reference sizes in Fig. 29.3 to create the moulding as a single polyline or create your own outline design if required.
- 7 Make the lower left viewport active
- 8 Set ISOLINES to 12
- 9 Select the EXTRUDE icon from the Solids toolbar and:
 - prompt* Select objects
 - respond* **pick the red polyline then right-click**
 - prompt* Specify height of extrusion or [Path]
 - enter* **P <R>** – the path option
 - prompt* Select extrusion path
 - respond* **pick the blue polyline**
 - and* the red outline is extruded along the blue path
- 10 Shade the model, 3D orbit then restore to original wireframe model
- 11 Save the model layout

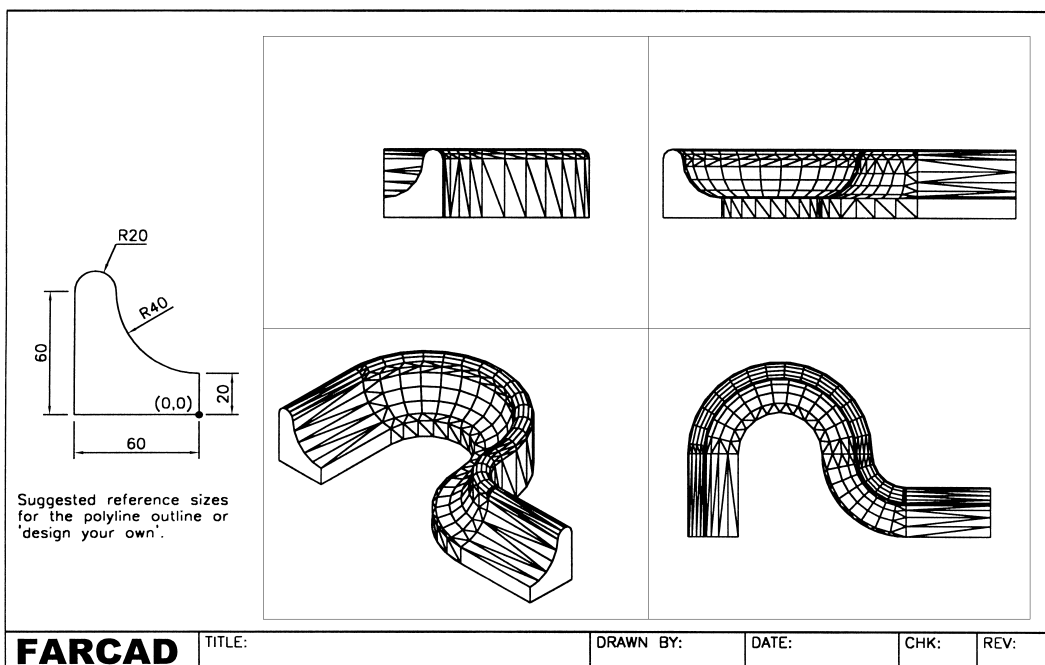


Figure 29.3 Extruded example 3 – a moulding.

Extruded example 4: a polygon type duct arrangement

- 1 Open the A3SOL template file with normal settings
- 2 Create a 5 segment 3D polyline with the menu bar sequence **Draw-3D Polyline** and:
 - prompt* Specify start point of polyline and enter: **0,0 <R>**
 - prompt* Specify endpoint of line and enter: **@0,0,100 <R>**
 - prompt* Specify endpoint of line and enter: **@100,0,100 <R>**
 - prompt* Specify endpoint of line and enter: **@100,100,0 <R>**
 - prompt* Specify endpoint of line and enter: **@0,100,100 <R>**
 - prompt* Specify endpoint of line and enter: **@100,100,-100 <R>**
 - prompt* Specify endpoint of line and: **right-click/enter**
- 3 Change the colour of the 3D polyline to green
- 4 In each viewport, zoom centre about 150,150,175 at 400 mag.
- 5 Draw a polygon with:
 - a) sides: 6
 - b) centre: 0,0
 - c) inscribed in a circle of radius 40
- 6 Select the EXTRUDE icon and:
 - a) objects: pick the red polygon then right-click
 - b) enter P <R> for the path option
 - c) path: pick the green 3D polyline
- 7 The polygon will be extruded along the green path to give a ducting effect.
- 8 Shade, 3D orbit then save
- 9 *Question:* Green polyline but red model – why?
This completes the extrusion exercises.

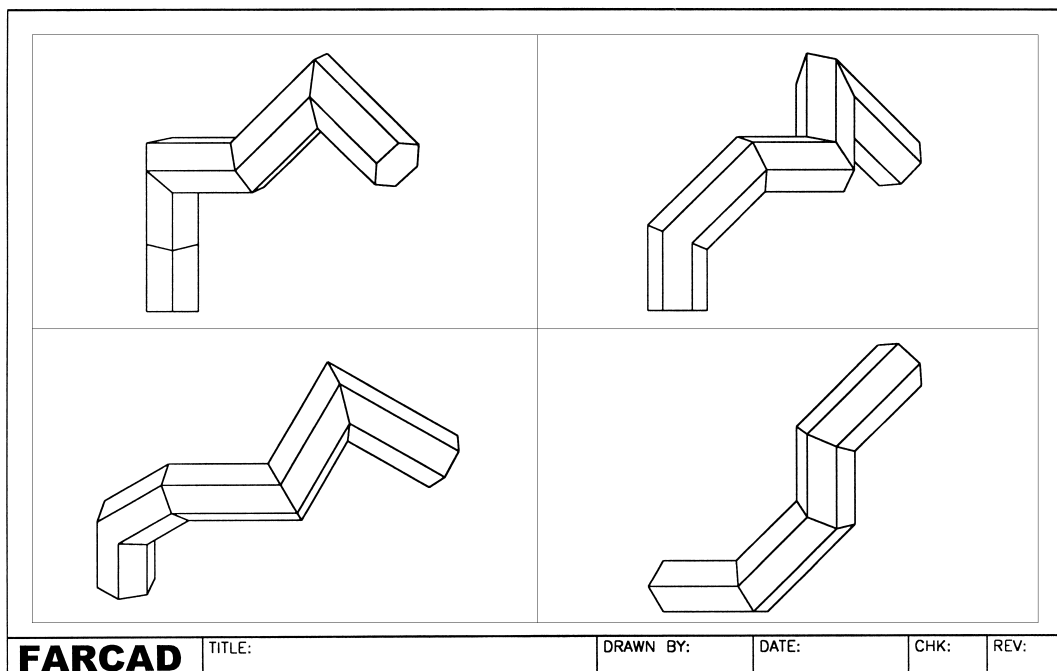


Figure 29.4 Extruded example 4 – a polygon duct effect.

Revolved solids

Solid models can be created by revolving objects (closed polylines, polygons, circles, ellipses, closed splines, regions):

- a) about the X and Y axes
- b) about selected objects by a specified angle.

As with extrusions, very complex models can be obtained from relatively simple shapes.

Revolved example 1: a flagon

- 1 Open the A3SOL template file as normal and in each viewport zoom centre about 0,80,0 at 200 mag.
- 2 a) Set ISOLINES to 8
b) In the top left viewport, alter the viewpoint with VPOINT at the command line, select the ROTATE option and enter angles of 30 and -20
- 3 Restore UCS FRONT and make the top right viewport active
- 4 Refer to Fig. 29.5 and draw a **closed** polyline outline using the sizes given as a reference – or design your own
- 5 Menu bar with **Draw-Solids-Revolve** and:
 - prompt* Select objects
 - respond* **pick the polyline then right-click**
 - prompt* Specify start point for axis of revolution or define axis by [Object/X (axis)/Y (axis)]
 - enter* **Y <R>** – the Y axis option
 - prompt* Specify angle of revolution
 - respond* **right-click** to accept the 360 (full circle) default
- 6 The polyline outline will be revolved into a swept revolved solid
- 7 Shade one of the 3D viewports and use 3D orbit
- 8 Save the completed model

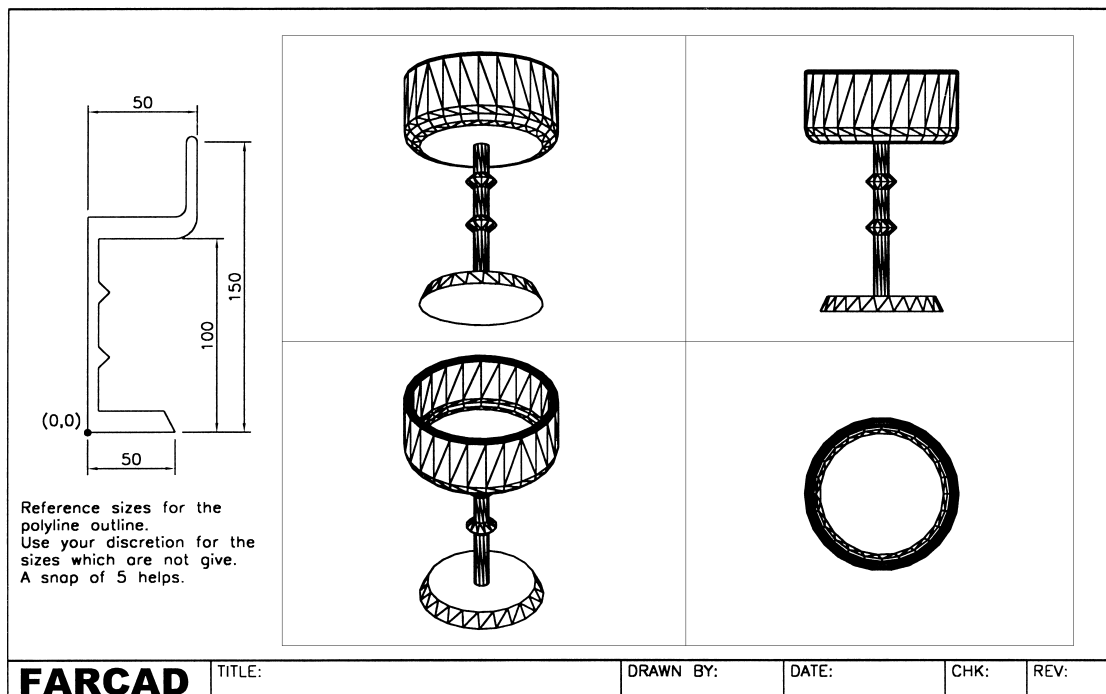


Figure 29.5 Revolved example 1 – a flagon.

Revolved example 2: a shaft

- 1 Open the A3SOL template file with normal settings
- 2 Make the lower right viewport active
- 3 In each viewport zoom centre about 50,0,0 at 200 magnification
- 4 Refer to Fig. 29.6 and draw a **closed** polyline outline using the sizes given. Use the (0,-30) start point. Note that I have only displayed the 3D viewport with this exercise. This was for convenience purposes only
- 5 Set ISOLINES to 12
- 6 Menu bar with **Draw-Solids-Revolve** and:
 - prompt* Select objects
 - respond* **pick the polyline then right-click**
 - prompt* Specify start point of axis of revolution or define axis by [Object/X (axis)/Y (axis)]
 - enter* **X <R>** – the X axis option
 - prompt* Specify angle of revolution
 - enter* **360 <R>**
- 7 The polyline outline will be revolved about the X axis as fig(a)
- 8 Observe the effect then undo the revolved effect to restore the original polyline outline

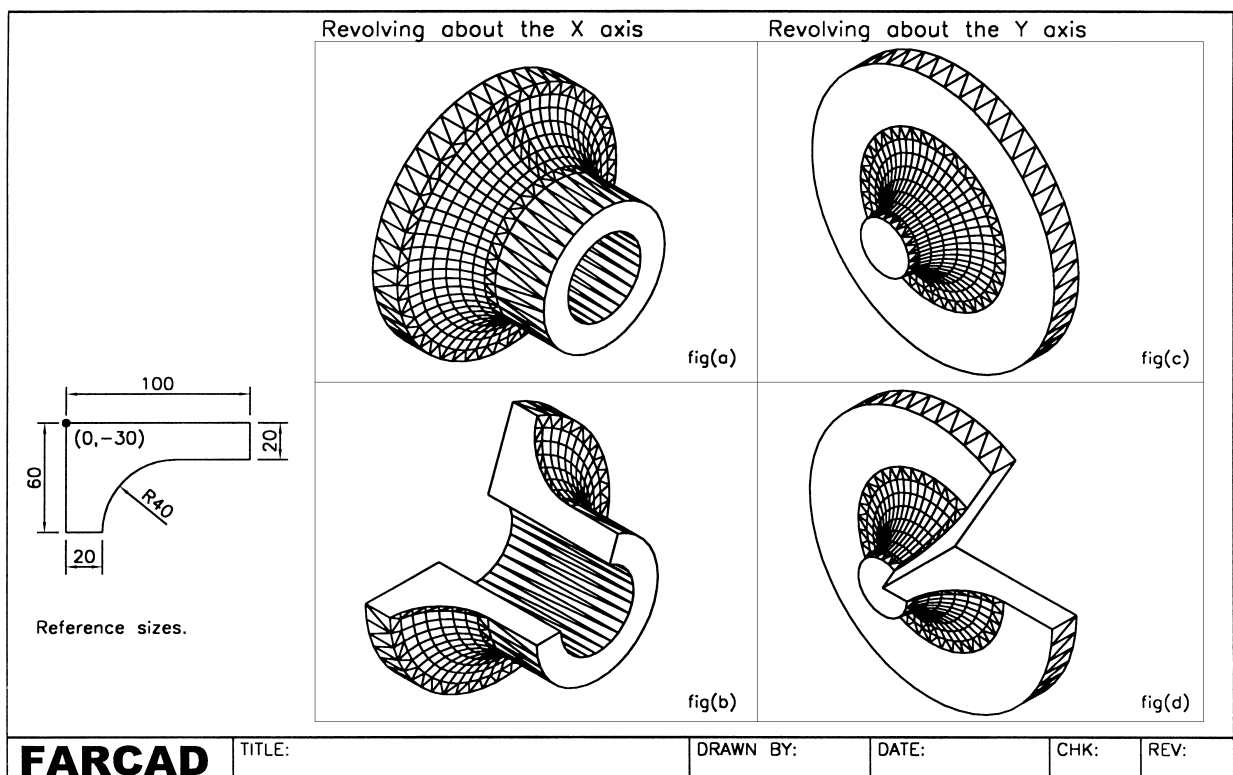


Figure 29.6 Revolved example 2 – a shaft.

- 9 Select the revolve icon from the solids toolbar and:
 - a) objects: select the polyline outline then right-click
 - b) axis of revolution: X axis
 - c) angle of revolution: 250
 - d) the revolved effect will be as fig(b). Observe then undo
- 10 At the command line enter **REVOLVE <R>** and:
 - a) objects: select polyline then right-click
 - b) axis of revolution: enter Y <R>
 - c) angle of revolution: 360
 - d) the result will be as fig(c) – pan needed?
 - e) observe then undo the revolved effect
- 11 Finally revolve the polyline:
 - a) about the Y axis
 - b) with a 300 angle of revolution – fig(d)
- 10 Shade, 3D orbit, etc. then save if required



This completes the swept primitive exercises.

Plotting multiple viewport layouts with hide

To obtain a plot of a multiple layout with hide, the user needs to use the **MVIEW** command. To demonstrate this command:

- 1 Open any multiple viewport drawing, e.g. the splined shaft example
- 2 Enter paper space
- 3 At the command line enter **MVIEW <R>** and:


```

prompt Specify corner of viewport or [various options]
enter H <R> – the hideplot option
prompt Hidden line removal for plotting [ON/OFF]
enter ON <R>
prompt Select objects
respond pick the borders of all viewports which have to have hidden line
removal
then right-click
      
```
- 4 The command line is returned
- 5 Proceed with the normal plot sequence, but ensure that the Hide Plot option is **NOT ACTIVE** (i.e. no tick)
- 6 This procedure should be used at all times when multi-viewport plots are required with hidden line removal

Summary

- 1 Swept solids are obtained with the extrude and revolve commands
- 2 The two commands can be activated:
 - a) by icon selection from the Solids toolbar
 - b) from the command line with Draw-Solids
 - c) by entering EXTRUDE and REVOLVE at the command line
- 3 Very complex models can be obtained from simple shapes
- 4 Only certain 'shapes' can be extruded/revolved. These are closed polylines, circles, ellipses, polygons, closed splines and regions (more on this in a later chapter)
- 5 Objects can be extruded:
 - a) to a specified height
 - b) with/without a taper angle
 - c) along a path curve
- 6 The extruded height is in the Z direction and can be positive or negative
- 7 The taper angle can be positive or negative
- 8 Objects can be revolved:
 - a) about the X and Y axes
 - b) about an object
 - c) by specifying two points on the axis of revolution
- 9 The angle of revolution can be full (360) or partial.

Assignment

During the excavation of the ancient city of CADOPOLIS, the intrepid diggers uncovered two artefacts, both of which they attributed to our master builder MACFARAMUS. They decided (how we will never know), that the artefacts were scale models of a pyramid and the partial wheel from a chariot. It is these that you have to create as solid models from swept primitives.

Activity 19: Two swept primitive models designed by MACFARAMUS.

These two models have to be created from closed polylines. The dimensions taken by the site engineers were not complete and only the basic sizes have been given. You have to use your own discretion when drawing the two outlines. The procedure for both models is:

- 1 Open your template file as normal
- 2 Draw the outline of the model as a closed polyline, both with UCS BASE, layer MODEL and with the lower right viewport active
- 3 Extrude the pyramid outline for a height of 150 and a taper angle of 10
- 4 Revolve the wheel outline about the X or Y axis dependent on how the original outline was drawn. The partial angle of revolution is to be 270 degrees
- 5 Centre the models in the viewports
- 6 *Note:* I have displayed both models on the one sheet of paper with:
 - a) pyramid: 3D and front views
 - b) wheel: two 3D views, from above and from below

Boolean operations and composite solids

The basic and swept solids which have been created are called **primitives** and are the 'basic tools' for solid modelling. With these primitives the user can create **composite solids**, so called because they are 'composed' of two or more solid primitives, i.e.

- a) primitive: a box, wedge, cylinder, extrusion, etc.
- b) composite: a solid made from two or more primitives.

Composite solids are created from primitives using the three **Boolean** operations of union, subtraction and intersection. Figure 30.1 demonstrates these operations with two primitives:

- a) a box
- b) a cylinder 'penetrating' the box

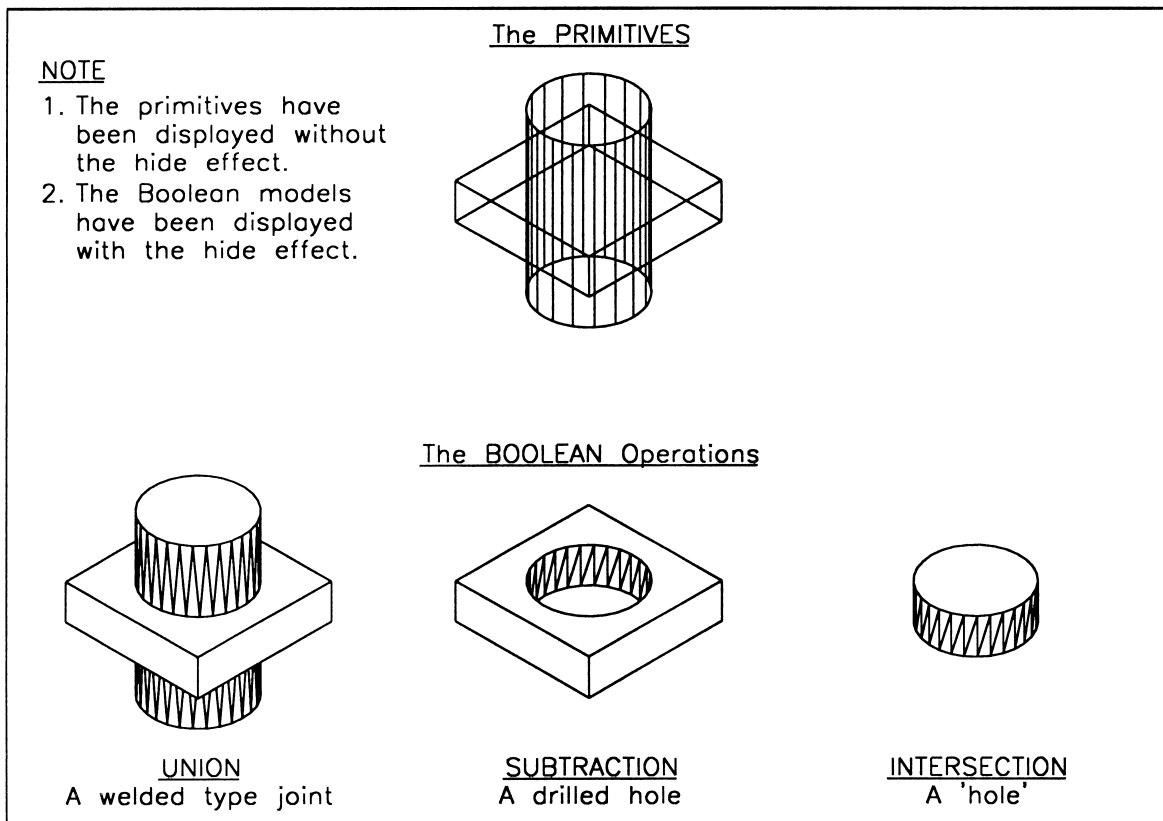


Figure 30.1 The three BOOLEAN operations.

Union

- 1 This operation involves 'joining' two or more primitives to form a single composite, the user selecting *all objects to be unioned*
- 2 The operation can be considered similar to welding two or more components together.

Subtraction

- 1 This involves removing one or more solids from another solid thereby creating the composite. The user selects:
 - a) *the source solid*
 - b) *the solids to be subtracted from the source solid*
- 2 The result of a subtraction operation can be likened to a drilled hole, i.e. if the cylinder is subtracted from the box, a hole will be obtained in the box.
- 3 *Note:* the source solid is generally 'the larger solid' i.e. you cannot normally take a large solid from a small solid

Intersection

- 1 This operation gives a composite solid from other solids which have a common volume, the user selecting *all objects which have to be intersected*
- 2 The box/cylinder illustration of the intersection operation gives a 'disc shape' or 'hole' i.e. if the box and cylinder are intersected, the common volume is the disc shape.

Creating a composite solid from primitives

There is no 'correct or ideal' method of creating a composite, i.e. the Boolean operations selected by one user may be different from those selected by another user, but the final composite may be the same. To demonstrate this, we will create an L-shaped component by three different methods, so:

- 1 Open your A3SOL template file refer to Fig. 30.2 and:
 - a) enter paper space
 - b) erase any text and the four viewports
 - c) with layer VP current create a single viewport with:
 - i) first point: pick to suit in lower left corner area
 - ii) other corner: enter @360,220
 - d) return to model space, UCS BASE, layer MODEL
 - e) set a SE Isometric viewpoint
- 2 Create two box primitives:

<ol style="list-style-type: none"> a) corner: 0,0,0 cube option length: 100 colour: red 	<ol style="list-style-type: none"> b) corner: 100,100,100 length: -60 width: -100 height: -70 colour: blue
---	---
- 3 Create another two box primitives:

<ol style="list-style-type: none"> a) corner: 125,125,0 length: 100 width: 100 height: 30 colour: red 	<ol style="list-style-type: none"> b) corner: 125,125,30 length: 40 width: 100 height: 70 colour: blue
--	---

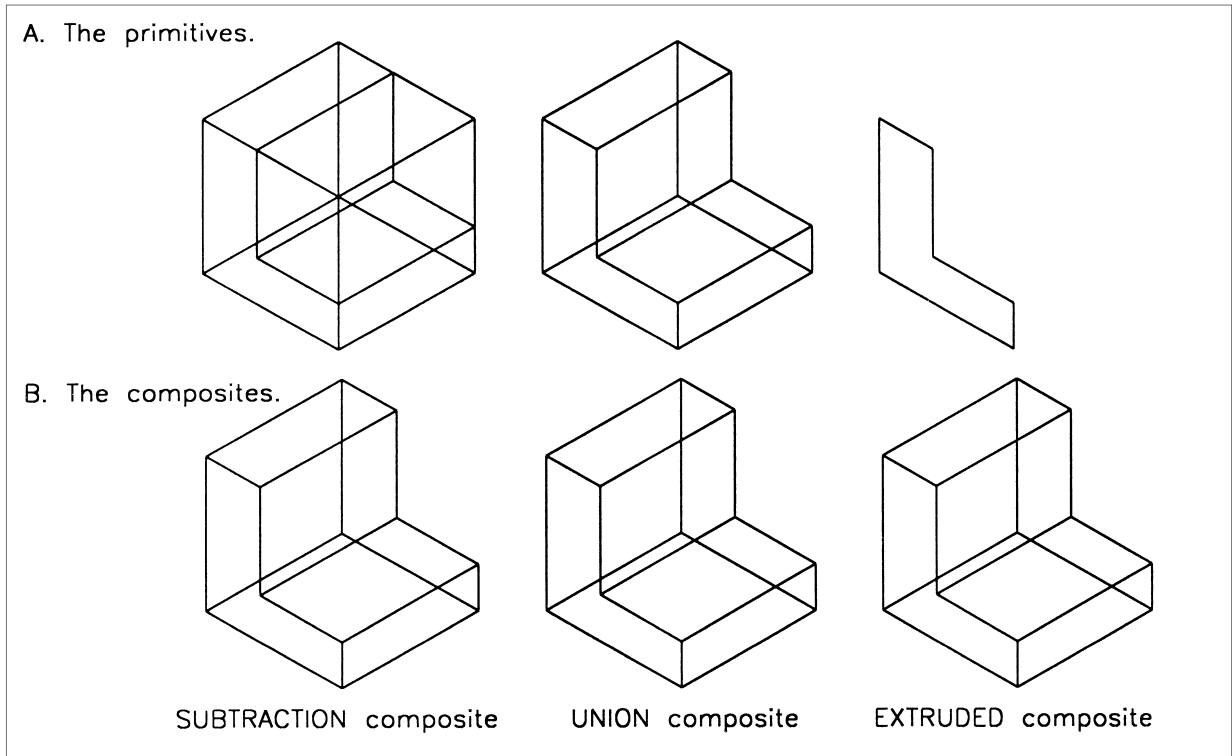


Figure 30.2 Creating the same composite by three different methods.

- 4 Restore UCS FRONT and draw a 2D polyline shape:
start pt: 500,250 next pt: @100,0 next pt: @0,30
next pt: @-60,0 next pt: @0,70 next pt: @-40,0
next pt: close
- 5 Restore UCS BASE and zoom centre about 100,250,0 at 400 mag.
- 6 From the menu bar select **Modify-Solids Editing-Subtract** and:
prompt Select solids and regions to subtract from..
Select objects
respond **pick the left red box then right-click**
prompt Select solids and regions to subtract..
Select objects
respond **pick the left blue box then right-click**
and the blue box is subtracted from the red box
- 7 Menu bar with **Modify-Solids Editing-Union** and:
prompt Select objects
respond **pick the middle red and blue boxes then right-click**
and the two boxes will be unioned
- 8 Restore UCS FRONT and:
a) select the EXTRUDE icon from the Solids toolbar
b) pick the L shaped polyline then right-click
c) enter an extruded height of -100 with 0 taper
d) the L-shape polyline is extruded into a composite

9 *Task*

- a) Hide the models – all the same
 - b) Gouraud shade the models and note colour effect between the union and subtraction composites – any comment?
 - c) Return to 2D wireframe representation
 - d) Menu bar with **Tools-Inquiry-Region/Mass Properties** and:
 - prompt* Select objects
 - respond* **pick left composite then right-click**
 - and* AutoCAD Text Window with:
 - Mass: 580000.00
 - Volume: 580000.00
 - Bounding box, Centroid etc.
 - enter* **N <R>** in response to 'Write analysis to file' prompt
- d) Repeat the MASSPROP command and select the middle and right composites – same mass and volume?

10 *Questions*

- a) Why are the mass and volume the same?
Answer: AutoCAD 2002 assumes a density value of 1 and does not support different material densities
 - b) Is the volume of 580000 correct for the L shape?
 - c) What are the volume units?
- 11 Now that we have investigated the Boolean operations, we will create some composite solid models which (I hole) will be interesting.

Summary

- 1 There are three Boolean operations – union, subtraction and intersection
- 2 The three operations can be activated:
 - a) from the menu bar with Modify-Solids Editing
 - b) in icon form from the Solids Editing toolbar
 - c) by entering the command from the keyboard.
- 3 The Boolean operations are derived from Boolean Algebra (set theory) and are essential for the creation of solid composites from primitives.

Composite model 1 – a machine support

In this exercise we will create a composite solid from the box, wedge and cylinder primitives using the three Boolean operations. Once created, we will dimension the model using viewport specific layers.

The exercise is quite simple and you should have no difficulty in following the various steps in the model construction. Try and work out why the various entries are given – do not just accept them.

- 1 Open your A3SOL template file with normal settings and display the Solids, Solids Editing and other toolbars to suit.
- 2 Refer to Fig. 31.1 which displays only the 3D viewport of the model at various stages of its construction

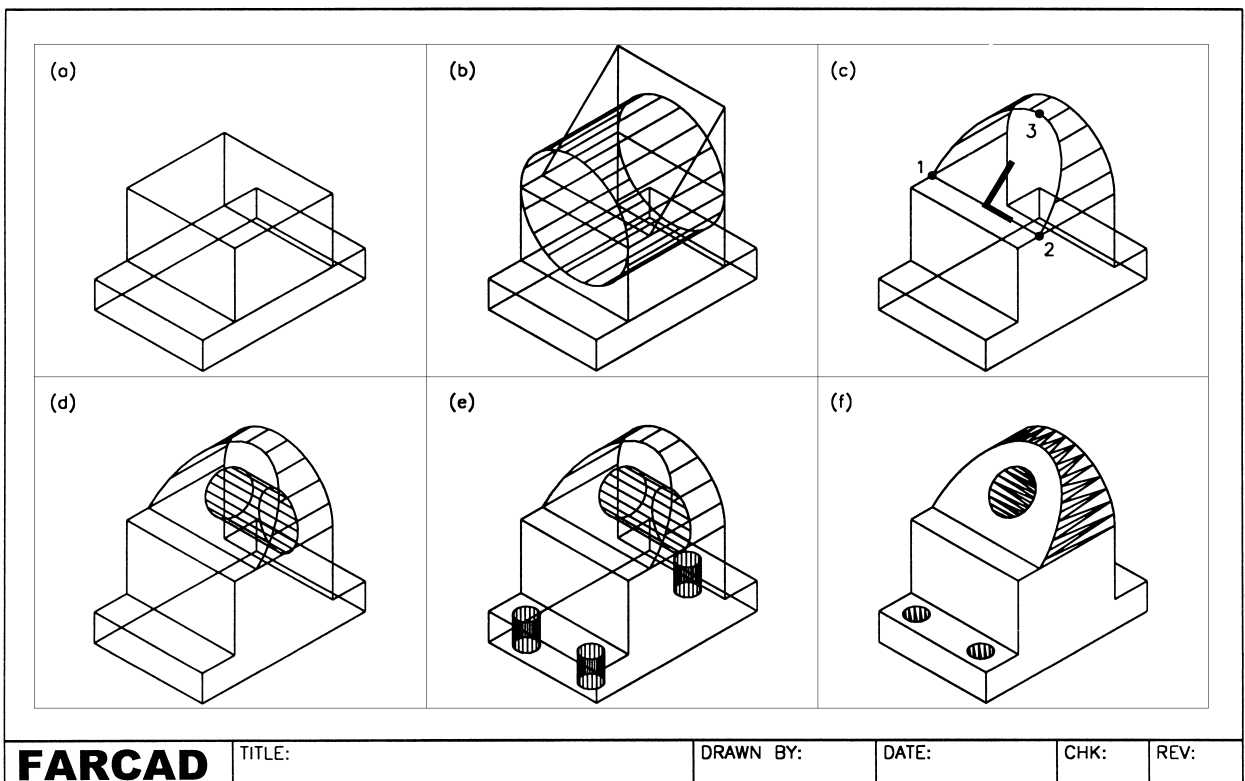


Figure 31.1 Creation of composite model 1 – a machine support.

- 3 In each viewport, zoom-extents then zoom to 1.5 scale
- 4 With the 3D viewport active, use the BOX icon to create two primitives as fig(a) with:

a) corner: 0,0,0	b) corner: 100,120,25
length: 100	length: -100
width: 150	width: -90
height: 25	height: 60
colour: red	colour: blue
- 5 Create a cylinder on top of the blue box with:
 - a) centre: 50,30,85
 - b) radius: 50
 - c) height/other end: enter **C <R>** then **@0,90,0 <R>**
 - d) colour: green
- 6 Create a wedge with:
 - a) corner: 0,120,85
 - b) length: 70; width: 100; height: 70
 - c) colour: magenta
- 7 Rotate (2D rotate) the magenta wedge:
 - a) about the point: 0,120,85
 - b) by: -90 degrees to give fig(b)
- 8 Select the INTERSECTION icon from the Solids Editing toolbar and:

prompt Select objects

respond **pick the green cylinder and magenta wedge then right-click**
- 9 Select the UNION icon from the Solids Editing toolbar and:

prompt Select objects

respond **pick the red and blue boxes and the intersected wedge/cylinder then right-click**
- 10 The model now appears as fig(c)
- 11 Refer to fig(c) and menu bar with **Tools-New UCS-3 Point** and set a new UCS with:
 - a) origin: MIDpoint icon and pick line 12
 - b) X axis: ENDpoint icon and pick pt2
 - c) Y axis: QUADrant icon and pick pt3 on curve
- 12 The UCS icon will move and align itself on the sloped surface
Note: if icon does not move, menu bar with View-Display-UCS Icon and ensure that On and Origin are ticked
- 13 Save this UCS position as SLOPE
- 14 Create a cylinder with:
 - a) centre: 0,35,0
 - b) radius: 18
 - c) height: -100
 - d) colour: number 54 – use the CHPROP command
- 15 Select the SUBTRACT icon from the Solids Editing toolbar and:

prompt Select solids and regions to subtract from..

Select objects

respond **pick the composite model then right-click**

prompt Select solids or regions to subtract

Select objects

respond **pick the cylinder then right-click – fig(d)**

- 16 Restore UCS BASE
- 17 Create a cylinder with:
 - a) centre: 20,15,0
 - b) radius: 9
 - c) height: 25
 - d) colour: number 174
- 18 Multiple copy the cylinder:
 - a) from: 20,15
 - b) by: @60,0 and by: @30,120
- 20 Using the SUBTRACT icon:
 - a) select the original composite then right-click
 - b) pick the three cylinders then right-click
- 21 The model is now complete and is displayed in fig(e) without hide and in fig(f) with hide.
- 22 Select the Model tab and Gouraud shade the model, then use the 3D orbit command to rotate the shaded model
- 23 Restore the model to 2D wireframe representation, then make the MVLAY1 tab active
- 24 At this stage save the composite as **MODR2002\MACHSUPP**

Making the viewport specific layers

The MVLAY1 tab screen displays the model in a four viewport configuration and we now want to add some dimensions. These dimensions must be added on viewport specific layers and these layers must now be created.

- 1 At the command line enter **VPLAYER <R>** and:
 - prompt* Enter an option [?/Freeze/Thaw/Reset/Newfrz/Vpvisdflt]
 - enter* **N <R>** – the Newfrz (new viewport freeze) option
 - prompt* Enter name(s) of new layers frozen in all viewports
 - enter* **DIMTL,DIMTR,DIMBR <R>**
 - prompt* Enter an option [?/Freeze/Thaw/..
 - respond* **right-click** – as finished with command
- 2 The command line entry VPLAYER is to activate viewport layer
- 3 Make the top left viewport active and menu bar with **Format-Layer** and:
 - prompt* Layer Properties Manager dialogue box
 - note* three new layers – DIMBR, DIMTL, DIMTR with:
 - i) Frozen in current viewport – blue icon
 - ii) Frozen in new viewport – blue icon
 - respond*
 1. activate Show Details
 2. **pick DIMTL** (highlights) and note details – Freeze in current viewport is on (tick)
 3. **pick blue icon Freeze in active viewport** to Thaw the layer and the tick is removed from Freeze in current viewport details list
 4. pick OK
- 4 With the top right viewport active, Format-Layer and:
 - a) pick layer **DIMTR**
 - b) toggle the blue Freeze in current viewport icon to yellow to Thaw layer DIMTR in the top right viewport
 - c) pick OK

- 5 With lower right viewport active, Format-Layer and:
 - a) pick layer **DIMBR**
 - b) toggle Freeze icon in current viewport from blue to yellow, i.e. from Frozen to Thaw to Thaw layer DIMBR in the lower right viewport
 - c) pick OK
- 6 What has been achieved in this section?
 - a) three new viewport specific layers have been made
 - b) these layers have been named DIMTL for the top left viewport, DIMTR for the top right viewport and DIMBR for the bottom left viewport
 - c) the three layers were originally created :
 - i) frozen in new viewports
 - ii) currently frozen in all viewports
 - d) each layer was currently thawed in a specific viewport, e.g. layer DIMTL is currently thawed in the top left viewport but is currently frozen in the other three viewports. Layers DIMTR and DIMBR are currently frozen in the top left viewport.
- 7 Before adding the dimensions, change the colour of the three new layers (DIMTL, DIMTR, DIMBR) to magenta using the Layer Properties Manger dialogue box.

Adding the dimensions

- 1 Before the dimensions are added to the model, menu bar with **Dimension-Style** and using the Dimension Style Manager dialogue box:
 - a) 3DSTD the current (and only) style?
 - b) pick Modify
 - c) pick the Fit tab
 - d) Alter Scale for Features: Use overall scale of 2
 - e) pick OK to return to Dimension Style Manager dialogue box
 - f) pick Close
- 2 This will scale all the dimension parameters by 2
- 3 Make the lower right viewport active and:
 - a) restore UCS BASE
 - b) make layer DIMBR current
 - c) refer to Fig. 31.2
- 4 With Dimension-Linear from the menu bar, or with the LINEAR dimension icon from the Dimensions toolbar, add:
 - a) the horizontal dimension
 - b) the four vertical dimensions using the baseline option
- 5 With the top right viewport active:
 - a) restore UCS FRONT
 - b) make layer DIMTR current
 - c) add the two linear dimensions
- 6 Make the top left viewport active and:
 - a) restore UCS RIGHT
 - b) make layer DIMTL current
 - c) add the six dimensions
- 7 The composite model is now complete with dimensions added and can be plotted with the layer VP frozen for effect – Fig. 31.2
- 8 These dimension additions do not need to be saved.

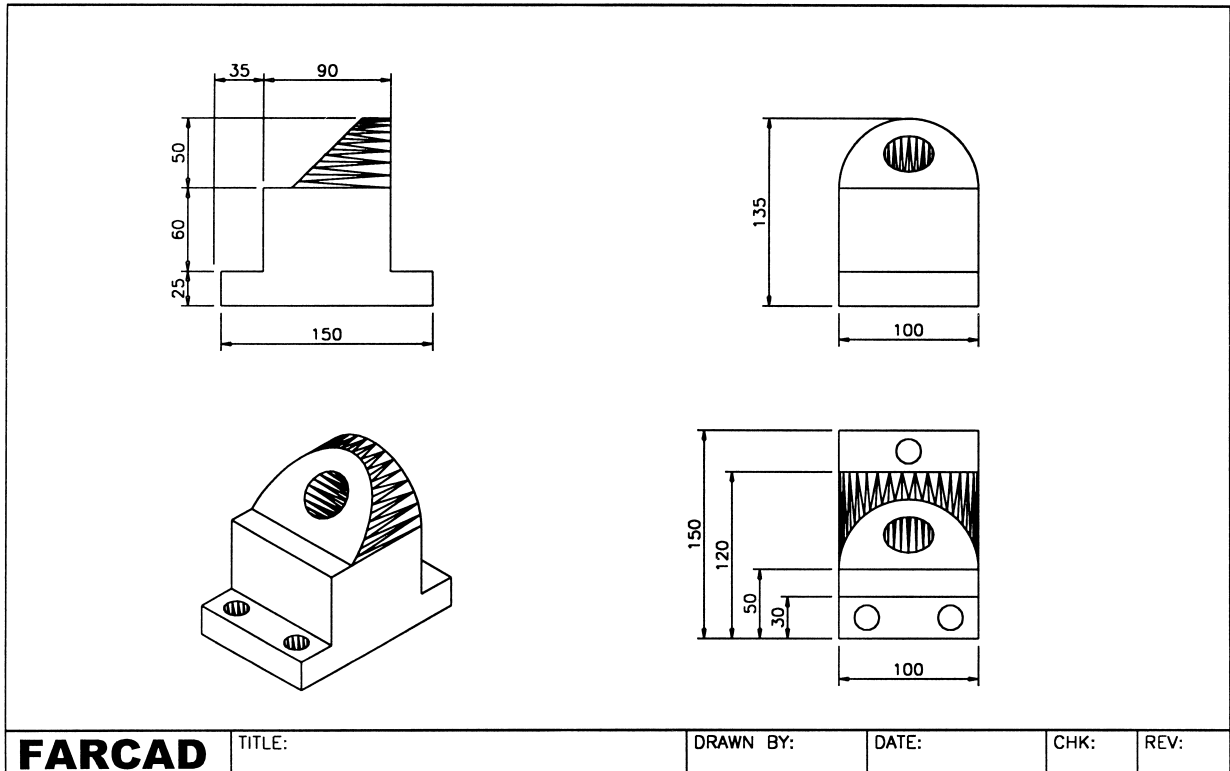


Figure 31.2 Completed solid composite MACHSUPP with dimensions.

Summary

- 1 Primitives and the Boolean operations can be used to create composite solid models.
- 2 Viewport specific layers can be used when adding dimensions to a multi-view model.

Note

In a later chapter we will investigate adding dimensions to a model in paper space.

Composite model 2 – a backing plate

In this exercise we will create a solid from an extruded swept primitive and then subtract various 'holes' to complete the composite. The exercise will also involve altering the viewport layout of the A3SOL template file which is an interesting exercise in itself. As with all the exercises, do not just accept the entries – work out why the various values are being used.

The model

Refer to Fig. 32.1 which details the model to be created and gives the relevant sizes. As an aside draw the three orthogonal views as given and then add the isometric (the arc 'hole' is interesting to complete in an isometric view). Time how long it takes to complete this 2D drawing. I spent about an hour to complete the four views with dimensions.

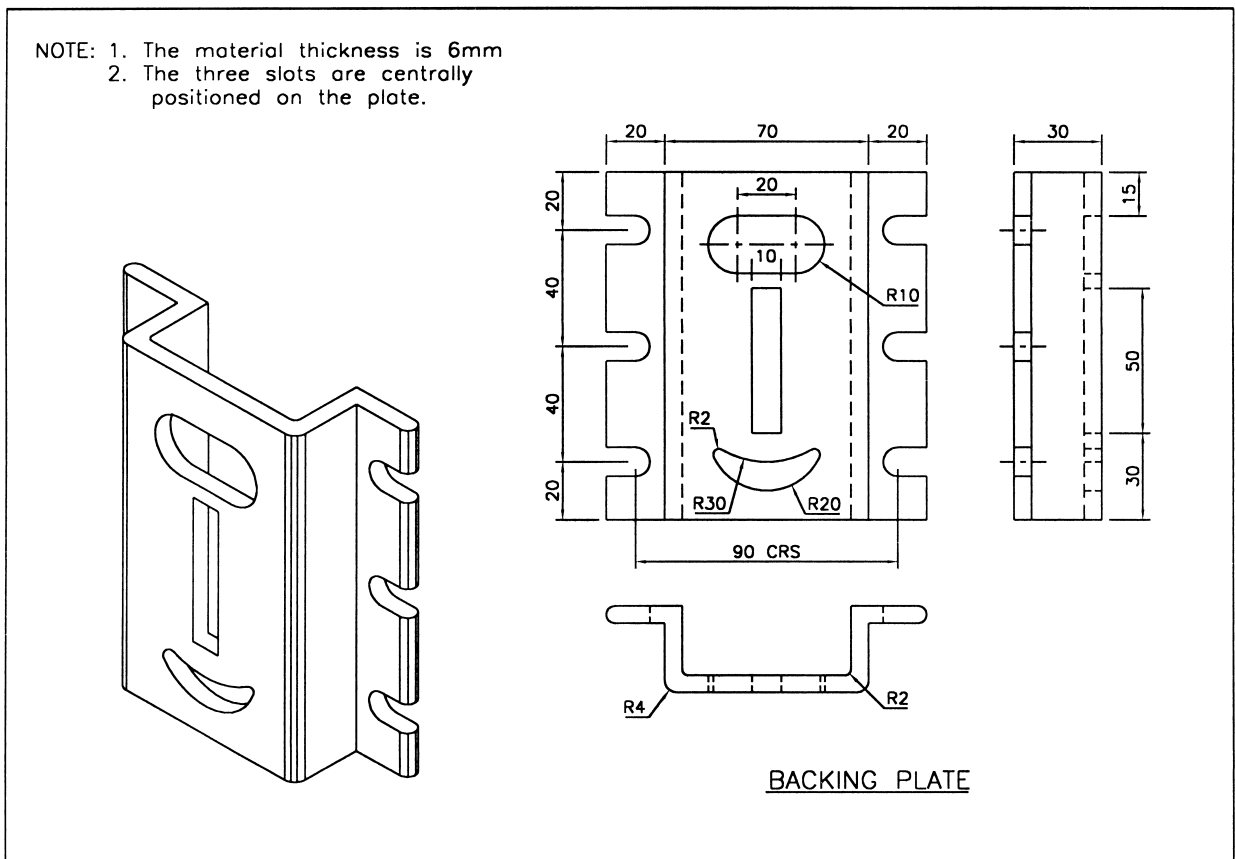


Figure 32.1 Backing plate drawn as orthogonal views and as isometric.

Setting the viewports

- 1 Open your A3SOL template file and with MVLAY1 tab active:
 - a) enter paper space
 - b) erase the four viewports
 - c) make layer VP current
 - d) refer to Fig. 32.2
- 2 Menu bar with **View-Viewports-1 Viewport** and:

prompt Specify first corner and enter: **10,25 <R>**

prompt Specify other corner and enter: **@155,175 <R>**
- 3 Create another three single viewports using the following coordinate entries:
 - a) first corner: 165,25
other corner: @160,70
 - b) first corner: 165,95
other corner: @160,155
 - c) first corner: 325,95
other corner: @70,155
- 4 In model space:
 - a) make layer MODEL current
 - b) set UCSVP to 0 in each viewport
 - c) set the 3D viewpoints in the viewports as fig(a)
 - d) restore UCS BASE
 - e) zoom centre about 0,10,60 at **IXP** – yes enter IXP
 - f) in paper space zoom in on viewport B then model space with viewport B active

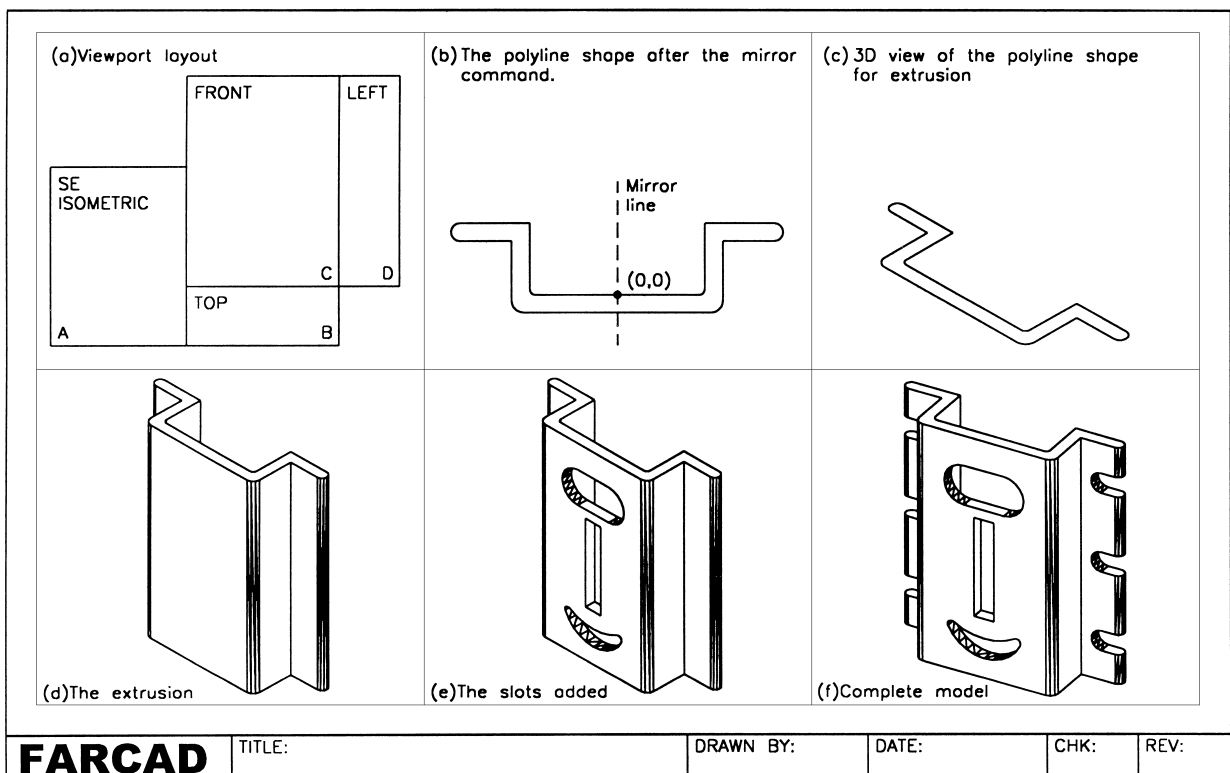


Figure 32.2 Steps in the creation of the backing plate composite.

Creating the extrusion

- 1 Using the polyline icon from the Draw toolbar, create a single polyline from line and arc segments with the following entries:
 - Start point* 0,0
 - Next point* @27,0
 - Next point* Arc option, with endpoint of arc @2,2
 - Next point* Line option, to @0,22
 - Next point* @23,0
 - Next point* Arc option, with endpoint of arc @0,-6
 - Next point* Line option, to @-17,0
 - Next point* @0,-20
 - Next point* Arc option, with endpoint of arc @-4,-4
 - Next point* Line option, to @-31,0
 - Next point* right-click/enter
- 2 Mirror the polyline shape about the points 0,0 and 0,50 and do not delete source objects
- 3 Use the menu bar sequence **Modify-Object-Polyline** and:
 - a) pick any point on the right-hand polyline
 - b) enter **J <R>** – the join option
 - c) pick the two polyline shapes then right-click
 - d) enter **X <R>** to end the command
- 4 The two ‘halves’ of the polyline have been joined into a single polyline as fig(b) in plan view and fig(c) in 3D.
- 5 At the command line enter **ISOLINES <R>** and:
 - prompt* Enter new value for ISOLINES<24>
 - enter* **3 <R>**
- 6 Note: the ISOLINES system variable has been reduced from 18 to 3 due to the ‘corner edges’ of the model. With a value of 18, the extrusion would result in these corner edges being displayed ‘very dense’.
- 7 Select the EXTRUDE icon from the Solids toolbar and:
 - prompt* Select objects
 - respond* **pick any point on the polyline then right-click**
 - prompt* Specify height of extrusion or [Path] and enter: **120 <R>**
 - prompt* Specify angle of taper for extrusion and enter: **0 <R>**
- 8 The polyline shape will be extruded as fig(d)
- 9 Paper zoom-previous then return to model space

Creating the ‘holes’

- 1 Restore UCS FRONT with viewport C active
- 2 Create a box primitive with:
 - a) corner: -5,30,0
 - b) length: 10; width: 50; height: 6
 - c) colour: blue
- 3 Create the following three primitives:

<i>box</i>	<i>cylinder</i>	<i>cylinder</i>
corner: -10,85,0	centre: -10,95,0	centre: 10,95,0
length: 20	radius: 10	radius: 10
width: 20	height: 6	height: 6
height: 6	colour: green	colour: green
colour: green		
- 4 Union the three green primitives
- 5 Draw two circles:
 - a) centre: 0,30 with radius: 20
 - b) centre: 0,50 with radius: 30
 - c) both circles colour green
- 6 Trim the circles ‘to each other’ and fillet the ‘corners’ with a radius of 2
- 7 Convert the four arcs into a single polyline with the menu bar sequence **Modify-Object-Polyline** using the **Join** option
- 8 Extrude the arced polyline for a height of 6 with 0 taper, then change the colour of the extrusion to green
- 9 Subtract the green and blue ‘holes’ from the red extrusion to display the model as fig(e)
- 10 Create the following two primitives:

<i>cylinder</i>	<i>box</i>
centre: 45,20,-24	corner: 45,15,-24
radius: 5	length: 30
height: 6	width: 10
colour: magenta	height: 6
	colour: magenta
- 11 Union these two magenta primitives
- 12 Rectangular array the magenta cylinder/box composite:
 - a) for 3 rows and 1 column
 - b) row distance: 40
- 13 Mirror the three arrayed magenta composites about the points 0,0 and 0,50 and do not delete the source objects
- 14 Subtract the six magenta composites from the red extrusion
- 15 The model is now complete – fig(f) and the four viewport layout should be displayed as Fig. 32.3
- 16 Save the model layout as **MODR2002\BACKPLT**
- 17 Shade and use the 3D orbit command in the 3D viewport, then return the model to wireframe representation.

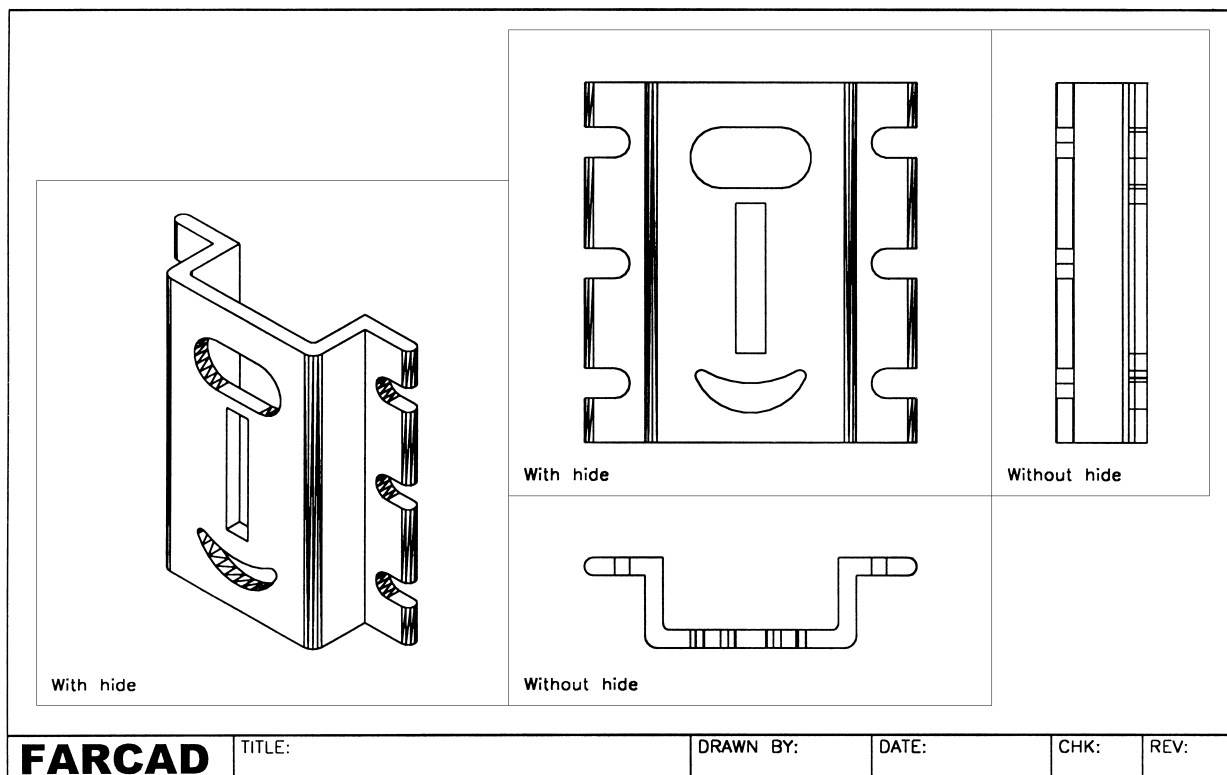


Figure 32.3 Complete solid model composite of backing plate.

Investigating the model

- 1 Display the Inquiry toolbar and make the 3D viewport active
- 2 Select the LIST icon from the Inquiry toolbar and:
 - prompt* Select objects
 - respond* **pick the composite then right-click**
 - and* AutoCAD Text Window
 - with* details of the selected model
 - respond* view the details then cancel the text window
- 3 Select the AREA icon from the Inquiry toolbar and:
 - prompt* Specify first corner or [Object..
 - enter* **O <R>** – the object option
 - prompt* Select objects
 - respond* **pick the composite**
 - and* Area = 38171.70, Perimeter = 0.00
- 4 This is the surface area of the model.
- 5 Select the REGION/MASS PROPERTIES icon and:
 - prompt* Select objects
 - respond* **pick the composite then right-click**
 - prompt* AutoCAD Text Window
 - with* Mass = 98283.83 and other 'technical' information about the model
 - enter* **N <R>** i.e. do not write to file. More on this later.

This completes the exercise on the extruded model.

Composite model 3 – a flange and pipe

This exercise will involve creating a composite mainly as a revolved swept primitive. Cylinder primitives will be subtracted from the revolved primitive to complete the composite.

- 1 Open A3SOL template file as normal
- 2 With the lower left viewport active, restore UCS RIGHT and draw:
 - a) circle: centre at 0,0 with radius: 30
 - b) circle: centre at 0,0 with radius: 40
 - c) line: start point at -200,0 and next point at @0,100
- 3 Select the REVOLVE icon from the Solids toolbar and:
prompt Select objects
respond **pick the smaller circle then right-click**
prompt Specify start point for axis of revolution or define axis
[Object..
enter **O <R>** – the object option
prompt Select an object
respond **pick the lower end of the vertical line**
prompt Specify angle of revolution
enter **70 <R>**
- 4 Change the colour of the revolved pipe to green
- 5 Revolve the larger circle using the same entries as step 3 and change the colour of the pipe to blue
- 6 Erase the line then subtract the green pipe from the blue pipe.
- 7 You may need a paper space zoom for this operation?
- 8 Make the lower right viewport active and restore UCS BASE
- 9 With the POLYLINE icon from the Draw toolbar, draw a continuous closed polyline with the following entries:
Start point: 0,-30
Next point: @0,-10
Next point: @10,0
Next point: Arc option with endpoint: @10,-10
Next point: Line option with endpoint: @0,-50
Next point: @30,0
Next point: @0,70
Next point: close option

- 10 Using the REVOLVE icon from the Solids toolbar:
 - a) pick a point on the polyline then right-click
 - b) enter **X <R>** as the axis of revolution
 - c) enter **360 <R>** as the angle
- 11 Change the colour of the revolved solid to magenta
- 12 With the top left viewport active, restore USC RIGHT
- 13 Create a cylinder with:
 - a) centre: 0,100,20
 - b) radius: 25
 - c) height: 30
 - d) colour: number 222
- 14 Polar array this cylinder about the point 0,0 for five items with 360 fill angle
- 15
 - a) Subtract the five cylinders from the magenta flange
 - b) Union the flange and pipe
- 16 Centre the model with zoom-extents, then zoom to a scale of 1
- 17 The model is complete and is displayed in Fig. 33.1. It can now be saved as **MODR2002\FLPIPE**
- 18 Select the model tab, Gouraud shade then use 3D orbit. The result is impressive?
- 19 *Note:* The three composite models MACHSUPP, BACKPLT and FLPIP will be used in later chapters to demonstrate other commands.

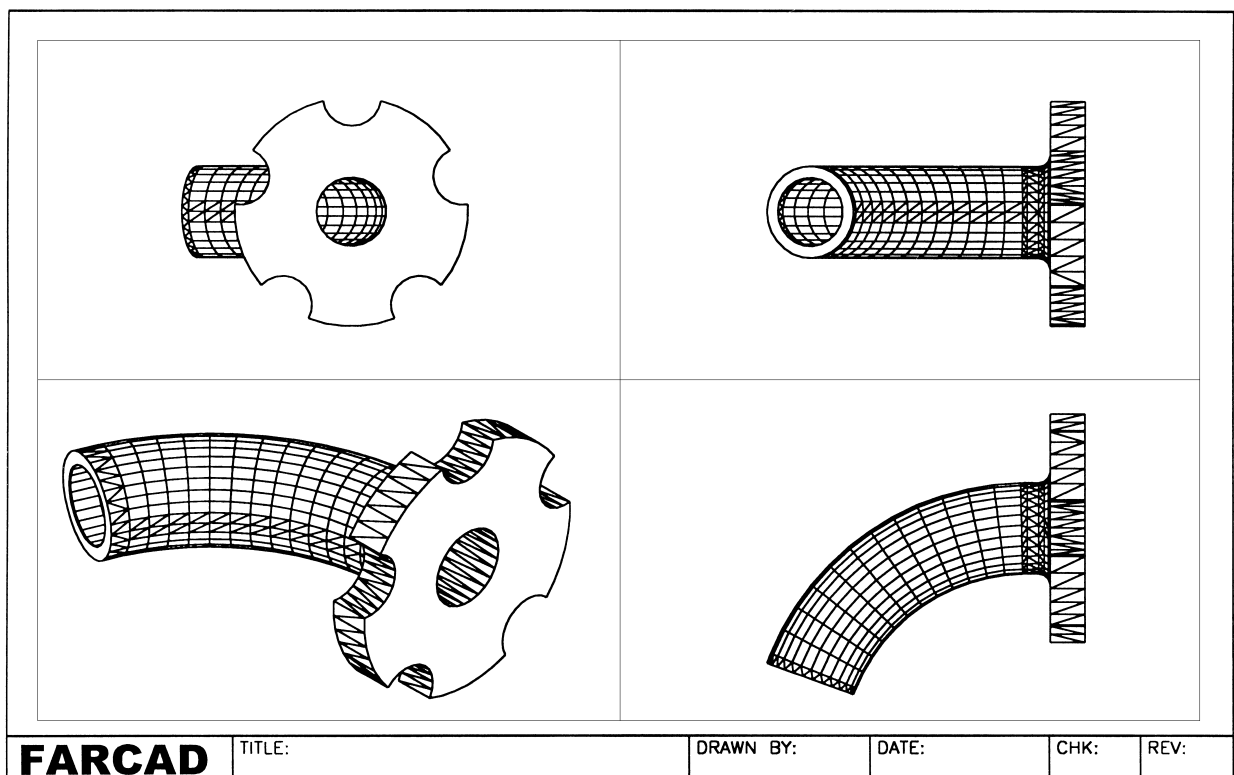


Figure 33.1 The completed composite model 3 – pipe/flange.

The tab layouts

Until now all models have been created with the MVLAY1 tab active, simply because this is the layout I generally use when creating models.

Figure 33.2 is a compilation of the four tab displays on the one sheet. This in itself is an interesting exercise, i.e. can you set the viewport configuration as Fig. 33.2 to display the FLPIP model as shown. In Fig. 33.2 I centred the model with the zoom-extent, zoom 1 effect as used previously.

This completes the revolved composite exercise.

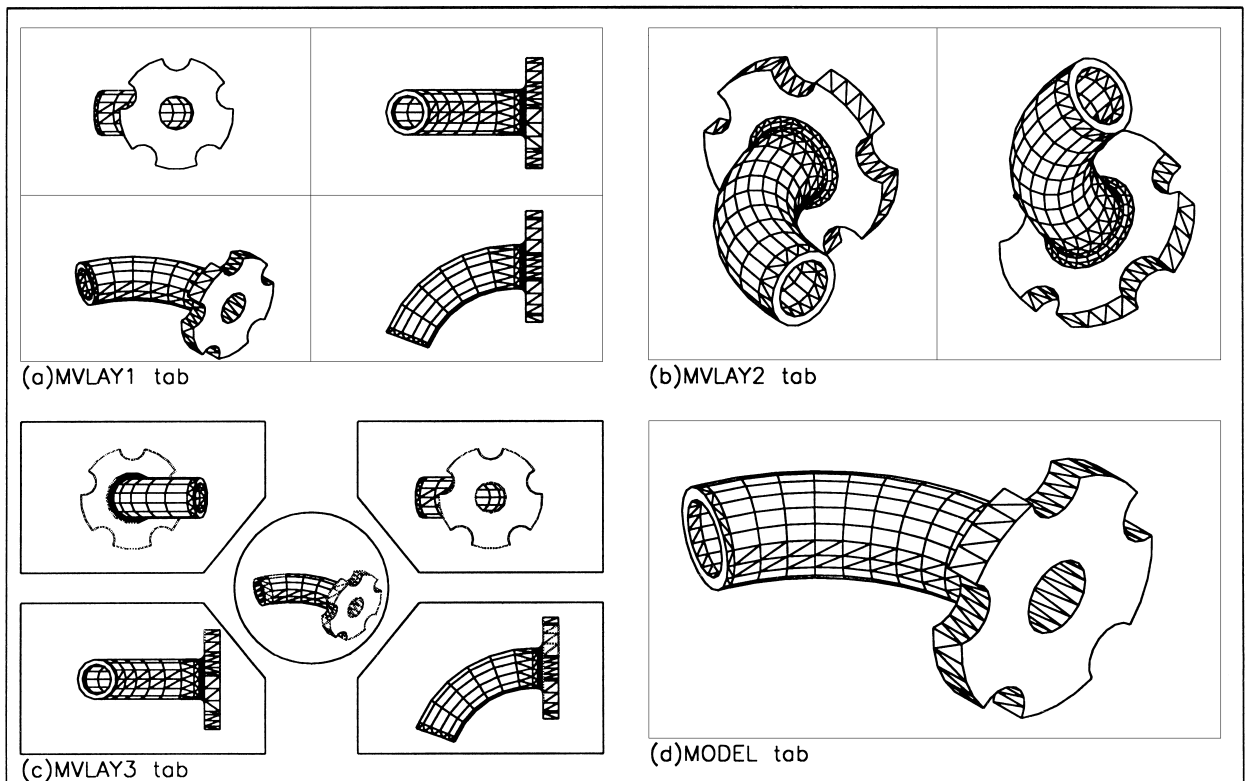


Figure 33.2 The four tab displays of the composite pipe/flange.

The edge primitives

Models can be modified to include a chamfer and fillet effect. These are the edge primitives – the third type of primitive which can be created. In this chapter we will investigate how solids can be constructed with these edge primitives.

Note:

The term 'edge primitive' is not now generally used, the models being considered to have been modified or edited.

Example 1 – a box solid

- 1 Open the A3SOL template file with normal settings.
- 2 Refer to Fig. 34.1(a)
- 3 Use the BOX icon and create a primitive with:
 - a) corner 0,0,0
 - b) cube option with length: 100
- 4 Zoom centre about 50,50,50 at 225 magnification
- 5 Select the CHAMFER icon from the Modify toolbar and:
prompt Select first line or [Polyline/Distance..
respond **pick any line on top surface**
prompt Base surface selection
and a) one face of the cube will be highlighted
b) it will be a 'side' or the 'top'
prompt Enter surface selection option [Next/OK (current)]
respond a) right-click/enter if top face is highlighted
b) enter N<R> if side face is highlighted to highlight the top face then right-click/enter
prompt Specify base surface chamfer distance and enter: **15 <R>**
prompt Specify other surface chamfer distance and enter: **25 <R>**
prompt Select an edge or [Loop]
respond a) pick any three sides on top surface
b) right-click/enter
- 6 The top surface will be chamfered at the selected three edges
- 7 *Note:* entering **L** for loop will allow all edges to be chamfered with a single pick
- 8 Select the FILLET icon from the Modify toolbar and:
prompt Select first object or [Polyline/Radius/Trim]
respond **pick any line on the base surface of the cube**
prompt Enter fillet radius and enter: **20 <R>**
prompt Select edge or [Chain/Radius]
respond **pick any three edges of the base then right-click**
- 9 The base of the cube will be filleted at the three selected edges and displayed as fig(a).

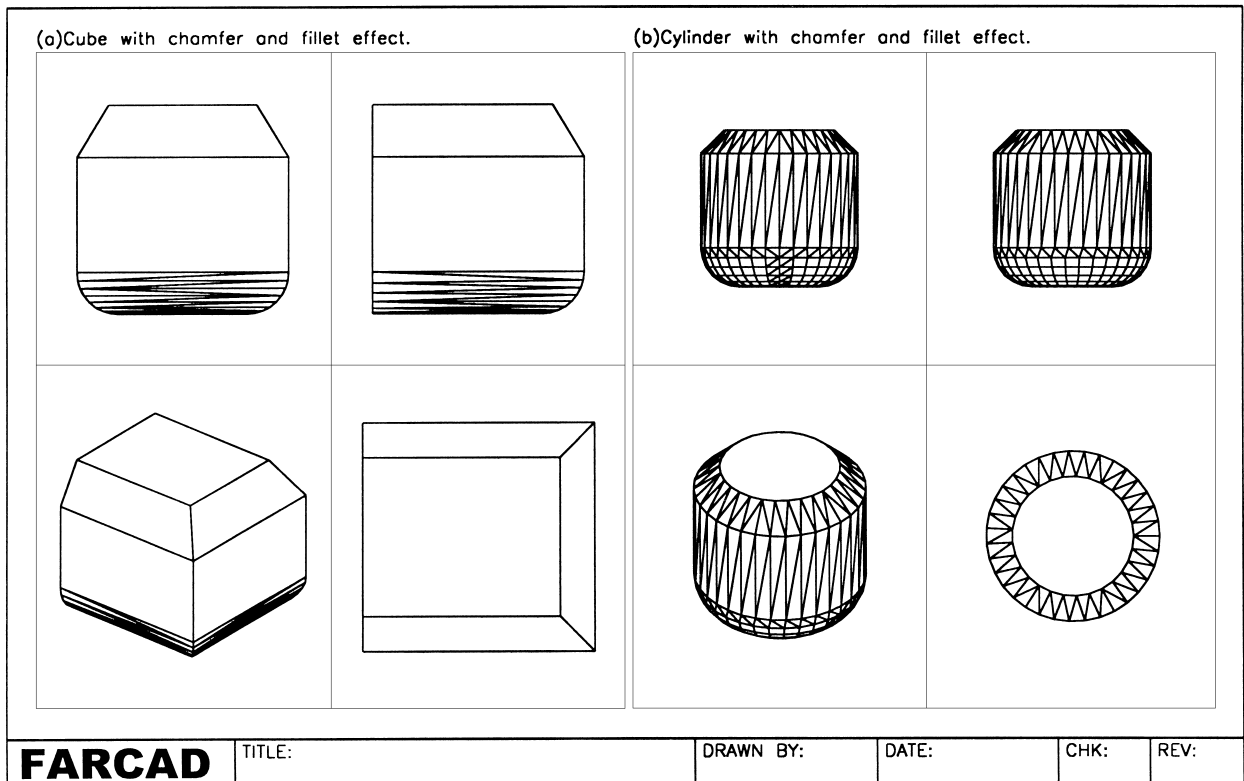


Figure 34.1 Modifying solids with the chamfer/fillet commands.

Example 2 – a cylinder solid

- 1 Erase the cube and at the command line enter **ISOLINES <R>** and check the value is 12
- 2 Use the CYLINDER icon to create a cylinder with:
 - a) centre: 0,0,0
 - b) radius: 50
 - c) height: 100
- 3 Zoom centre about 0,0,50 at 200 magnification
- 4 In paper space zoom-in on the 3D viewport then return to model space
- 5 Select the CHAMFER icon and:


```
prompt Select first line or..
respond pick the top surface circle edge
prompt Base surface selection
then Enter surface selection option
respond <RETURN> as the required surface is highlighted
prompt Specify base surface chamfer distance and enter: 15 <R>
prompt Specify other surface chamfer distance and enter: 15 <R>
prompt Select an edge or..
respond pick top circle edge then right-click/enter
```
- 6 The top of the cylinder is chamfered with the entered values

- 7 Select the FILLET icon and:
 - prompt* Select first object..
 - respond* **pick bottom circle of cylinder**
 - prompt* Enter fillet radius and enter: **25 <R>**
 - prompt* Select an edge..
 - respond* **right-click** as bottom edge already selected
- 8 The cylinder is filleted at the base
- 9 In paper space zoom-previous then return to model space
- 10 The chamfer/fillet effect of the cylinder is displayed as fig(b)

Example 3: A composite edge primitive solid

- 1 Erase the cylinder model and with MVLAY1 tab, UCS BASE, layer MODEL and the lower left viewport active, create four primitives with:

<i>box</i>	<i>cylinder</i>	<i>box</i>	<i>cylinder</i>
corner: 0,0,0	centre: 75,60,0	corner: 50,0,40	centre: 150,0,0
length: 150	radius: 40	length: 60	radius: 30
width: 120	height: 100	width: 120	height: 100
height: 100	colour: green	height: 40	colour: magenta
colour: red		colour: blue	
- 2 In all viewports, zoom centre about 75,60,50 at 200 mag in the 3D viewport and 150 mag in the other viewports
- 3 Subtract the green, blue and magenta primitives from the red box and note the ‘interpenetration’ effect

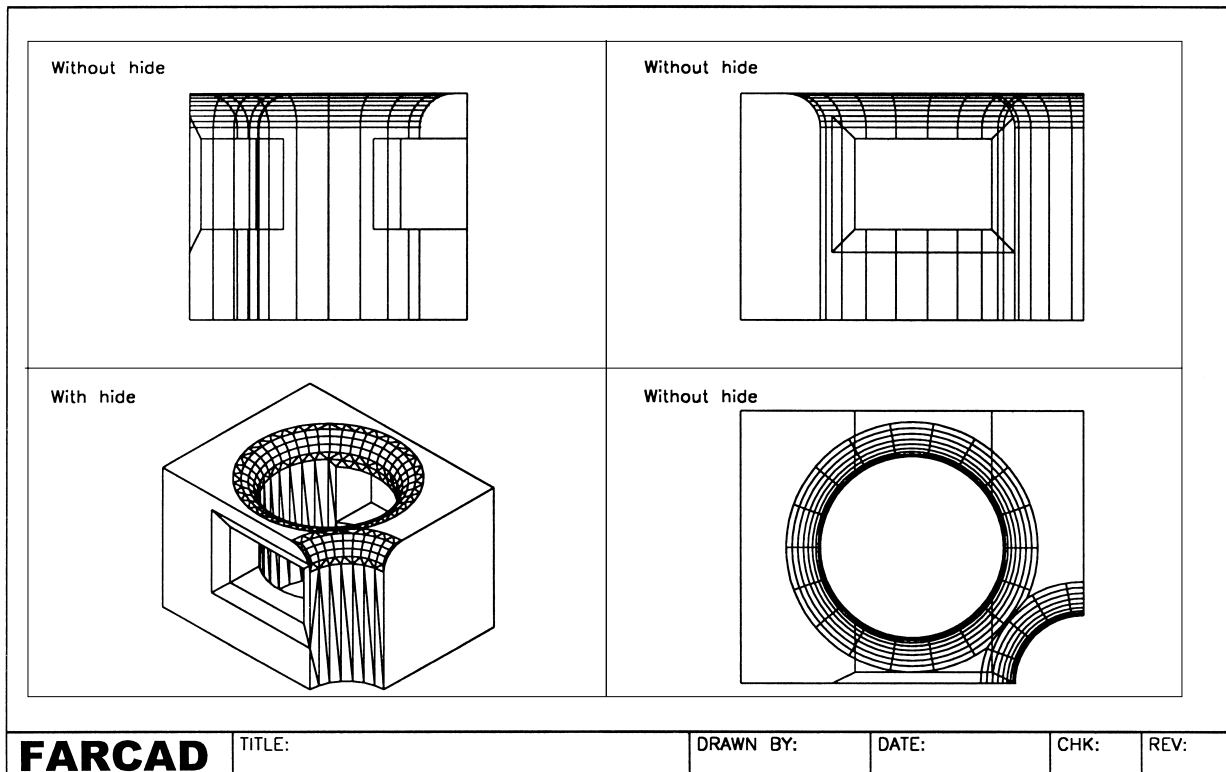


Figure 34.2 Chamfered and filleted composite.

- 4 Select the FILLET icon and:
 - a) pick the top circle of the green object
 - b) enter a radius of 15
 - c) select an edge and right-click/enter
- 5 The top edge of the cylinder is filleted 'outwards' and is red. Why is the fillet red and not green?
- 6 Select the CHAMFER icon and:
 - a) pick the top long front edge of the red box
 - b) press <RETURN> if the front vertical face is highlighted or N <R> until front vertical face is highlighted then right-click/enter
 - c) enter base surface distance of 10
 - d) enter other surface distance of 5
 - e) pick the four front edges of the blue primitive then right-click/enter
- 7 The blue box primitive is chamfered in red – not blue?
- 8 Now fillet the top curve of the magenta cylinder with a radius of 15
- 9 The completed model is displayed as Fig. 34.2 and can be shaded and the 3D orbit command used.
- 10 The model can be saved if required, but will not be used again

A practical use for fillet/chamfer with solids

Before leaving this chapter, we will investigate a practical use of the fillet and chamfer commands with solids so:

- 1 Erase the composite on the screen and ensure UCS BASE, layer MODEL and lower left viewport active. Refer to Fig. 34.3

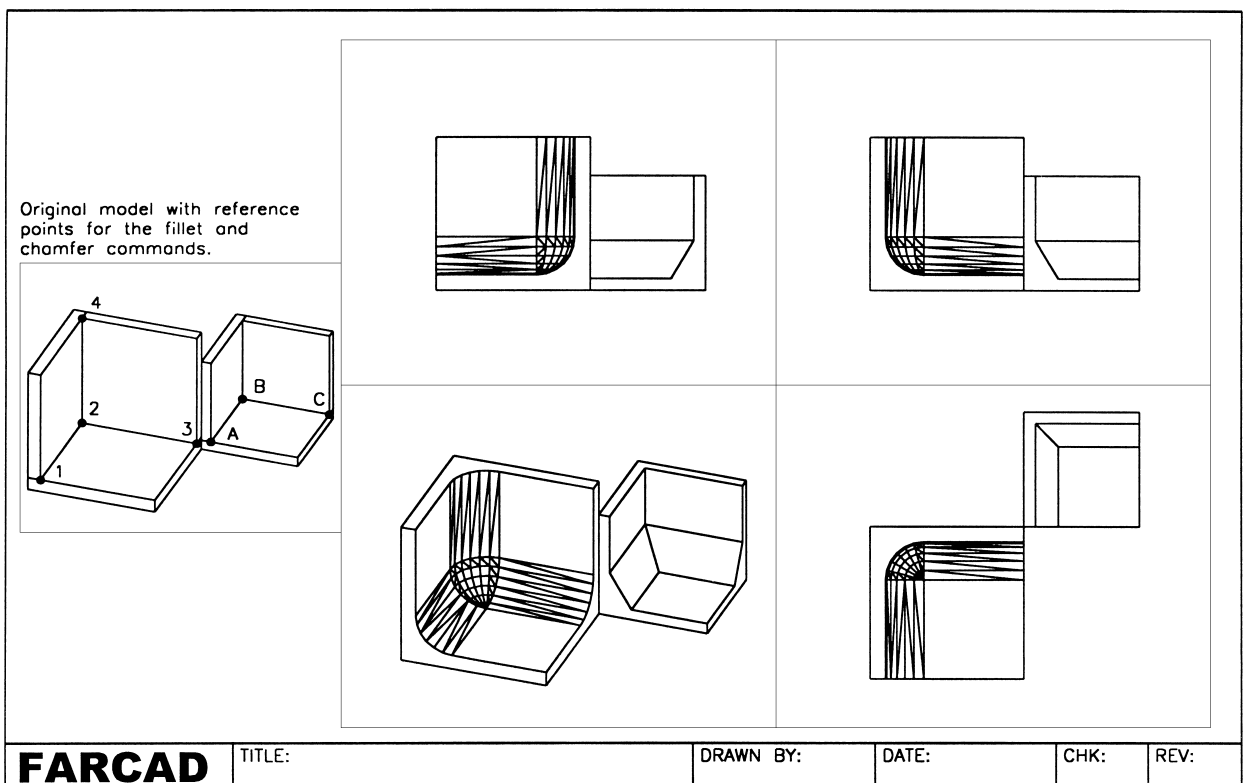


Figure 34.3 Practical use for chamfer/fillet.

- 2 Create the following three box primitives:

	<i>box1</i>	<i>box2</i>	<i>box3</i>
corner	0,0,0	0,0,100	10,100,10
length	100	10	90
width	100	100	-10
height	10	90	90

- 3 Union the three boxes and copy the unioned composite from: 0,0,0 to 100,100,0
- 4 Scale the copied composite about the point 100,100 by 0.75
- 5 Change the viewpoint in the 3D viewport with VPOINT-ROTATE and angles of 300 and 30
- 6 Zoom-centre in each viewport about the point 80,80,50 for 225 mag
- 7 In paper space zoom in on the 3D viewport then model space
- 8 Select the FILLET icon and:
- prompt* Select first object
respond **pick edge 12**
prompt Enter fillet radius and enter: **25 <R>**
prompt Select an edge
respond **pick edges 12,23 and 24 then right-click/enter**
- 9 Select the CHAMFER icon and:
- prompt* Select first line
respond **pick edge AB**
prompt Enter surface selection option
respond a) enter N <R> until horizontal surface is highlighted
 b) right-click/enter
prompt Specify base surface chamfer distance and enter: **15 <R>**
prompt Specify other surface chamfer distance and enter: **25 <R>**
prompt Select an edge
respond **pick edges AB and BC then right-click/enter**
- 10 The two composites will be filleted and chamfered at the selected edges
- 11 *Questions*
- a) Is it possible to fillet more than three edges at the one time?
- b) Can a chamfer be added to three adjacent surfaces?

This completes the edge primitive exercise.

Summary

- 1 Primitives and solids can be chamfered and filleted with the 'normal' CHAMFER and FILLET commands
- 2 Solids and primitives can be chamfered/filleted:
 - a) inwards if a primitive
 - b) outwards if a 'hole'
- 3 Individual edges can be chamfered and filleted
- 4 The chamfer command has a LOOP option allowing a complete surface to be chamfered
- 5 The fillet command has a CHAIN option allowing a complete surface to be filleted
- 6 Error messages will be displayed if the chamfer distances or the fillet radius are too large for the model being modified, and the command line will display:
 - a) Failed to perform blend
 - b) Failure while chamfering/Filleting

Assignment

This activity requires a cube to be chamfered to give a 'truncated pyramid'. The model will be used in a later activity, so ensure that it is saved.

ACTIVITY 20: Penetrated pyramid of MACFARAMUS.

One of MACFARAMUS's model pyramids was unearthed from the desert and basically consisted of three primitives:

- a) a cube of side 200
- b) a cylinder with radius 25 with the centre 100 from the base
- c) a square box of side 80 with the lower edge 30 from the base

You have to create this model and the suggested approach is:

- 1 Position the cube (red) with corner at 0,0,0
- 2 Chamfer the cube to give a square topped pyramid, the top having a side length along the top surface 100
- 3 Position the square (blue) sided box
- 4 Position the cylinder (green) – use other end option
- 5 Subtract the box and cylinder from the pyramid
- 6 Chamfer the blue square box 'front', distances: 10
- 7 Fillet the green cylinder 'right side' with radius 10
- 8 Save the composite as **MODR2002\MODCOMP**.

Solids editing

AutoCAD 2002 allows solid primitives and composites to be edited, the commands being activated from the:

- a) menu bar with Modify-Solids Editing
- b) Solids Editing toolbar

The editing facilities available are:

- 1 Boolean: union, subtraction, intersection
- 2 Faces: extrude, move, offset, delete, rotate, taper, color, copy
- 3 Edges: color, copy
- 4 Body: imprint, clean, separate, shell, check

The Boolean editing features have been used in the creation of the previous composites, and in this chapter we will investigate some of the other editing features.

Solids editing example 1

- 1 Open your SOLA3 template file and refer to Fig. 35.1. Display the Solids, Solids Editing and Object Snap toolbars.

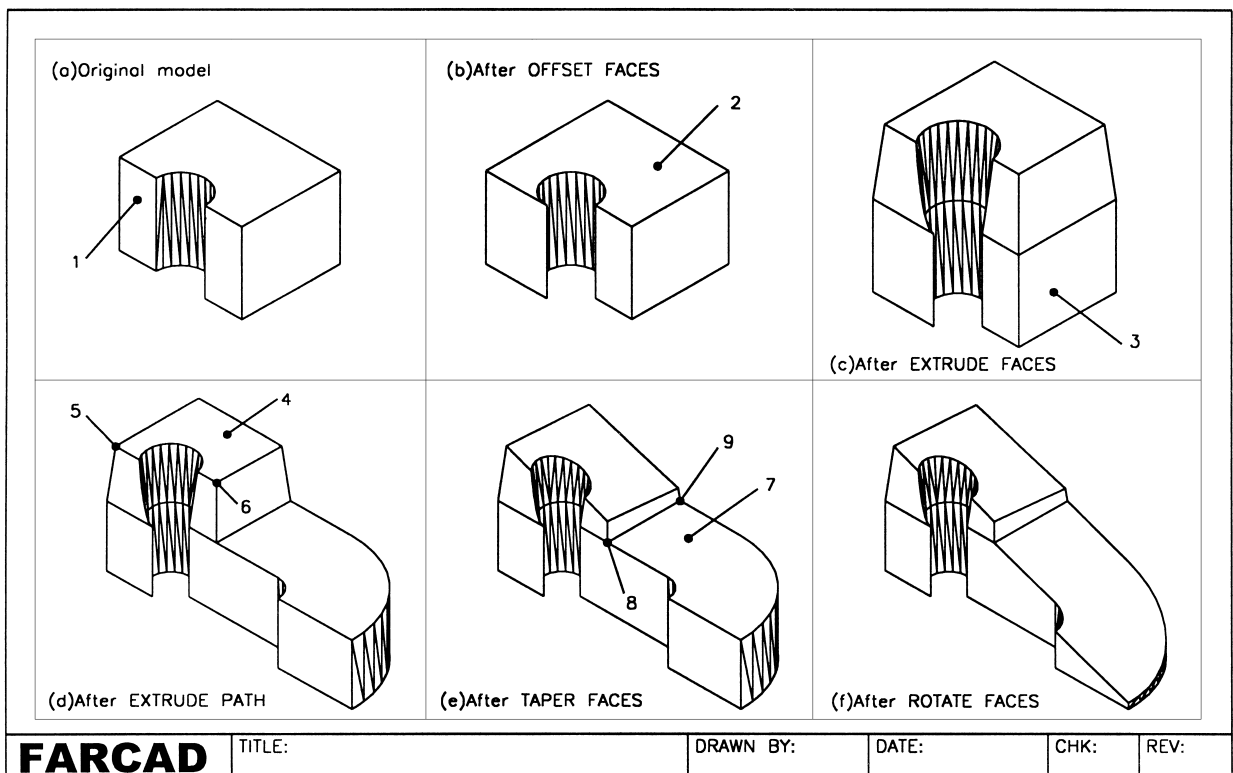


Figure 35.1 Solids editing example 1 – 3D viewport with hide.

- 2 Make the Model tab active and with layer MODEL current, UCS BASE create the following:
- Box: corner at 0,0,0 with L: 150, W: 120, H: 100, colour: red
 - Cylinder: centre at 75,0,0, R: 30, H: 100, colour: blue
 - Subtract the blue cylinder from the red box
 - Pan the composite to suit – fig(a)
- 3 Select the OFFSET FACES icon from the Solids Editing toolbar and:
- prompt* Select faces or [Undo/Remove]
respond **pick any pt1 on side surface then right-click/enter**
prompt Specify the offset distance
enter **30 <R>**
prompt Enter a face editing option and enter: **X <R>**
prompt Enter a solids editing option and enter: **X <R>**
and selected face is offset as fig(b)
- 4 Select the EXTRUDE FACES icon from the Solids Editing toolbar and:
- prompt* Select faces or [Undo/Remove]
respond **pick any pt2 on the top surface then right-click/enter**
prompt Specify height of extrusion or [Path]
enter **80 <R>**
prompt Specify angle of taper for extrusion
enter **5 <R>**
prompt Enter a face editing option and enter: **X <R>**
prompt Enter a solids editing option and enter: **X <R>**
and top surface of model is extruded as fig(c)
note colour and taper of extruded part cylinder
- 5 Draw a polyline using:
 start point: 150,120
 next point: @100,0
 arc option with arc endpoint: @150,-150
 arc endpoint: right-click/enter
- 6 Pan the model to suit
- 7 Menu bar with **Modify-Solids Editing-Extrude** Faces and:
- prompt* Select faces or [Undo/Remove]
respond **pick any pt3 on right face then right-click/enter**
prompt Specify height of extrusion or [Path]
enter **P <R>** – the path option
prompt Select extrusion path
respond **pick any point on polyline**
prompt Enter a face editing option and enter: **X <R>**
prompt Enter a solid editing option and enter: **X <R>**
- 8 The selected face is extruded along the polyline path as fig(d)
- 9 Pan the model to suit
- 10 Select the TAPER FACES icon from the Solids Editing toolbar and:
- prompt* Select faces or [Undo/Remove]
respond **pick any pt4 on top surface then right-click/enter**
prompt Specify the base point
respond **Endpoint icon and pick pt5**
prompt Specify another point along the axis of tapering
respond **Endpoint icon and pick pt6**
prompt Specify the taper angle and enter: **20 <R>**
prompt Enter a face editing option and enter: **X <R>** and **X <R>**



- 11 The selected top surface will be tapered as fig(e)
- 12 Menu bar with **Modify-Solid Editing-Rotate Faces** and:
 - prompt* Select faces or [Undo/Remove]
 - respond* **pick any pt7 on face indicated then right-click/enter**
 - prompt* Specify an axis point or [Axis by object..
 - respond* **Endpoint icon and pick pt8**
 - prompt* Specify the second point on the rotation axis
 - respond* **Endpoint icon and pick pt9**
 - prompt* Specify a rotation angle
 - enter* **20 <R>**
 - prompt* Enter a face editing option and enter: **X <R>** and **X <R>**
- 13 The selected face is rotated about the selected side as fig(f)
- 14 Shade and 3D orbit the model then save if required
- 15 This exercise is now complete.

Solids editing example 2

- 1 Open your SOLA3 template file and refer to Fig. 35.2. Display the Solids, Solids Editing and Object Snap toolbars.
- 2 With Model tab active, layer MODEL current, UCS BASE create the following:
 - a) Box: corner at 0,0,0 with cube of length 200 and colour red
 - b) Cone: centre at 0,100,100; radius: 80; Apex at @300,0 and colour: green
 - c) Subtract the green cone from the red box
 - d) Pan to suit and alter the viewpoint with to suit – fig(a)

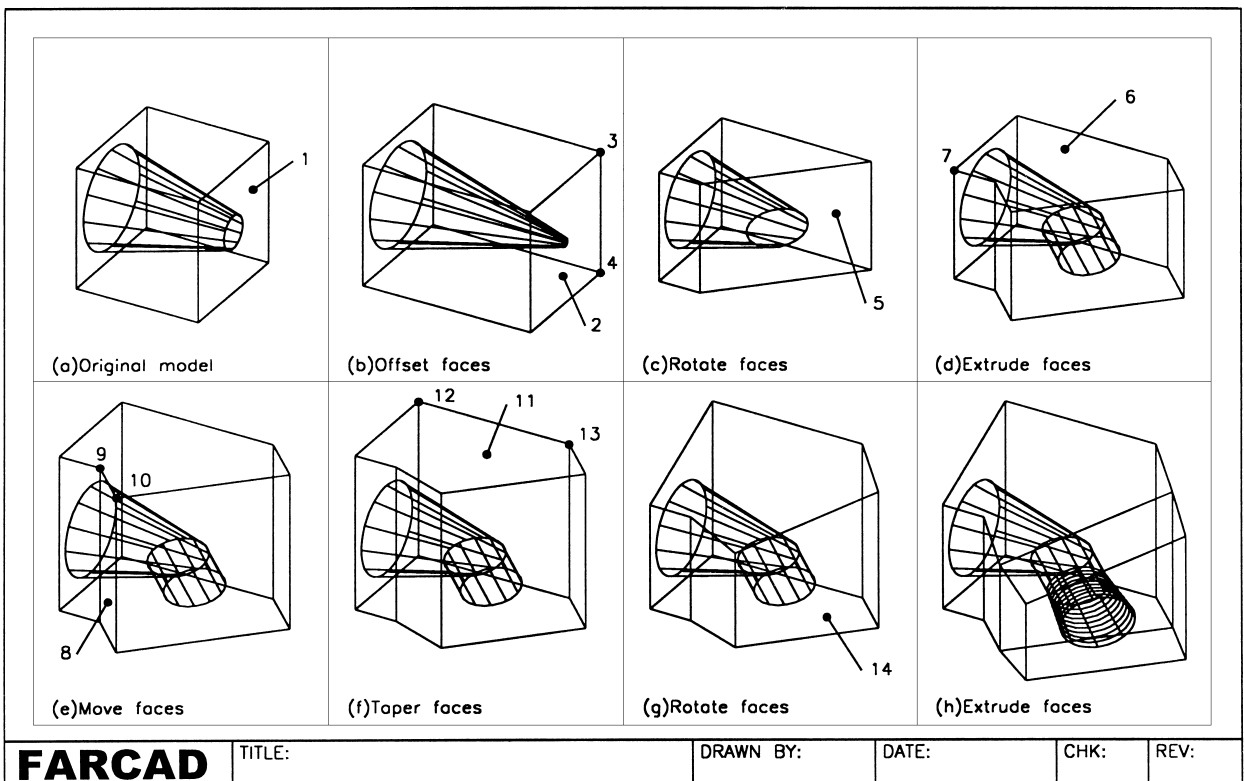


Figure 35.2 Solids editing example 2 – 3D viewport without hide.

- 3 The options for this example are given as a series of simple steps which you should be able to follow:
 1. *Offset faces*
 - a) pick face 1
 - b) distance: 75 – fig(b)
 2. *Rotate faces*
 - a) pick face 2
 - b) axis with endpoints 3 and 4 (in order shown)
 - c) angle 45 – fig(c)
 3. *Extrude faces*
 - a) pick face 5
 - b) height: 100 with taper: 0 – fig(d)
 4. *Move faces*
 - a) pick face 6
 - b) base point at point 7
 - c) second point: @0,0,85 – fig(e)
 5. *Taper faces*
 - a) pick face 8
 - b) axes points 9 and 10
 - c) taper angle: 25 – fig(f)
 6. *Rotate faces*
 - a) pick face 11
 - b) axis with endpoints 12 and 13 (in order shown)
 - c) angle 25 – fig(g)
 7. *Extrude faces*
 - a) pick face 14
 - b) height of extrusion: 120
 - c) taper angle: 5 – fig(h)

This completes the second example.

Note

- 1 The two examples have concentrated on the faces editing options
- 2 It is sometimes difficult to select a face for the solids editing options, and it is sometimes easier to select an edge. As an edge 'belongs' to two adjacent faces, the unwanted face can easily be removed from the 'selection' with the 'R' entry.
- 3 The solids editing command allows the user repetitive options, i.e. when one option has been completed, the command is still active. The UNDO option is very useful.
- 4 The solids editing command is exited with two X <R> or ESC.

Solids editing example 3

- 1 Open your SOLA3 template file and refer to Fig. 35.3. Display suitable toolbars.
- 2 With the Model tab active, layer MODEL current, UCS BASE, create the following:
 - a) Box: corner at 0,0,0 with cube of length 100 and colour red
 - b) Cylinder: centre at 0,0,100; radius: 50; height: 20
 - c) Cylinder: centre at 100,50,100; radius: 30; height: -50
 - d) Subtract the second cylinder from the box
 - e) Zoom and pan to suit – fig(a)
- 3 Select the IMPRINT icon from the Solids Editing toolbar and:

prompt Select a 3D solid
respond **pick the cube**
prompt Select an object to imprint
respond **pick the R50 cylinder**
prompt Delete the source object <N>
enter **Y <R>** then **X <R>** and **X <R>**
- 4 The cylinder outline is imprinted on the cube – fig(b)
- 5 With the EXTRUDE FACES option:
 - a) pick face 1
 - b) enter an extrusion height of 80
 - c) enter a taper angle of 5 – fig(c)

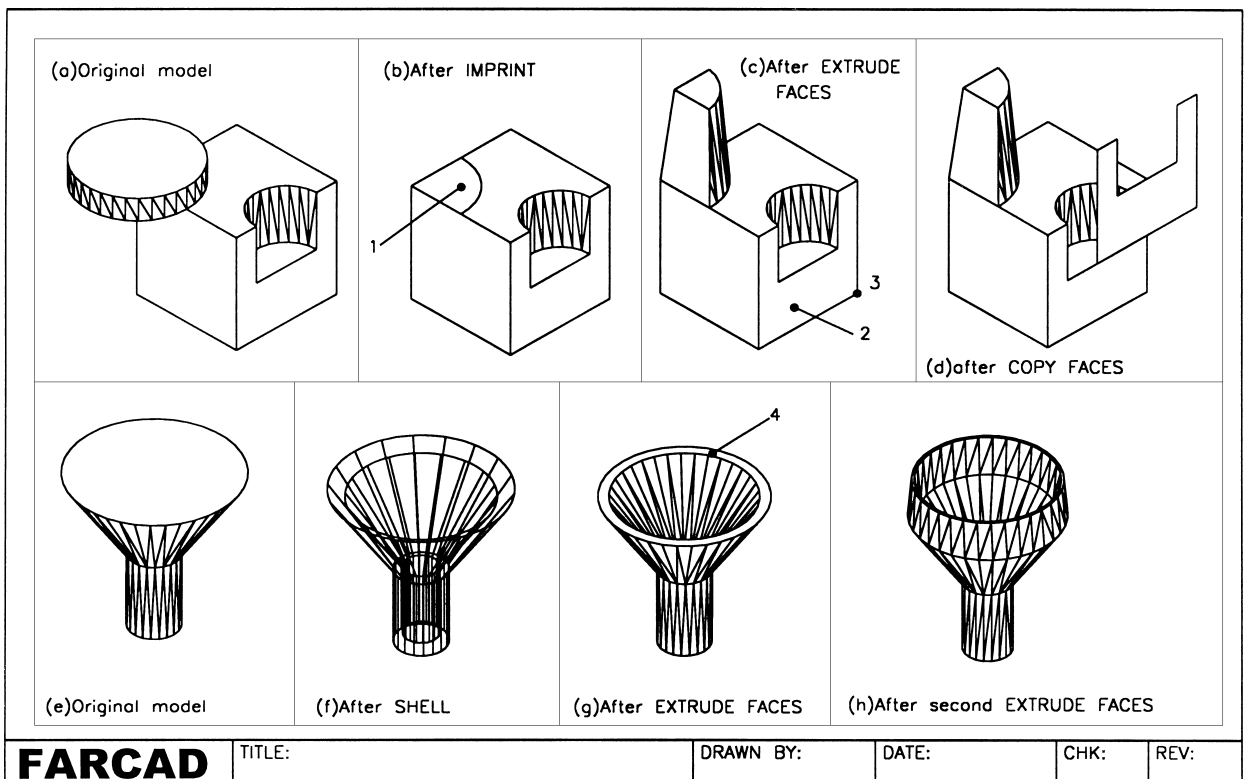


Figure 35.3 Solids editing example 3.

6 Select the COPY FACES icon from the Solids Editing toolbar and:

prompt Select faces
respond **pick face 2 then right-click/enter**
prompt Specify a base point or displacement
respond **Endpoint icon and pick pt3**
prompt Specify a second point of displacement
enter **@50,0,100 <R>** then exit the command

7 The selected face is copied as fig(d)

8 Erase all objects from the screen and create two primitives:

a) cylinder: centre 0,0,0; radius: 30; height: 100
 b) cone: centre: 0,0,200; radius: 100; height: -150
 c) union the cone and cylinder – fig(e)

9 Select the SHELL icon from the Solids Editing toolbar and:

prompt Select a 3D solid
respond **pick any point on the composite then right-click/enter**
prompt Enter the shell offset distance
enter **10<R>** then **X<R>** and **X<R>**

10 The cylinder/cone is offset by 10 in ‘all directions’ – fig(f)

11 Hide the model then regen

12 With the EXTRUDE FACES icon:

a) pick any point on the ‘top face’
 b) extrusion height: -10
 c) angle of taper: 0 – fig(g)

13 Repeat the extrude faces command and:

a) pick any point 4 on the ‘new rim’ of the composite
 b) extrusion height: 50
 c) angle of taper: 5 – fig(h)

This completes the third solids editing example.

Other solids editing options

Not all of the solids editing options have been used in the worked examples. The following is a brief description of those not considered:

- a) Clean: removes all redundant edges and vertices, e.g. imprinted edges
- b) Separate: separates 3D solid objects with disjointed volumes into independent 3D objects. It **DOES NOT** separate composites created by Boolean operations into the original primitives.
- c) Check: confirms that a selected object is a valid ACIS solid
- d) Color Edges/Faces: a very useful option as it allows individual edges and faces to be coloured, i.e. a cube could be created on layer MODEL (red) and the six faces of the cube assigned different colours.
- e) Copy Edges/Faces: should be obvious
- f) Delete faces: allows faces of a model to be deleted, but the option has obvious limitations. Useful with fillet/chamfer edges.

Solids editing errors

When a solids editing option is activated and completed, the command line will display:

- a) Solid validation started
- b) Solid validation completed

Solids editing may not always work due to the model selected or the option which has been activated. Errors which are displayed include:

- 1 No solution for an edge
- 2 No solution for a vertex
- 3 No loop through new edges and vertices
- 4 Could not taper surface as requested
- 5 Improper edge/edge intersection
- 6 Gap cannot be filled

If an error message is obtained, then the active option cannot be performed on the selected object. Try again.

Summary

- 1 Solids editing allows the user several options
- 2 These options can result in very interesting and complex models which may be difficult to achieve from basic primitives
- 3 The solids editing options are divided into four categories:
 - a) Boolean operations
 - b) Face options
 - c) Edge options
 - d) Body options
- 4 Solid editing options which cannot be performed result in an error message being displayed
- 5 The solids editing command allows repetitive entries and has an undo option.

Regions

A region is a *closed* 2D shape created from lines, circles, arcs, polylines, splines, etc. and can be used with the extrude and revolve command to create solid composites. When created, a region has certain characteristics:

- it is a solid of zero thickness
- it is coplaner, i.e. must be created on the one plane
- it consists of **loops** – outer and inner
- the loops must be continuous closed shapes
- every region has one outer loop
- there may be several inner loops
- inner loops must be in the same plane as the outer loop
- regions can be created with the *BOUNDARY* command
- regions can be used with the solid *EXTRUDE* and *REVOLVE* command

Regions allow the user another method for creating solid models and very complex models can be created with regions. They can also be used to apply hatching to models and extract details from models.

Example 1 – a splined shaft

- 1 Open your A3SOL template file as normal and display suitable toolbars
- 2 Refer to Fig. 36.1 (which only displays the 3D viewport) and create the layout from three circles having diameters 120, 40 and 16. The actual layout is your design but use the 0,0 point as indicated.
- 3 Zoom centre about 0,0,40 at 200 magnification – all viewports
- 4 Select the SUBTRACTION icon from the Solids Editing toolbar, pick the largest circle then right-click and:
prompt No solids or regions selected
- 5 Select the REGION icon from the Draw toolbar and:
prompt Select objects
respond **pick all circles then right-click**
prompt 13 loops extracted
13 Regions created



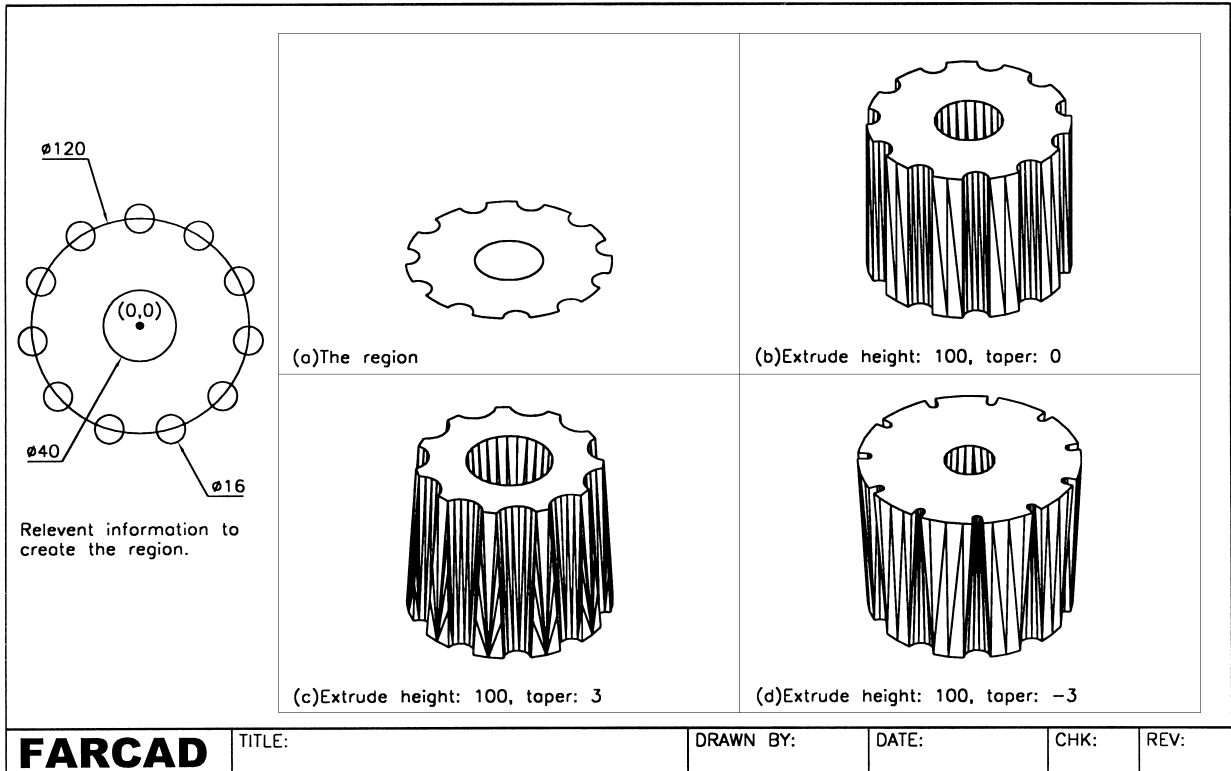


Figure 36.1 Region example 1 – an extruded component.

6 Menu bar with **Modify-Solids Editing-Subtract** and:

prompt Select solids or regions to subtract from
Select objects

respond **pick the largest circle then right-click**

prompt Select solids or regions to subtract
Select objects

respond **pick the 12 smaller circles then right-click**

and the region is created as fig(a)

7 At this stage save as **MODR2002\REGEX** for the next exercise

8 With the lower left viewport active select the EXTRUDE icon from the Solids toolbar and:

a) objects: pick the region

b) height: 100

c) taper angle: 0 – fig(b)

d) hide and shade

9 Undo the hide, shade and extrusion effects, then use the EXTRUDE icon with the following entries:

i) height: 100, taper angle: 3 – fig(c). Undo effect

ii) height: 100, taper angle: -3 – fig(d)

10 This exercise does not need to be saved.

Example 2 – a revolved component

- 1 Open drawing file MODR2002\REGEX saved from the previous exercise with UCS BASE as Fig. 36.2(a)
- 2 Menu bar with **Tools-New UCS-Origin** and:


```
prompt Origin point<0,0,0>
enter -100,-100,0 <R>
```
- 3 Zoom centre about 150,0,0 at 250 magnification
- 4 Select the REVOLVE icon from the Solids toolbar and:


```
prompt Select objects
respond pick the region then right-click
prompt Specify start point for axis of revolution or define axis by..
enter X <R> – the X axis option
prompt Specify angle of revolution
enter -90 <R>
then pan model to suit and hide – fig(b)
```
- 5 Undo the hide and revolve effect to leave the original region
- 6 Using the REVOLVE icon:
 - a) pick the region then right-click
 - b) enter Y as the axis of revolution
 - c) enter 180 as the angle
 - d) pan and hide – fig(c)
 - e) undo the hide and revolve effect

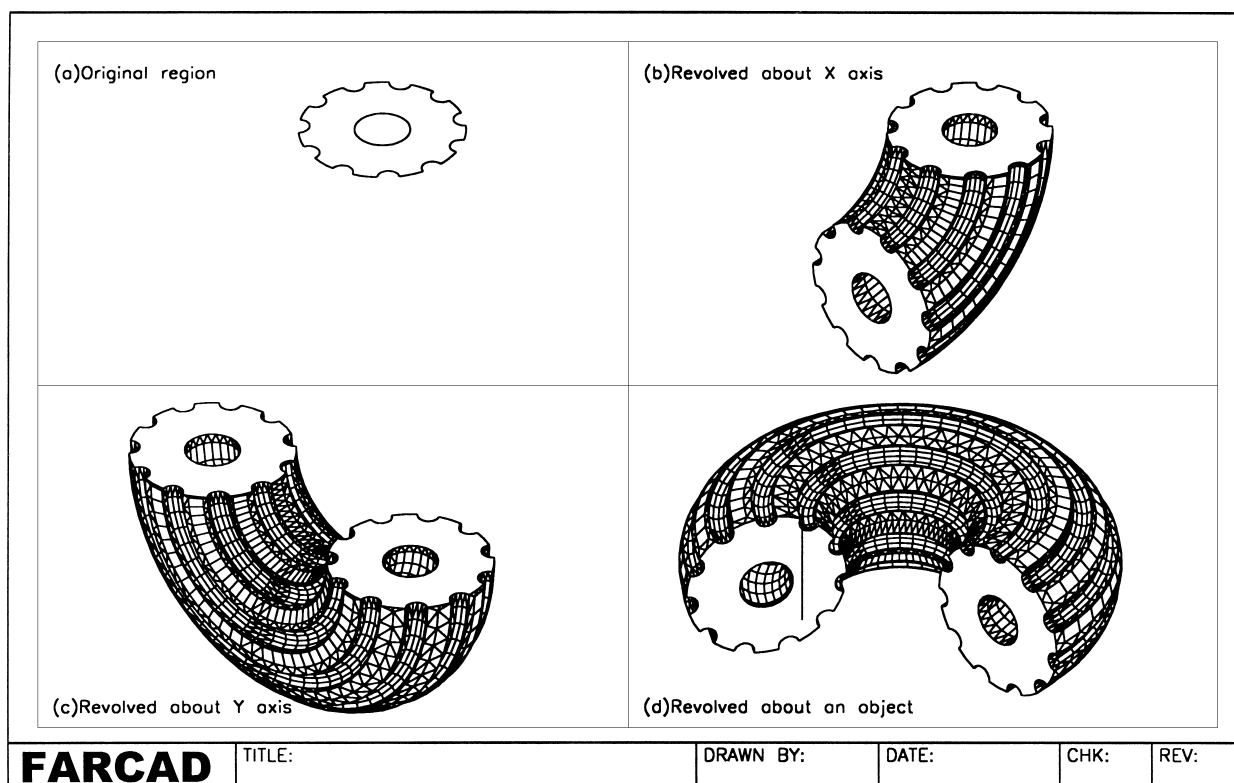


Figure 36.2 Region example 2 – a revolved component.

- 7 Draw a line from: 0,0,0 to: @0,0,100
- 8 Menu bar with **Modify-3D Operation-Rotate 3D** and:
 - a) pick the region then right-click
 - b) enter X <R> as the axis
 - c) enter 100,100,0 as a point on the axis
 - d) enter 90 as the rotation angle
- 9 With the REVOLVE icon:
 - a) pick the rotated region the right-click
 - b) enter **O <R>** – object option
 - c) pick lower end of vertical line
 - d) enter 240 as the angle of revolution
 - e) pan to suit then hide – fig(d)
- 10 This completes the second exercise. Save if required.

Example 3 – using a boundary

- 1 Open your A3SOL template file as normal and refer to Fig. 36.3
- 2 Draw three circles:
 - a) centre: 50,0, radius: 50
 - b) centre: 0,50, radius: 60
 - c) centre: 75,75, radius: 75
- 4 Make a new layer: BND, colour blue and current
- 5 Zoom centre about 50,50,50 at 200 magnification

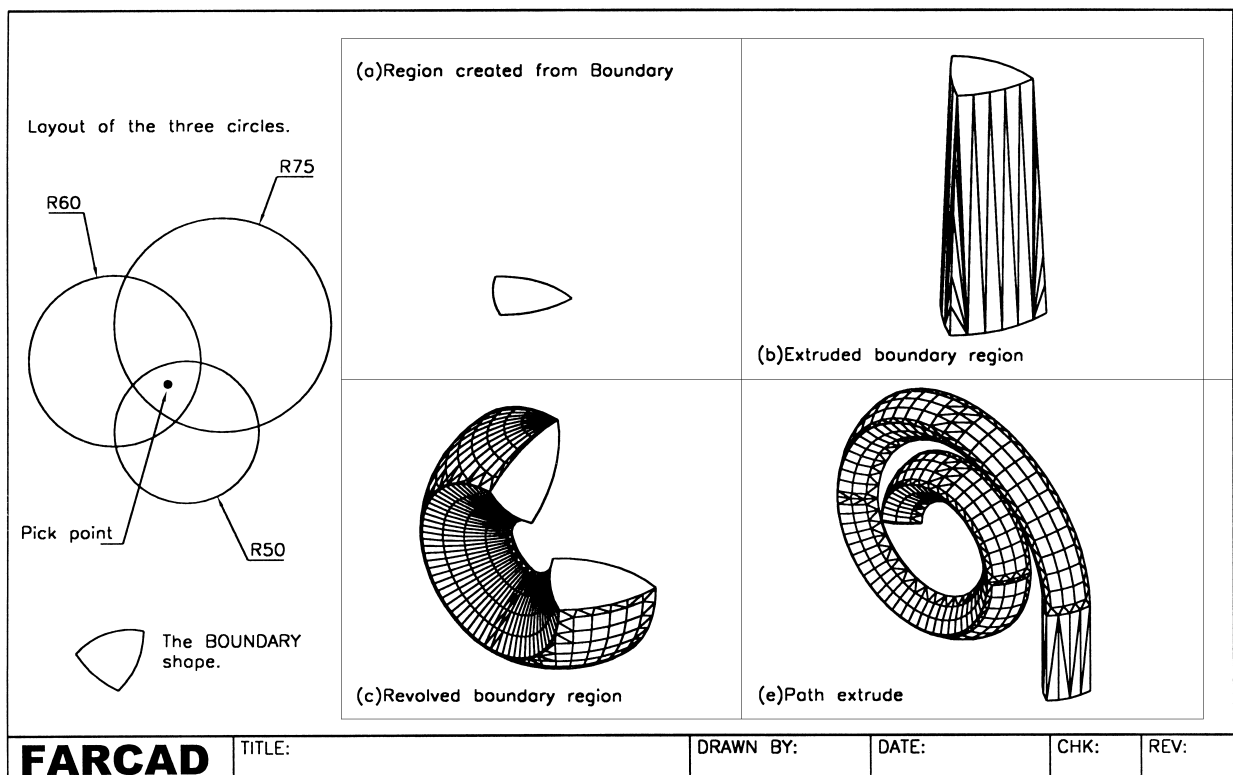


Figure 36.3 Region example 3 – created from a boundary.

- 6 Menu bar with **Draw-Boundary** and:
- prompt* Boundary Creation dialogue box – Fig. 36.4
- respond* 1. pick Object type: Region
2. pick Pick Points
- prompt* Select internal point
- respond* **pick a point indicated in Fig. 36.3**
- prompt* Selecting everything..
Selecting everything visible..
Analyzing the selected data
Analyzing internal islands
- then* Select internal point
- respond* **right-click**
- prompt* 1 loop extracted
1 Region created
BOUNDARY created 1 region
- 7 Erase the three circles to leave the blue region – fig(a)
- 8 Using the EXTRUDE icon:
- a) pick the blue boundary region then right-click
- b) enter a height of 125
- c) enter a taper angle of 2
- d) hide the model – fig(b)
- 9 Undo the hide and extrude effects to leave the blue region
- 10 With the REVOLVE icon:
- a) select the blue region then right-click
- b) enter Y as the axis of revolution
- c) enter 270 as the angle of revolution
- d) hide – fig(c)
- 11 Undo the hide and revolve effect
- 12 Restore UCS FRONT, and with layer MODEL current draw a polyline with:
start point: 0,0
next point: @0,100
next point: arc option with endpoint: @-200,0
next point: arc endpoint: @120,0
next point: arc endpoint: @-60,0 then right-click

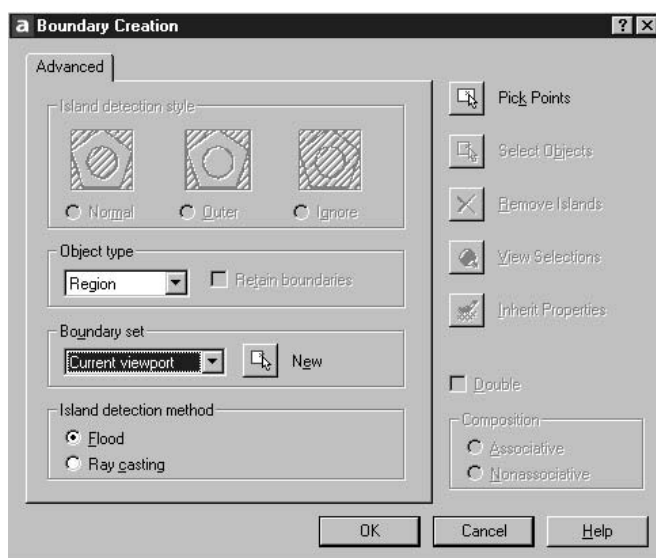


Figure 36.4 The Boundary Creation dialogue box.

- 13 Restore UCS BASE and make layer BND current
- 14 With the EXTRUDE icon:
 - a) select the blue region
 - b) enter **P <R>** for the path option
 - c) pick the red polyline
 - d) pan to suit
 - d) hide to give fig(d)
- 15 The exercise is complete, save?
- 16 *Note*
Although this exercise has been completed using the Boundary command, it could have also been completed by making the three circles into regions and then using the Boolean intersection command.

Summary

- 1 A region is created from closed shapes, e.g. polylines, arcs, circles, ellipses etc
- 2 Regions can be created with the BOUNDARY command
- 3 Regions consist of loops and all regions must have an outer loop. There can be several inner loops.
- 4 Regions can be extruded and revolved
- 5 All parts of a region are extruded/revolved to the same height or angle
- 6 Regions are extruded along the Z-axis of the current UCS. The height of the extrusion can be positive or negative
- 7 Regions can be extruded along a path

Assignment

This activity requires a region to be created from circles, copied, scaled and then extruded to different heights.

ACTIVITY 21: Ratchet mechanism of Macfaramus

While digging in a water bed outside the city of CADOPOLIS, a device was discovered which was thought to be a ratchet mechanism for a primitive type of waterwheel. This mechanism has three distinct 'parts' to it, each having the same shape.

Using the reference sizes given, create the outline from circles, convert these circles into regions, then subtract the smaller circles from the larger. The other parts of the mechanism are scaled by 0.85 and 0.55 from the original. When the three parts have been created, they have to be:

- a) lower part: extruded to a height of -50 with -15 taper
- b) middle part: extruded to a height of 60 with 0 taper
- c) top part: revolved about a line object for -90 degrees and then array for 3 items about the top 'circle' centre. The line for the object can be drawn at your discretion. The line I used is indicated in the activity drawing.

Notes:

- 1 Use a 0,0 centre point for the large circle and for copying and scaling purposes
- 2 Each scaled part is positioned on top of the previous part
- 3 Use your discretion for sizes not given

Inquiring into models

In this chapter we will create two new composites and then use the AutoCAD inquiry commands to determine the properties of these solids. We will also investigate how to create a material properties file.

Composite model 4 – a slip block

- 1 Open the A3SOL template file with normal settings, display the Inquiry toolbar and refer to Fig. 37.1
- 2 Create the following primitives:

<i>Box</i>	<i>Wedge</i>
corner: 0,0,0	corner: 120,0,0
length: 120	length: 50
width: 100	width: 100
height: 100	height: 70
colour: red	colour: blue
- 3 Centre the model about 80,50,50 at 200 magnification

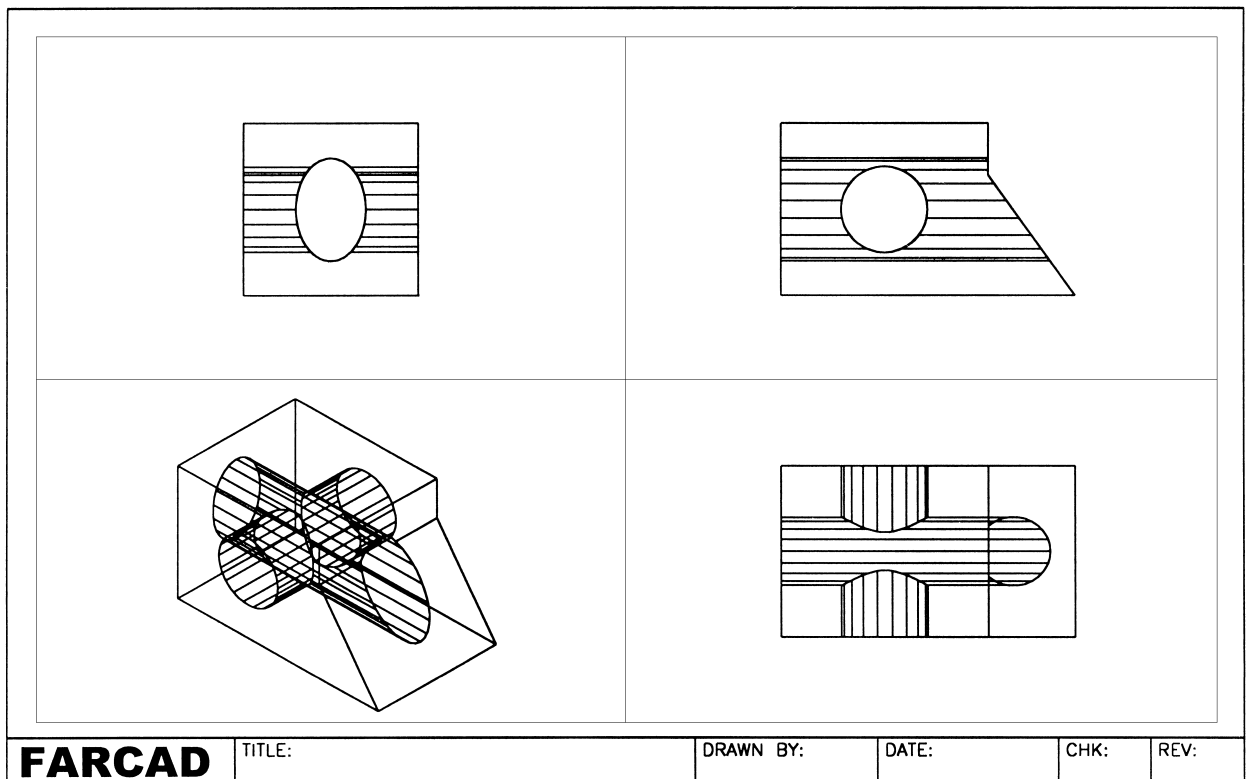


Figure 37.1 Composite model 4 – a slip block (plotted without hide).

- 4 Create two green cylindrical primitives with:
 - a) centre: 60,0,50
radius: 25
centre of other end option: @0,100
 - b) **elliptical** option
centre of ellipse: 0,50,50
axis endpoint: @20,0
length of other axis: @0,0,30
centre of other end: @180,0
- 5
 - a) union the red box and blue wedge
 - b) subtract the two green cylinders from the composite
 - c) note the 'curves of interpenetration'
 - d) shade and note the colour effect
 - e) use the 3D orbit command with the model, then restore the original 2D wireframe representation.
- 6 At this stage save the composite as **MODR2002\SLIPBL**
- 7 Select the AREA icon from the Inquiry toolbar and:


```
prompt Specify first corner point or [Object/Add/Subtract]
enter O <R> – the object option
prompt Select objects
respond pick the composite
prompt Area = 98062.84, Perimeter = 0.00
```
- 9 The area value displayed is the surface area of the composite in square units (mm?). A solid object has no perimeter, hence the 0 value
- 9 Select the REGION/MASS PROPERTIES icon from the Inquiry toolbar and:


```
prompt Select objects
respond pick the composite then right-click
prompt AutoCAD Text Window
with details about the model including:
      Mass = 995882.69
      Volume = 995882.69
prompt Write analysis to a file?<N>
enter Y <R>
prompt Create Mass and Area Properties File dialogue box
respond 1. check – Save in: named folder active
        2. check – Save as type *.mpr – materials properties
        3. enter File name: SLIPBL
        4. pick Save (more on properties file later)
```
- 10 Note that the mass and volume values are the same as AutoCAD 2002 assumes a density of 1.

Composite model 5 – a casting block

- 1 Close and existing drawings then open A3SOL template file with the normal settings and:
 - a) refer to Fig. 37.2
 - b) zoom centre about 37,50,18 at 150 magnification in all viewports
- 2 With the 3D viewport active, create a box primitive with:
 - a) corner: 0,0,0
 - b) length: 75, width: 100, height: 36

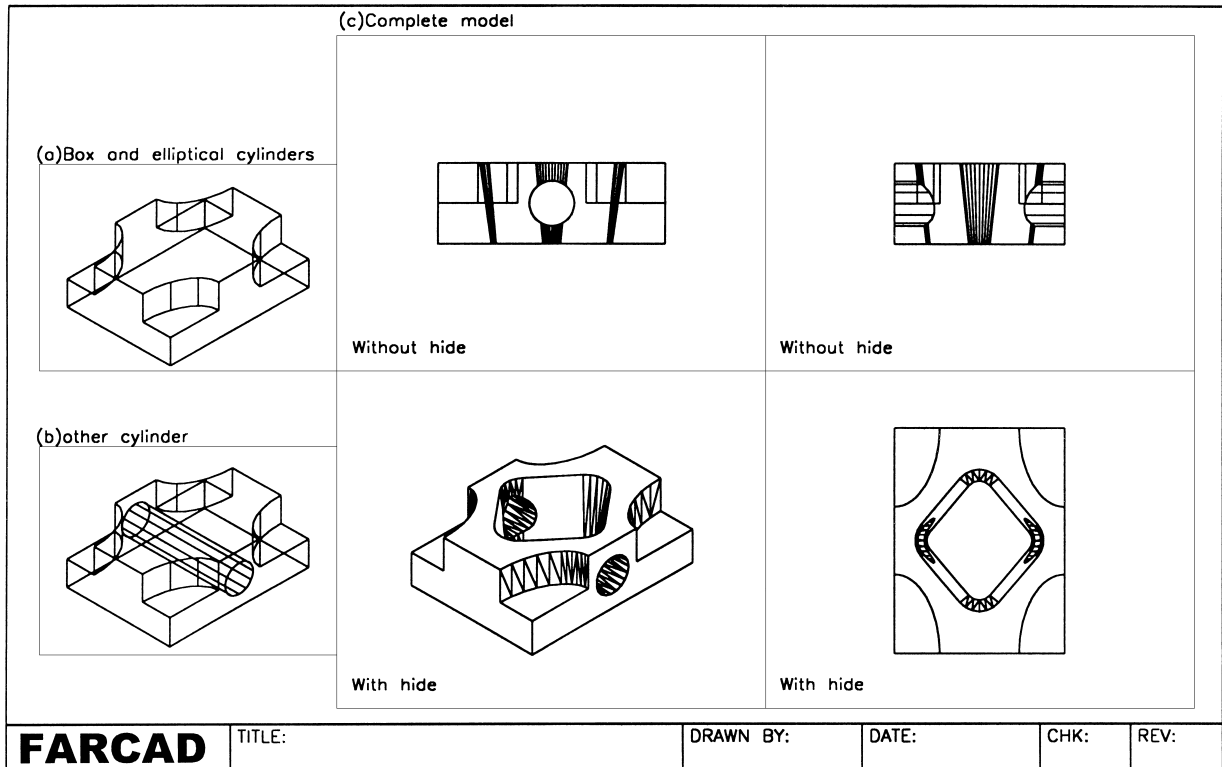


Figure 37.2 Composite model 5 – a casting block.

- 3 Create a **elliptical** cylinder with:
 - a) centre: 0,0,36 (remember to enter C for this option)
 - b) axis endpoint: @20,0
 - c) length of other axis: @0,35
 - d) height: -18
- 4 Rectangular array the elliptical cylinder:
 - a) 2 rows and 2 columns
 - b) row distance: 100, column distance: 75
- 5 Subtract the four cylinders from the box – fig(a)
- 6 Create another cylinder with:
 - a) centre: 0,50,18
 - b) radius: 10
 - c) centre of other end option: @75,0,0
- 7 Subtract the cylinder from the composite – fig(b)
- 8 a) Draw a polyline:
 - start: 0,0 next: 25,30 next: 0,60 next: -25,30 to: close
 - b) Fillet the polyline with a radius of 6
 - c) Move the polyline from: 0,0 by: @37.5,20
 - d) Extrude the polyline for a height of 36 with -8 taper
- 9 Complete the model by subtracting the extruded polyline from the box composite – fig(c)
- 10 Save the model as **MODR2002\CASTBL**

11 Menu bar with **Format-Units** and using the Drawing Units dialogue box alter:

- a) Length type: Engineering
- b) Precision: 0'-0.00"
- c) Insert units: inches
- d) pick OK

12 Select the AREA icon from the Inquiry toolbar and:

- a) use the object option and pick the composite
- b) Area = 29416.67 square in (204.2824 square ft)

13 Select the MASS PROPERTIES icon, pick the composite and:

```
prompt  AutoCAD Text Window
with    Mass    155027.84 lb
        Volume 155027.84 cu in

respond <RETURN> to the prompt
prompt  Write Analysis to a file?<N>
enter   Y <R>
prompt  Create Mass and Area Properties File dialogue box
with    SLIPBL listed from previous exercise
respond 1. enter file name: CASTBL
        2. pick Save
```

14 *Investigate*

a) The area and mass properties have been 'calculated' with the UCS BASE. There are two other UCS saved positions, FRONT and RIGHT. Using these saved UCS's, determine the area and mass/ volume values with engineering units.

My values were:

UCS	Area(sq in)	Mass (lb)/Volume (cu in)
BASE	29416.67	155027.84
FRONT	29416.67	155029.90
RIGHT	29416.67	155027.59

These slight variations in mass/volume are due to the way in which AutoCAD 2002 performs the various calculations.

b) Using the relevant commands, find the area and mass for the composite models created in previous chapters. My values with UCS BASE and decimal units were:

Model/chap	Area	Mass
Machine support(31)	81270.75	1022333.68
Backing plate(32)	38171.70	98283.83
Pipe/flange(33)	190938.19	1305275.35

Mass Properties file

When the mass properties command is used with a solid model, the AutoCAD Text Window will display 'technical' information about the model including the mass and volume.

The user has the option of saving this information to a Mass Property file with the extension **.mpr**. As the mass property file is a 'text file' it can be opened in any 'text editor' type package.

To demonstrate this:

1 Select the Windows Start icon from the Windows bar at the bottom of the screen then select **Programs-Accessories-Notepad** and:

```
prompt  Untitled Notepad screen
respond 1. menu bar with File-Open
        2. at Look in, scroll and to named folder
        3. alter file name to *.mpr then <R>
        4. pick SLIPBL
        5. pick Open
```

2 The screen will display the saved material property file for the slip block composite – Fig. 37.3(a)

- 3 This file can then be printed if required
- 4 Fig. 37.3 also displays the material property file for the casting block – fig(b)
- 5 Exit Notepad to return to AutoCAD
- 6 *Task:* can you import the two mpr files into an AutoCAD drawing?

The exercise is now complete.

```
(a)Material Properties file: SLIPBL.mpr

----- SOLIDS -----

Mass:                995882.69
Volume:              995882.69
Bounding box:        X: 0.00  --  170.00
                    Y: 0.00  --  100.00
                    Z: 0.00  --  100.00
Centroid:            X: 71.51
                    Y: 50.00
                    Z: 45.60
Moments of inertia:  X: 6600251050.50
                    Y: 10169084802.67
                    Z: 10478010529.73
Products of inertia: XY: 3560930335.79
                    YZ: 2270747372.92
                    ZX: 2944997049.96
Radii of gyration:   X: 81.41
                    Y: 101.05
                    Z: 102.57
Principal moments and X-Y-Z directions about centroid:
                    I: 1943214694.71 along [0.95 0.00 -0.30]
                    J: 3004980586.96 along [0.00 1.00 0.00]
                    K: 2991529207.67 along [0.30 0.00 0.95]

(b)Material Properties file: CASTBL.mpr

----- SOLIDS -----

Mass:                155027.84 lb
Volume:              155027.84 cu in
Bounding box:        X: 0.00  --  75.00 in
                    Y: 0.00  --  100.00 in
                    Z: 0.00  --  36.00 in
Centroid:            X: 37.50 in
                    Y: 50.00 in
                    Z: 15.15 in
Moments of inertia:  X: 599835677.34 lb sq in
                    Y: 344913627.59 lb sq in
                    Z: 841241799.96 lb sq in
Products of inertia: XY: 290677192.05 lb sq in
                    YZ: 117437118.81 lb sq in
                    ZX: 88077839.11 lb sq in
Radii of gyration:   X: 62.20 in
                    Y: 47.17 in
                    Z: 73.66 in
Principal moments (lb sq in) and X-Y-Z directions about centroid:
                    I: 176681563.73 along [1.00 0.00 0.00]
                    J: 91321212.53 along [0.00 1.00 0.00]
                    K: 235664305.14 along [0.00 0.00 1.00]
```

Figure 37.3 Material properties files from Notepad.

Summary

- 1 Mass properties can be obtained from solid models
- 2 The mass properties include mass, volume, centroid, radius of gyration etc.
- 3 The mass properties can be saved to an mpr text file
- 4 AutoCAD 2002 does not support a materials library and hence mass and volume are always the same, as density is assumed to be 1.

Assignment

MACFARAMUS designed many different types of objects including garden walls and paths. It is one of these 'garden blocks' that you now have to create as a solid model. The block was one of many used in the design of the famous walled gardens of CADOPOLIS and the block was made from the mud of the river CLYDEBER on whose banks CADOPOLIS was built.

Activity 22: Garden block of MACFARAMUS

The creation of the garden block is straightforward:

- 1 Open you template file as normal
- 2 Using the reference sizes in the activity drawing, create the model from primitives or regions – your choice
- 3 When the model is complete, save as **MODR2002\GARDBL** as it will be used in a later chapter
- 4 *Task*
 - a) when the model is complete, obtain the mass
 - b) MACFARAMUS obtained a quantity of mud from the banks of the river CLYDEBER for making the blocks. If the mass of this mud was 1000000 units, how many blocks could he make (assuming no waste)
 - c) I worked out he could create 5748 blocks.

Slicing and sectioning solid models

Solid models can be sliced (cut) and sectioned relative to:

- a) the XY, YZ and ZX coordinate planes
- b) three points defined by the user
- c) the viewing plane of the current viewpoint
- d) user-defined objects


The two commands are very similar in operation and when used:

- a) the slice command results in a new composite model. This model retains the layer and colour properties of the original solid
- b) the section command adds a 2D region to the model. The region is displayed on the current layer **BUT NO HATCHING IS ADDED TO THE REGION.**

The two commands will be demonstrated using previously created composite models.

Slice example 1 – using the three coordinate planes

- 1 Open the composite model MODR2002\SLIPBL created in the last chapter.
- 2 Make the Model tab active with UCS BASE and layer Model current with UCS BASE, layer
- 3 Select the SLICE icon from the Solids toolbar and:



```

prompt Select objects
respond pick the composite then right-click
prompt Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3 point]
enter XY <R> – the XY plane option
prompt Specify a point on the XY plane<0,0,0>
enter 0,0,50 <R> – why these coordinates?
prompt Specify a point on desired side of plane or [keep Both sides]
enter B <R> – both sides option
and an XY slicing plane added to the model

```
- 4 Erase that part of the model above the slicing plane to give the Fig. 38.1(a) effect
- 5 Undo the erase and slice commands to restore the original model
- 6 Repeat the SLICE command and:
 - a) pick the model then right-click
 - b) enter XY as the slicing plane
 - c) enter 0,0,50 as a point on the plane
 - d) enter @0,0,-10 as a point on the desired side of the plane
 - e) effect is the same as step 3, i.e. fig(a)

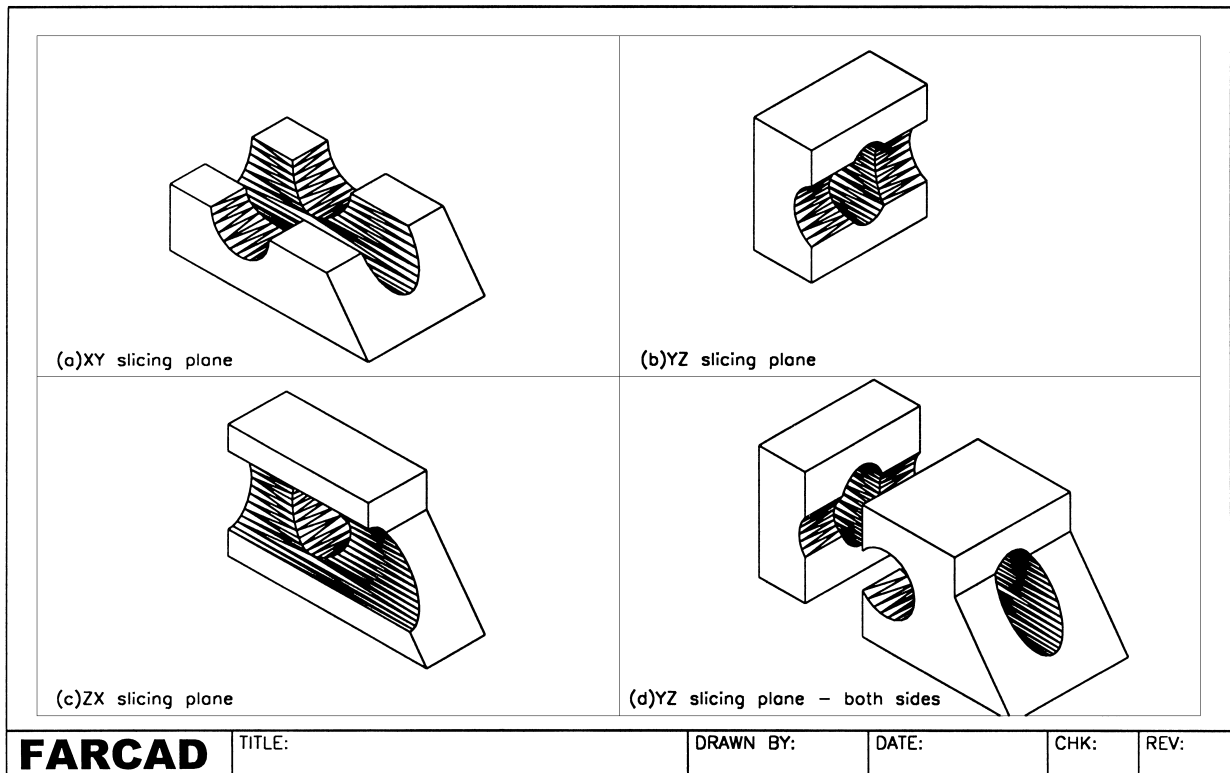


Figure 38.1 Slice example 1 – using the slicing planes with model SLIPBL.

- 7 With the slice command, the user can either:
 - a) specify a point on the desired side of the slicing plane to keep
 - b) keep both models on either side of the slicing plane then erase or move one of them
- 8 Now undo the slice effect or re-open SLIPBL
- 9 Menu bar with Draw-Solids-Slice and:
 - a) pick the composite then right-click
 - b) enter YZ <R> as the slicing plane
 - c) enter 50,0,0 <R> as a point on the plane
 - d) enter @-10,0,0 <R> to keep part 'to left' of slice plane
 - e) the new composite is created as fig(b)
- 10 Undo the slice effect
- 11 At the command line enter **SLICE <R>** and:
 - a) pick the composite then right-click
 - b) enter ZX as the slicing plane
 - c) enter 0,50,0 as a point on the plane
 - d) enter @0,10,0 to keep part 'below' the slice plane
 - e) new composite as fig(c)
- 12 Undo the slice effect, activate the slice command and:
 - a) pick the composite then right-click
 - b) select the YZ plane
 - c) enter 40,0,0 as a point on the plane
 - d) enter **B <R>** to keep both sides
 - e) model is now 'sliced' into two new composites

- 13 With the MOVE command:
- pick the blue wedge then right-click
 - base point: enter 170,0
 - second point of displacement: enter @50,0
 - the two new composites are separated – fig(d)
- 14 This completes the first exercise which does not need to be saved

Slice example 2 – using user-defined slicing planes

- Open the drawing file MODR2002\MACHSUPP – the first composite solid created in chapter 31. Make the Model tab active with USC BASE and layer MODEL. Refer to Fig. 38.2
- Activate the SLICE command and:

prompt Select objects
respond **pick the composite then right-click**
prompt Specify first point on slicing plane ..
respond **right-click/enter** – accepting the 3 point default
prompt Specify first point on plane and: **ENDpoint icon and pick pt1**
prompt Specify second point on plane and: **ENDpoint icon and pick pt2**
prompt Specify third point on plane and: **QUAdrant icon and pick pt3**
prompt Specify a point on desired side of plane..
enter **@0,0,-10 <R>** – why this entry?
- A new composite will be created as fig(a). This has been ‘sliced’ through an inclined plane.

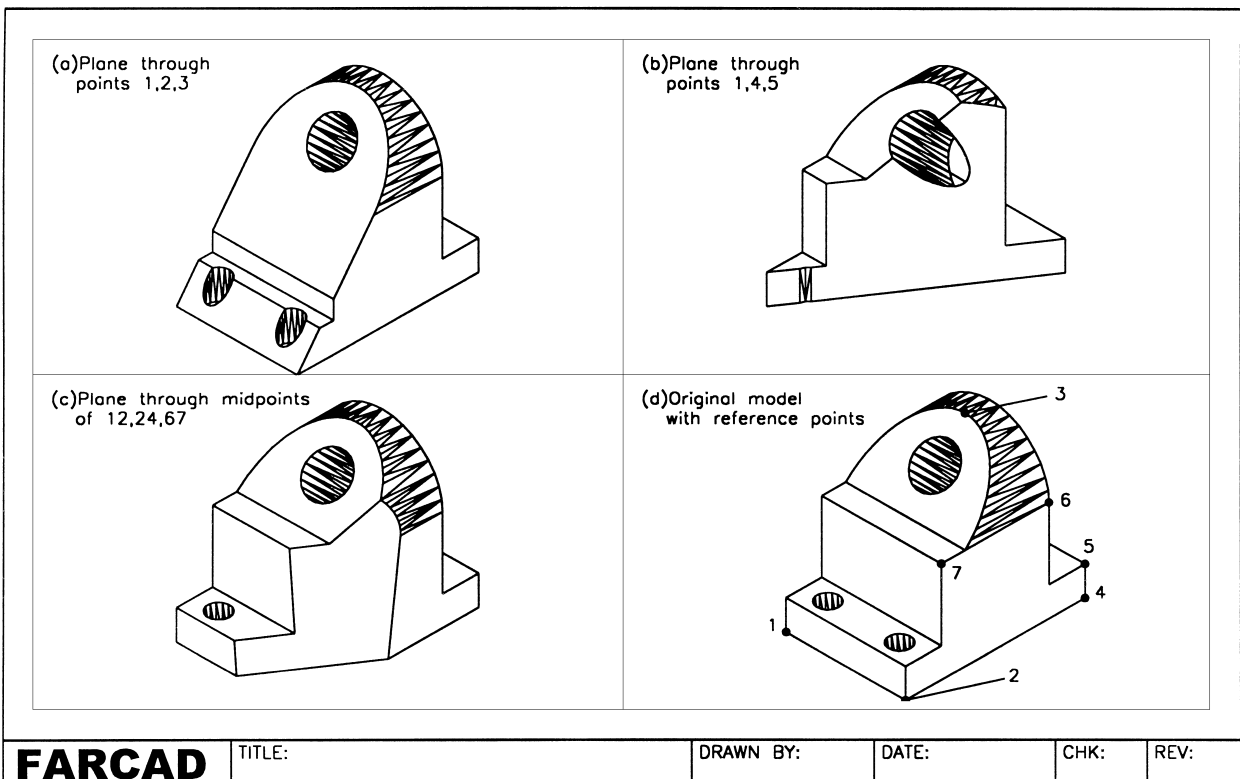


Figure 38.2 Slice example 2 – MABHSUPP using three-defined points.

- 4 Hide and shade the model
- 5 Undo the shade, hide and slice commands to leave the original composite – or re-open MACHSUPP.
- 6 With the SLICE command:
 - a) pick the composite then right-click
 - b) enter <R> to activate the 3 point option
 - c) first point on plane: ENDpoint and pick pt1
 - d) second point on plane: ENDpoint and pick pt4
 - e) third point on plane: ENDpoint and pick pt5
 - f) point on plane: enter @0,10,0
 - g) hide the new composite – fig(b)
- 7 This new composite has been sliced through a vertical plane from corner to corner.
- 8 Undo the hide and slice effects
- 9 Use the SLICE command on the composite, activate the 3 point option and:
 - a) first point on plane: MIDpoint of line 12
 - b) second point on plane: MIDpoint of line 24
 - c) third point on plane: MIDpoint of line 67
 - d) point on plane: enter: @-10,0,0
 - e) new composite displayed as fig(c)
 - f) hide and shade
- 10 The exercise is now complete.

Slice example 3 – view and object options

- 1 Open the drawing file MODR2002\FLPIPE of the flange-pipe composite from chapter 33
- 2 MVLAY1 tab active with UCS BASE and layer MODEL.
- 3 Refer to Fig. 38.3 and set the following 3D Views:

<i>viewport</i>	<i>3D View</i>
top left	NW Isometric
top right	NE Isometric
bottom right	SW Isometric
bottom left	SE Isometric – should be set to this
- 4 Re-centre about -100,-30,0 at 300 mag and pan to suit
- 5 With the top left viewport active, select the SLICE icon and:


```

prompt Select objects
respond pick the composite then right-click
prompt Specify first point on slicing plane or..
enter V <R> – the view option
prompt Specify a point on the current view plane<0,0,0>
enter 0,50,0 <R>
prompt Specify a point on desired side of plane..
enter 0,-10,0 <R> i.e. keep part 'behind' the plane
and new composite displayed as fig(a)
then hide the model
      
```
- 6 *Note:* the view option assumes a 'line of sight' which is perpendicular to the view plane
- 7 Undo the hide and slice effects to restore the original model

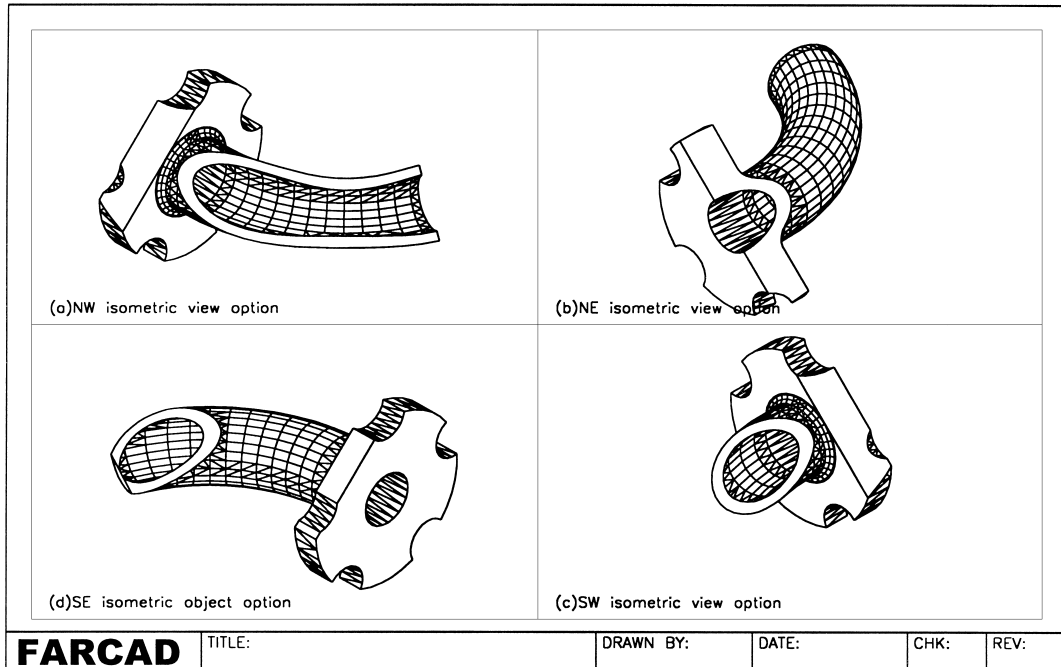


Figure 38.3 Slice example 3 – FLPIP model with view and object options.


- 8 With the top right viewport active, SLICE and:
 - a) pick the composite then right-click
 - b) enter V <R> for the view option
 - c) enter 0,50,0 <R> as a point on the view plane
 - d) enter 0,-10,0 <R> as a point on the desired side on the plane
 - e) new composite created
 - f) hide the model – fig(b)
- 9 Undo the hide and slice effects
- 10 Make the lower right viewport active and slice with:
 - a) the composite
 - b) the view option
 - c) 0,-50,0 as a point on the plane
 - d) 0,10,0 as a point on the desired side of the plane
 - e) new composite as fig(c)
 - f) hide then undo both the hide and slice effects
- 11 Finally with the lower left viewport active:
 - a) draw a circle with centre: 0,-100 and radius: 100
 - b) rotate 3D this circle:
 1. about the X axis
 2. with the circle centre as a point on the axis
 3. for a reference angle of 45 degrees
 - c) with the SLICE command:
 1. pick the composite then right click
 2. enter O <R> for the object option
 3. pick the circle as the object
 4. enter 0,10,0 as a point on desired side
 5. erase the circle then hide and composite as fig(d)

The slice examples have now been completed.

Section example – three point option

The section command is very similar in operation to the slice command, and only one example will be demonstrated

- 1 Open the drawing file MODR2000\SLIPBL from chapter 37, MVLAY1 tab, UCS BASE, layer MODEL and the lower left viewport active. Refer to Fig. 38.4
- 2 Make layer SECT current – or make a new section layer
- 3 Select the SECTION icon from the Solids toolbar and:



```

prompt Select objects
respond pick the composite then right-click
prompt Specify first point on section plane by..
respond <R> – to activate the three point option
prompt Specify first point on plane and ENDpoint of pt1
prompt Specify second point on plane ENDpoint of pt2
prompt Specify third point on plane pick ENDpoint of pt3
      
```
- 4 A coloured outline as added to the model
- 5 Menu bar with **Tools-Inquiry-List** and:


```

prompt Select objects
respond pick any point on the added outline then right-click
prompt AutoCAD Text Window
with details about the selected object and it is a REGION
respond cancel the text window
      
```

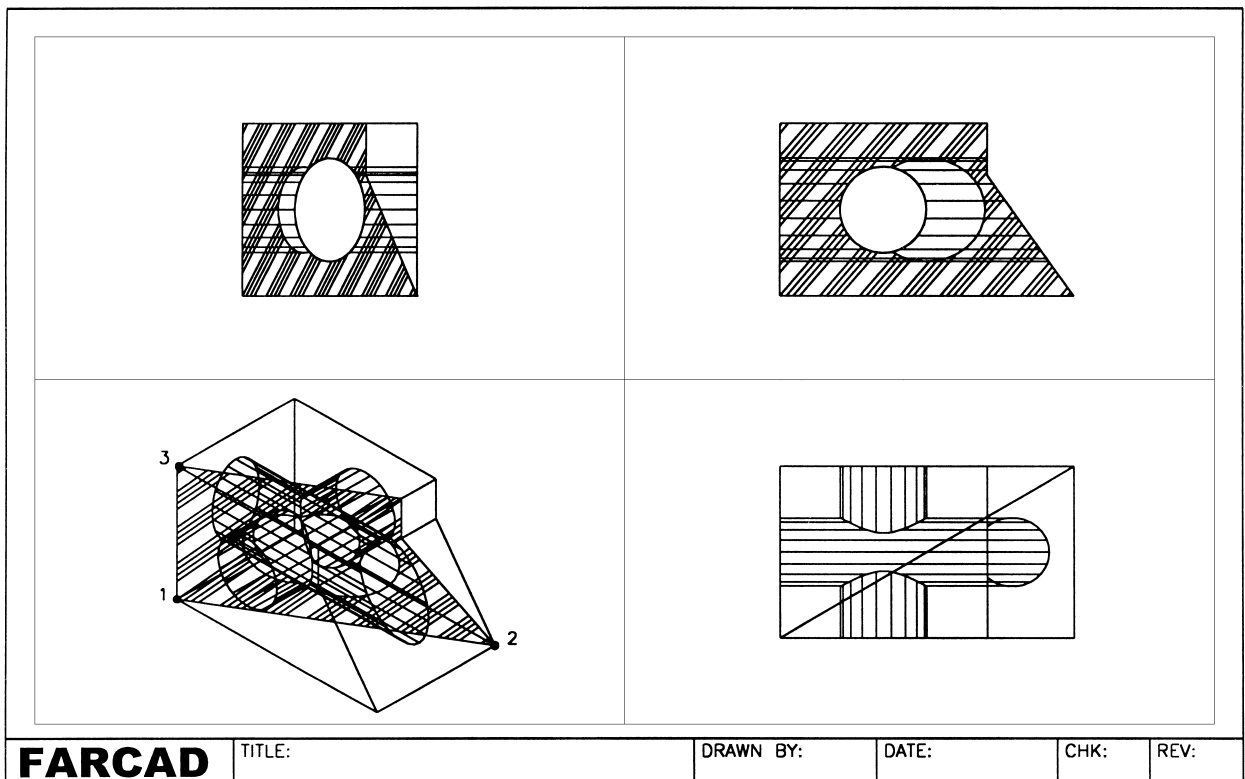


Figure 38.4 Section example with model SLIPBL plotted without hide.

6 Note

- a) Hatching is not automatically added to the region
- b) Hatching must be added to the section region by the user.
- c) The section region is displayed in all viewports, and the viewport specific layer concept can then be used to 'currently freeze' the section outline in specific viewports.
- d) If hatching is to be added to the section region (as it should?), remember to alter the UCS position as hatching is a 2D concept

7 Task

Add hatching to the added section plane using the following information:

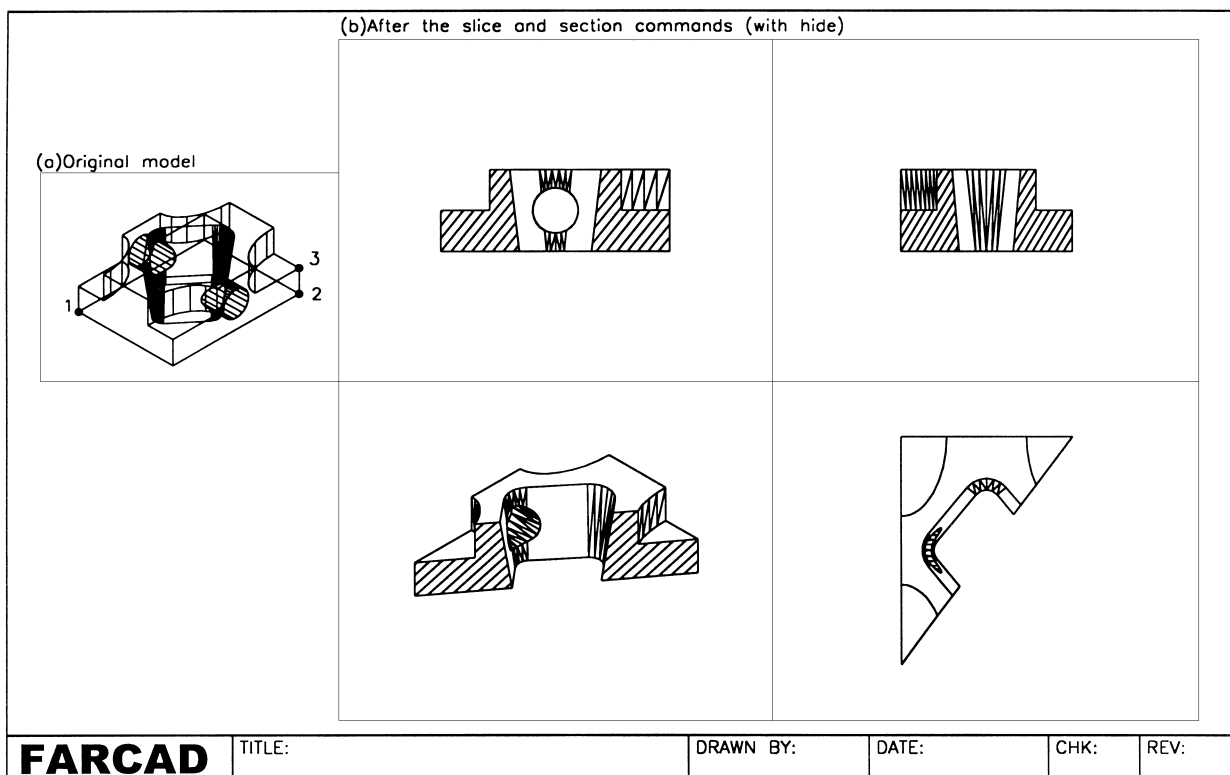
- a) Hatching to be Predefined-ANSI-ANSI33
- b) Scale: 1 and Angle: 0
- c) Hide the model in each viewport with interesting results
- d) A new 3-point UCS will be required and the points should be obvious, as they are the same as the section plane points

8 In a later chapter we will section models 'more realistically'

Using the slice and section commands

If a 'true section' has to be obtained from a solid composite, then the slice and section commands can both be used. To demonstrate this concept:

- 1 Open drawing file MODR2002\CASTBL from chapter 37 with MVLAY1 tab, UCS BASE, layer MODEL and the lower left viewport active.
- 2 Refer to Fig. 38.5 which displays the original model in fig(a)

**FARCAD**

TITLE:

DRAWN BY:

DATE:

CHK:

REV:

Figure 38.5 Using the SLICE and SECTION commands with CASTBL.

- 3 With the SLICE command:
 - a) pick the composite then right-click
 - b) use the three points on slicing plane option and pick the ENDpoints of points 1,2 and 3
 - c) enter 0,100,0 as the point on the desired side on the plane
 - d) a new composite will be displayed
- 4 With layer SECT current, use the SECTION command and:
 - a) pick the new composite then right-click
 - b) use the three point options with the same selection as step 3
 - c) region added to composite
- 5 Hatch the region with correct UCS setting
- 6 Fig. 38.5(b) displays the result of the SLICE and SECTION commands

Summary

- 1 The SLICE command produces a new composite on the same layer as the original model
- 2 The SECTION command adds a region to the section plane of the model, but does not add any hatching to this region. Hatching must be added by the user and is UCS dependent
- 3 Both commands are very similar with the following options:
 - a) the XY, YZ and ZX slicing/sectioning planes
 - b) defining any three points on the slice/section plane
 - c) relative to an object
 - d) relative to the viewing plane in the current viewpoint
- 4 The SLICE command requires:
 - a) a point on the slicing plane
 - b) a point on the desired side of the plane which is to 'be kept'
- 5 The SECTION command only requires a point on the sectioning plane
- 6 The orientation of the UCS will affect the XY, YZ and ZX planes
- 7 The two commands can be activated:
 - a) by icons from the Solids toolbar
 - b) from the menu bar with Draw-Solids
 - c) by entering SLICE or SECTION from the keyboard.

Profiles and true shapes

When the HIDE command is used with a solid composite, the model is displayed with hidden line removal – as expected. From an ‘engineering viewpoint’ this may not be what is wanted, as the front, top and end views should have lines which represent hidden detail. AutoCAD 2002 allows this hidden detail to be ‘added’ to a solid model with the PROFILE command and when used, new layers are automatically added to the existing drawing.

A profile is defined as: *an image which displays the edges and silhouettes of curved surfaces of the solid for the current view*

The command will be demonstrated with previously created models.

Example 1 – slip block

- 1 Open drawing MODR2002\SLIPBL from chapter 37.
- 2 In paper space, use the LIST command and pick the top right viewport border. The AutoCAD Text Window will display information about the viewport including the **HANDLE** number. Take a note of this handle number, then repeat the LIST command selecting the other three viewport borders. In each case note the handle number as it will be referred to in the exercise. My handle numbers for the four viewports were:
top right: 70 top left: 72
lower right: 76 lower left: 74
- 3 *Note:*
 - a) all objects created in AutoCAD are given a handle number and this number increases every time a new object is added to the drawing
 - b) the handle numbers are in hexadecimal format – don’t worry about this if you have never heard of it. It is not important at this level of AutoCAD
 - c) the handles are for assisting with the AutoCAD database
 - d) your four viewport handle numbers will probably differ from mine. This is perfectly normal. Simply note them.
- 4 After the viewport border handle numbers have been noted, return to model space with UCS BASE.
- 5 Make layer 0 current and the top left viewport active.
- 6 Refer to Fig. 39.1

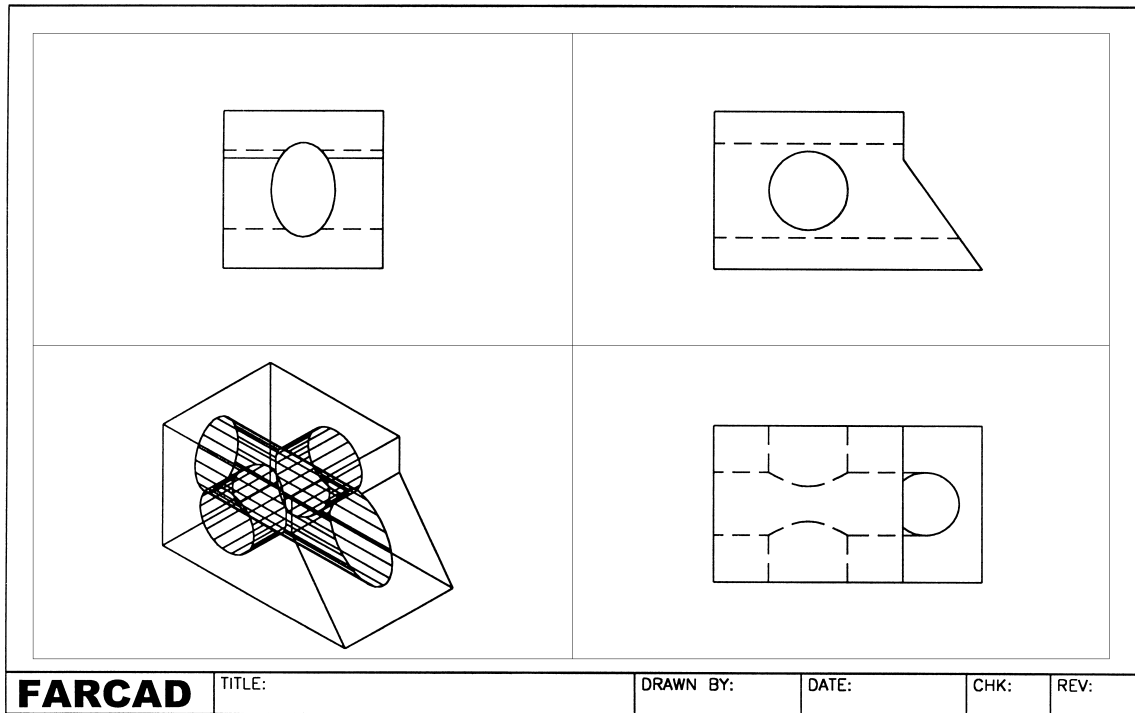


Figure 39.1 Using the PROFILE command with SLIPBL.

- 7 Select the SETUP PROFILE icon from the Solids toolbar and:

prompt Select objects

respond **pick the composite then right-click**

prompt Display hidden profile lines on a separate layer?[Yes/No]<Y>

enter **Y <R>**

prompt Project profile lines onto a plane?[Yes/No]<Y>

enter **Y <R>**

prompt Delete tangential edges?{Yes/No}<Y>

enter **Y <R>**

and the model in the top left viewport will be displayed with black lines over the model lines

- 8 Menu bar with **Format-Layer** and:

prompt Layer Properties Manager dialogue box

with two new layers (*):

a) **PH-72** for hidden profile lines – Hidden linetype?

b) **PV-72** for visible profile lines – Continuous

respond 1. check the linetype for layer PH-72

2. if it is not Hidden, then LOAD the HIDDEN linetype and set it to layer PH-72

3. freeze layer MODEL in the current viewport

4. pick OK

- 9 The two new layers (*) will have the same handle number as the top left viewport, in my case 72

- 10 The top left viewport will now only display the profile visible and hidden lines for the composite as layer MODEL has been frozen in the current viewport. Do you understand why the linetype for layer PH-72 was set to HIDDEN?

- 11 Make the top right viewport active and from the menu bar select **Draw-Solids-Setup-Profile** and:
- prompt* Select objects
 - respond* **pick the composite then right-click**
 - prompt* Display hidden profile lines.. and <R>
 - prompt* Project profile lines.. and <R>
 - prompt* Delete tangential edges.. <R>
- 12 The model will be displayed with black visible and hidden lines
- 13 Menu bar with **Format-Layer** and:
- prompt* Layer Properties Manager dialogue box
 - with* Two new layers:
 - a) PH-70 with HIDDEN linetype
 - b) PV-70 with CONTINUOUS linetype
 - respond* 1. freeze layer MODEL in current viewport
2. pick OK
 - and* a) remember that PH-70 and PV-70 are the visible and hidden layers for my viewport handles. You may have different PH and PV handle numbers
b) layer PH-70 has HIDDEN linetype as this linetype was loaded in step 8
- 14 The top right viewport will display visible and hidden lines for the model.
- 15 Make the lower right viewport active, still with layer 0 current
- 16 At the command line enter **SOLPROF <R>** and:
- a) pick the model then right-click
 - b) enter Y <R> to the display hidden profile lines prompt
 - c) enter Y <R> to the project profile lines prompt
 - d) enter Y <R> to the delete tangential edges prompt
 - e) layer properties manager dialogue box and:
 - 1. two new layers: PH-76 and PV-76
 - 2. freeze layer MODEL in current viewport
 - 3. pick OK
- 17 The viewport will display the visible and hidden detail lines for the model in the viewport
- 18 At this stage the layout should resemble Fig. 39.1 and can now be saved if required, but not as SLIPBL.
This exercise is now complete.

Explanation

The PROFILE command is **viewport specific** and when activated:

- 1 Two viewport specific layers are automatically created, these being **PH-??** and **PV-??**
- 2 The PH layer is for hidden detail
- 3 The PV layer is for visible detail
- 4 The ?? with the layer name is the current viewport handle number and is not controlled by the user
- 5 The PV linetype should always be continuous
- 6 The PH linetype should always be hidden, but **MAY HAVE TO BE LOADED** by the user
- 7 The command must be used in each viewport in which profile detail has to be extracted
- 8 The command is generally used in viewports which display top, front and side views of the model
- 9 The command is generally not used in a viewport which displays a 3D view of the model
- 10 The hidden and visible detail added are blocks, i.e. there is a hidden detail block and a visible detail block. These blocks can be exploded if required.

Example 2 – the pipe and flange

- 1 Open the pipe/flange model created in chapter 33 with MVLAY1 tab, UCS BASE, layer MODEL, lower left viewport active and refer to Fig. 39.2
- 2 In the top left viewport, activate the PROFILE command and and:
 - a) pick the model then right-click
 - b) enter Y <R> to the three prompts
 - c) activate the Layer Properties Manager dialogue box and:
 1. load the HIDDEN linetype and set to the new PH layer
 2. freeze layer MODEL in the current viewport
- 3 Repeat step 2 in the top right and lower right viewports, but do not load the HIDDEN linetype – it is already loaded?
- 4 Extracting profiles is as simple as this!
- 5 Profile drawings can be dimensioned using viewport specific layers using the same procedure as chapter 25. I will let you try and add the dimensions for yourself, but remember to:
 - a) create a new layer for each viewport
 - b) set the correct UCS – dimensioning is a 2D concept.
- 6 Save the layout when complete.

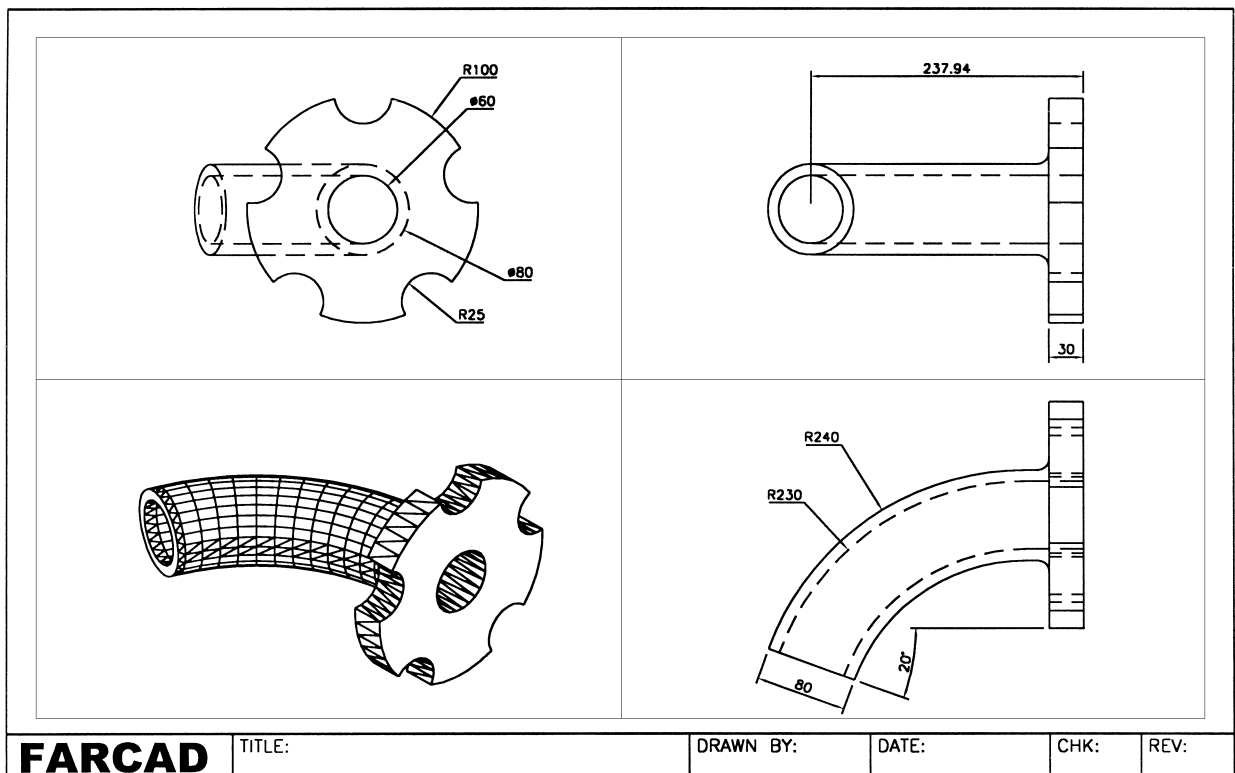


Figure 39.2 The PROFILE command with the flange/pipe model.

Profile explanation

When the profile command is used with a solid model, three prompts are displayed and it is usual to enter Y to these prompts.

a) *Display hidden profile lines on separate layers*

This creates two blocks, one for visible lines and one for hidden lines. Two new viewport specific layers are created for this block information PH-?? and PV-??. The actual ?? number is the handle of the current viewport, i.e. it is unique. The PV (visible detail) has a continuous linetype, while the PH (hidden detail) has a hidden linetype. The hidden linetype must be loaded before it can be 'assigned' to the PH layer

b) *Project profile lines onto a plane*

The profile detail is displayed as 2D objects and is projected onto a plane perpendicular to the viewing direction and passing through the UCS origin

c) *Delete tangential edges*

A tangential edge is an imaginary edge where two faces meet at a tangent. Tangential edges are not shown for most drawing applications.

TRUE SHAPES

A true shape is obtained when a surface (face) is viewed at right angles. In AutoCAD 2002 this can be obtained with the PLAN and Solids Editing commands. We will use a model from a previous chapter to demonstrate how a true shape can be obtained.

- 1 Open the machine support model from chapter 31 and refer to Fig. 39.3.

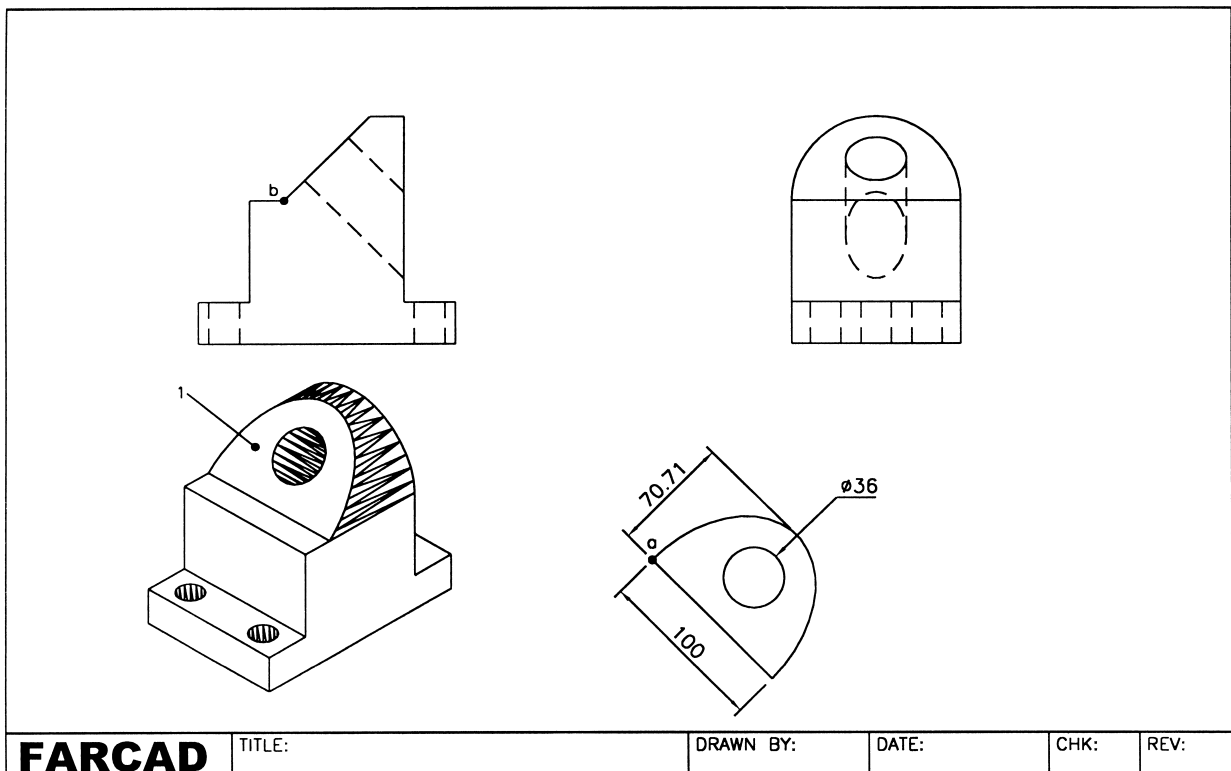


Figure 39.3 True shape extraction using composite model 1.

- 2 In the top two viewports, use the PROFILE command and:
 - a) accept the three Y prompts
 - b) freeze layer MODEL in these two active viewports
 - c) load linetype HIDDEN and set to the two new PH layers
- 3 Create a new layer TS, colour to suit and current
- 4 With the lower right viewport active, restore UCS SLOPE which should have been saved with the model during the chapter 31 exercise
- 5 Menu bar with **View-3D Views-Plan View-Current UCS** to display the sloped surface of the model as a 'true shape'
- 6 Menu bar with **Modify-Solids Editing-Copy faces** and:

prompt Select objects

respond **pick any pt 1 within the sloped face then right-click and enter** (toggle between the two lower viewports)

prompt Specify a base point and enter: **0,0 <R>**

prompt Specify a second point of displacement and enter: **@0,0,10 <R>**
- 7 a) freeze layer MODEL in the current (lower right) viewport to display the true shape of the sloped surface
 - b) making each of the other viewports active, freeze layer TS in the other three current viewports
 - c) make the lower right viewport active, still with UCS SLOPE
- 8 Menu bar with **Modify-Rotate** and:

prompt Select objects

respond **pick any point on the true shape**

prompt Specify base point and enter: **0,0 <R>**

prompt Specify rotation angle and enter: **-45 <R>**
- 9 The true shape is rotated and now has to be positioned relative to the sloped surface 'from which it was copied'
- 10 In the lower right viewport, zoom extents the zoom 1.5 to 'scale' the true shape to the same 'value' as the model
- 11 In paper space, activate the MOVE command and:

prompt Select objects

respond **pick any point on the lower right viewport border then right-click**

prompt Specify base point or displacement

respond **Endpoint icon and pick pt a**

prompt Specify second point of displacement

respond **Endpoint icon and pick pt b**
- 12 Repeat the MOVE command with:
 - a) objects: pick the moved viewport border
 - b) base point: endpoint icon and pick pt b
 - c) second point: enter **@170<-45**
- 13 The true shape is moved to another part of the screen
- 14 *Task*
 - a) dimension the true shape with the correct UCS and layer. This layer will need to be currently frozen in the other viewports
 - b) freeze the layer VP to give the layout as Fig. 39.3
- 15 The exercise is now complete and should be saved as it will be used in the next chapter.

Summary

- 1 Profiles can be extracted from models
- 2 Profiles display views of solid models with visible and hidden details
- 3 New layers are created with the PROFILE command, PV for visible objects and PH for hidden objects
- 4 The PH and PV layers are not controlled by the user and are assigned handle numbers. These handle numbers are those of the viewport in which the profile was extracted.
- 5 True shapes can be extracted from models with the copy faces solids editing command.

Dimensioning in model and paper space

All users will know that AutoCAD has two drawing environments, these being:

- a)* model space: used to create the model
- b)* paper space: used to layout the drawing sheet for plotting

When a multiple viewport layout has been created in paper space and dimensions have to be added to the 'model', many users are unsure whether these dimensions should be added in model space or paper space. In this chapter we will investigate both model and paper space dimensioning.

We will use a previously created model for the demonstration.

Getting started

- 1 Open the SLIPBL model from chapter 37 with MVLAY1 tab active
- 2 In model space with the top left viewport active:
 - a)* set a FRONT 3D view
 - b)* zoom extents then zoom at 1.5 scale
 - c)* extract a profile of the model
 - d)* load linetype HIDDEN if required
 - e)* freeze layer Model in the current viewport
- 3 Repeat step 2 in the top right and lower right viewports
- 4 You should now have the same view in three viewports
- 5 Refer to Fig. 40.1, restore UCS FRONT and make layer DIM current
- 6 If you have the ability, modify the 3DSTD dimension style to increase the text height to 6. This is not greatly important
- 7 Linear dimension the top line of the model and diameter dimension the circle. These two dimensions will be displayed in all four viewports, as dimensioning is global.
- 8 Erase these two dimensions

Viewport specific layer

- 1 With **Format-Layer** from the menu bar, use the Layer Properties Manager dialogue box and:
 - a) pick the DIM layer then New
 - b) alter the Layer1 new name to DIMTL
 - c) pick Current then OK
- 2 With the top right viewport current, activate the Layer Properties Manager dialogue box and:
 - a) pick the DIMTL line
 - b) pick freeze in current viewport
 - c) pick OK
- 3 Repeat step 2 in the lower left and lower right viewports, then make the top left viewport current
- 4 Dimension the top horizontal line and the circle as before and these two dimensions will only be displayed in the top left viewport due to the viewport specific layer DIMTL. These two dimensions are designated by (a) in Fig. 40.1

Paper space dimensioning

- 1 Enter paper space
- 2 Dimension the same two objects as before, i.e. the top horizontal line and the circle. Use the DIM layer.
- 3 These two dimensions are designated by (b) in Fig. 40.1 and are obviously not correct
- 4 How then can paper space be used to give 'true' dimensions?

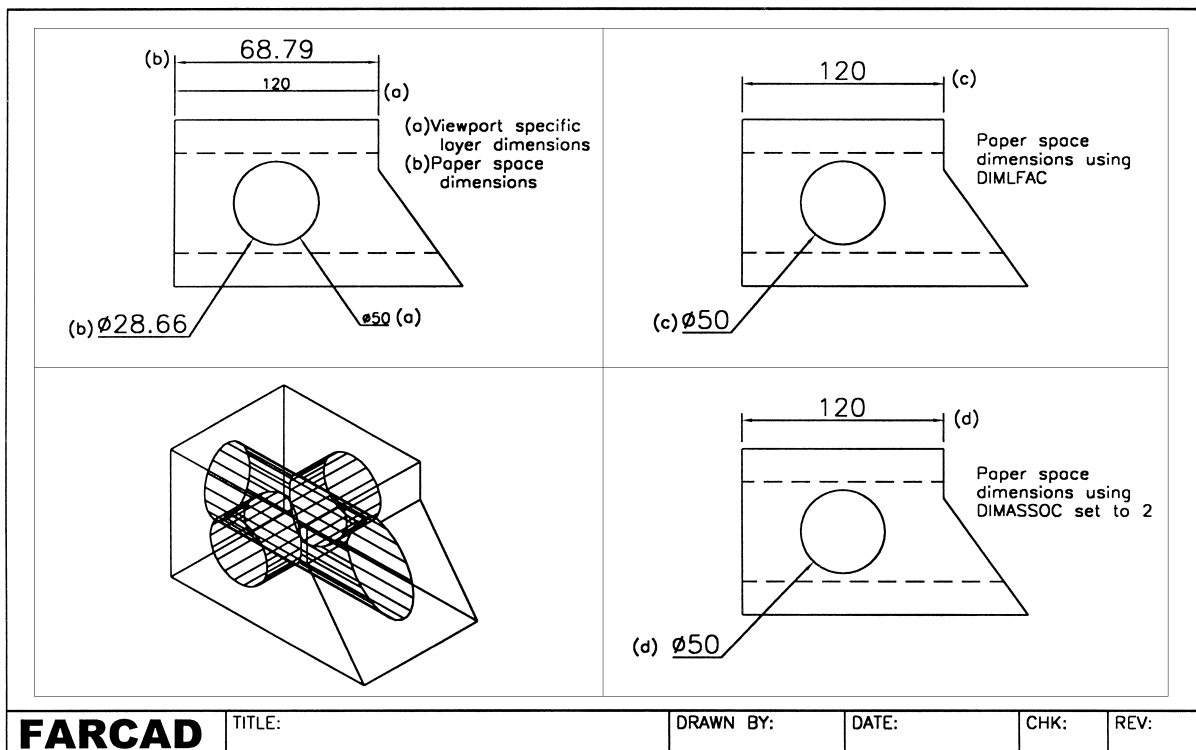


Figure 40.1 Adding dimensions in model and paper space.

Using DIMLFAC

DIMLFAC (Dimension Linear scale Factor) is a system variable which when set to the correct value will scale linear and radial measurements and allow paper space dimensioning of model space objects to have the 'correct' value.

- 1 In model space make the top right viewport active, then enter paper space
- 2 At the command line enter **DIM <R>** and:


```
prompt Dim:
enter DIMLFAC <R>
prompt Enter new value for dimension variable, or Viewport
enter V <R> - the viewport option
prompt Select viewport to set scale
respond pick the border of the top right viewport
prompt DIMLFAC set to -1.74449
then Dim:
respond ESC to end the command line dimension sequence
```
- 3 Still in paper space, dimension the same linear and circular objects in the top right viewport. The dimensions should now be correct, designated by (c)

Using DIMASSOC

DIMASSOC is a system variable which controls dimension associativity, i.e. its value determines whether any added dimension will change when the object it is associated with is changed. DIMASSOC can have one of three values as follows:

- a) 0 : dimensions are displayed exploded, i.e. any part of the dimension can be selected
- b) 1 : the complete dimension is a single object and model space associativity applies.
This is the normal default value
- c) 2 : the complete dimension is a single object and paper space associativity applies

- 1 Paper space still active
- 2 At the command line enter **DIMLFAC <R>** and:


```
prompt Enter new value for DIMLFAC<-1.7445>
enter 1 <R>
```
- 3 At the command line enter **DIMASSOC <R>** and:


```
prompt Enter new value for DIMASSOC<?> - probably 1 value
enter 2 <R>
```
- 4 Now dimension the two objects as before using the lower right viewport. The dimensions should be correct, as (d)
- 5 This exercise is now complete and need not be saved.

Paper space dimension exercise

- 1 Open the profile drawing saved in the previous chapter and refer to Fig. 40.2
- 2 With command line entry, set DIMLFAC to 1 and DIMASSOC to 2
- 3 Enter paper space and add the linear and circular dimensions
- 4 Are these paper space dimensions correct?
- 5 Enter model space with layer TS current and make viewport with the true shape active
- 6 Copy the true shape to another part of the viewport – you may have to ‘paper space’ stretch this viewport
- 7 Explode the copied shape
- 8 In paper space, with layer DIM current, align dimension the 100 line and diameter dimension the circle
- 9 Enter model space and:
 - a) activate the scale command
 - b) window the true shape
 - c) pick a suitable base point
 - d) enter a scale factor of 1.2
- 10 The model space shape **and the paper space dimensions** should both be scaled by 1.2
- 11 This is **true associativity**, i.e. paper space dimensions are associated with model space objects.

This exercise is complete and can be saved.

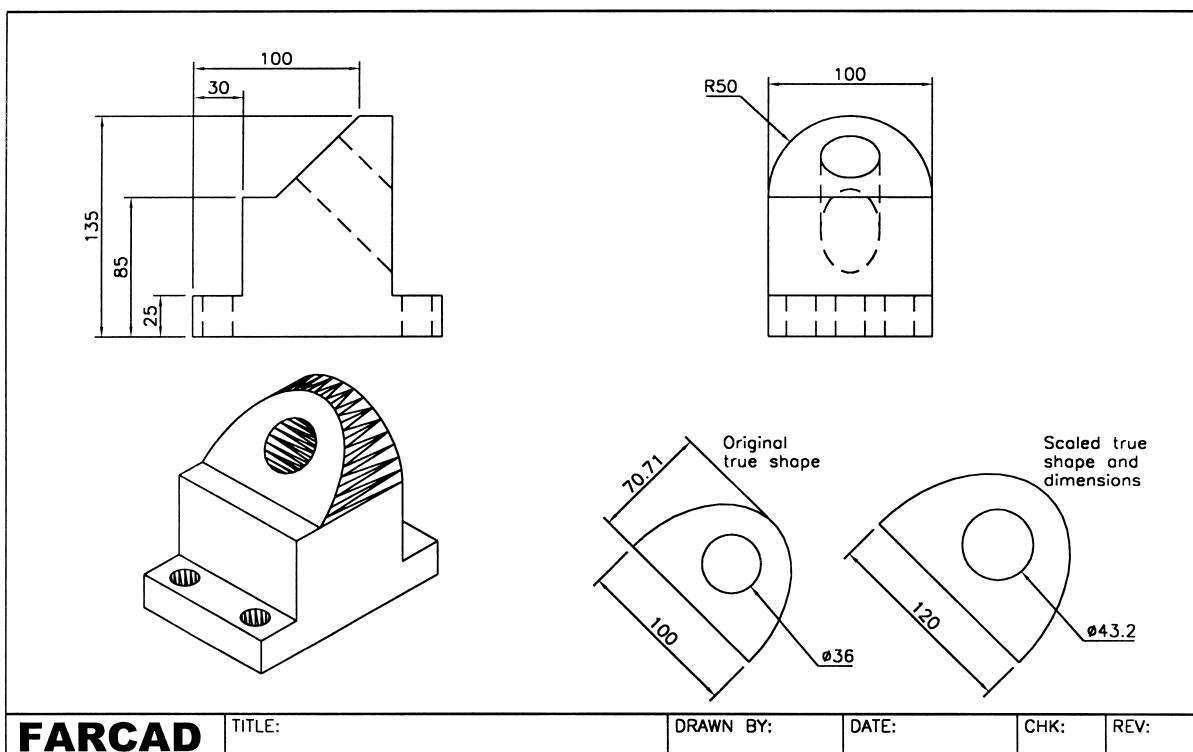


Figure 40.2 Paper space dimension exercise.

Summary

- 1 Dimension can be added to models in both model and paper space
- 2 Multiple viewport model space dimensioning requires viewport specific layers to be made by the user
- 3 Paper space dimensions can be added to multiple viewport models by setting the DIMLFAC system variable
- 4 True associative paper space dimensions can be added to models in a multiple viewport layout with the DIMASSOC system variable set to 2. This is the recommended method.

Assignment

This activity requires the garden block from the previous activity to be dimensioned.

Activity 23: Adding dimensions to the garden block of MACFARAMUS.

- 1 Open your activity 22
- 2 Set DIMASSOC to 2
- 3 Extract profiles from the top and front views then currently freeze the model layer in these viewports
- 4 Add suitable paper space dimensions to the model layout
- 5 Extract a diagonal section through the block and hatch using the AR-CONC predefined pattern at a suitable scale
- 6 Save with a suitable name

A detailed drawing

In this example a new composite will be created and used to display the model as a detailed drawing. The model to be created is a desk tidy, so open your A3SOL template file with MVLAY1 tab active, UCS BASE, layer MODEL, lower left viewport active.

Altering the viewports

- 1 In paper space select the STRETCH icon from the Modify toolbar and:
prompt Select objects
enter **C <R>** – the crossing option
prompt Specify first corner and enter: **160,100 <R>**
prompt Specify opposite corner and enter: **260,200 <R>**
prompt 4 found, Select objects
respond **right-click**
prompt Specify base point or displacement and enter: **200,135 <R>**
prompt Specify second point of displacement and enter: **@-25,25 <R>**
- 2 The viewport configuration will be altered
- 3 Return to model space with UCS BASE

The basic shape

- 1 Refer to Fig. 41.1
- 2 With the lower left viewport active restore UCS FRONT
- 3 Draw a polyline using the following entries:
Start point: 0,0 next point: @156,0 next point: @0,15
next point: @-132,0 next point: @0,10 next point: @-24,0
next point: close
- 4 With the EXTRUDE icon from the Solids toolbar, extrude the red polyline with:
a) height: **-85**
b) taper: **0**
- 5 Restore UCS BASE and zoom centre about the point 78,42,8 at **0.9XP** in the 3D viewport and **IXP** in the other three viewports. The XP entry is to allow for the different viewport sizes, i.e. the model is being zoomed about a centre point relative to the size of the viewport.
- 6 The extruded polyline will be displayed as fig(a)

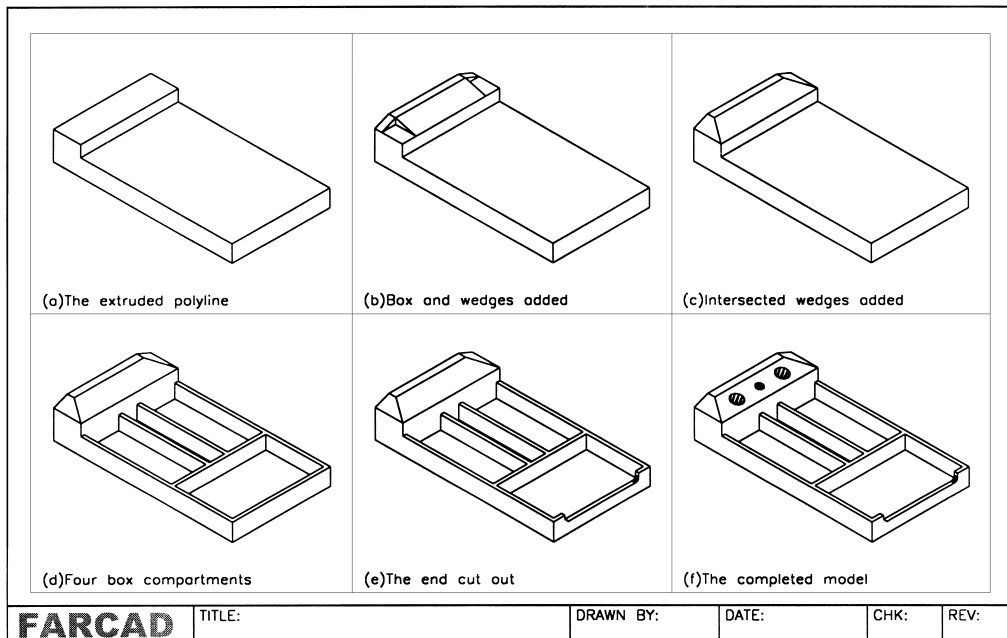


Figure 41.1 Steps in the construction of the desk tidy (3D view only with hide).

The top

- 1 Create a box primitive with:
 - a) corner: 0,12,25
 - b) length: 6; width: 61; height: 5
- 2 Create three wedges using the following information:
 - a) corner: 6,12,25; length: 18; width: 61; height: 5
 - b) corner: 6,12,25; length: 12; width: -6; height: 5
 - c) corner: 6,73,25; length: 12; width: 6; height: 5
- 3 With the ROTATE icon, rotate the following wedges:
 1. wedge (b) about the point 6,12,25 by **-90**
 2. wedge (c) about the point 6,73,25 by **90**
- 4 Union the box and the three wedges with the red extrusion
- 5 The model at this stage resembles fig(b)
- 6 Create another two wedges with:
 - a) corner: 6,0,25; length: 18; width: 12; height: 5
 - b) corner: 24,12,25; length: 12; width: -18; height: 5
- 7 Rotate the second wedge (b) about the point 24,12,25 by **-90**
- 8 Menu bar with **Modify-Solids Editing-Intersect** and:

prompt Select objects

respond **pick the two wedges then right-click**
- 9 Menu bar with **Modify-3D Operation-Mirror 3D** and:

prompt Select objects

respond **pick the intersected wedges then right-click**

prompt Specify first point of mirror plane or..

enter **ZX <R>** – the ZX plane option

prompt Specify point on ZX plane

enter **0,42.5,25 <R>**

prompt Delete source objects?<N> and enter: **N <R>**
- 10 Union the two sets of intersected wedges with the composite
- 11 The model should be displayed as fig(c)

The compartments

- 1 The desk tidy compartments will be created from boxes subtracted from the composite.
- 2 With lower left viewport active and UCS BASE, create the following four box primitives:

	<i>box1</i>	<i>box2</i>	<i>box3</i>	<i>box4</i>
corner	153,3,3	100,3,3	100,36,3	100,52,3
length	-50	-76	-76	-76
width	79	30	13	30
height	20	20	20	20
colour	magenta	blue	green	blue
- 3 Subtract the four boxes from the red composite – fig(d)
- 5 Shade the model in the 3D viewport then return to 2D wireframe representation

The end cut-out

- 1 Lower left viewport active with UCS BASE
- 2 Set a new UCS position with the 3 point option using:
 - a) origin: 156,0,0
 - b) x-axis position: 156,85,0
 - c) y-axis position: 156,0,15
 - d) save UCS position as NEWEND
- 3 Draw a polyline with the following keyboard entries:

Start point: 10,15
Next point: @0,-3
Next point: arc option with arc endpoint: @3,-3
Next point: line option with line endpoint: @59,0
Next point: arc option with arc endpoint: @3,3
Next point: line option with line endpoint: @0,3
Next point: close
- 4 Set ISOLINES to 6 and FACETRES to 0.5
- 5 Extrude the polyline for a height of -3 with 0 taper
- 6 Subtract the extruded polyline from the composite – fig(e)
- 7 Restore UCS BASE

The holes on the slope

- 1 In paper space zoom in on the 3D sloped area then model space
- 2 Set a new UCS position with the 3 point option using:
 - a) origin: 24,42.5,25
 - b) x-axis position: 24,85,25
 - c) y-axis position: 6,42.5,30
 - d) save UCS position as SLOPE

- 3 Create three cylinders, colour magenta with:

	<i>cylinder 1</i>	<i>cylinder 2</i>	<i>cylinder 3</i>
centre	0,9.34077,0	20,9.34077,0	-20,9.34077,0
radius	3	5	5
height	-12	-20	-20
- 4 Subtract the three cylinders from the red composite – fig(f)
- 5 In paper space, zoom previous and return to model space

The complete model

- 1 Restore UCS BASE
- 2 The model is now complete and your screen display should be similar to Fig. 41.2
- 3 Save the model at this stage as **MODR2002\DESKTIDY**
- 4 In the 3D viewport use the 3D orbit command with Gouraud shading then return the original view as wireframe

Extracting a profile

- 1 Refer to Fig. 41.3 and with the top left viewport active, select the SETUP PROFILE icon from the Solids toolbar and:
 - prompt* Select objects
 - respond* **pick the composite then right-click**
 - prompt* Display hidden profile lines.. and enter: **Y <R>**
 - prompt* Project profile line.. and enter: **Y <R>**
 - prompt* Delete tangential edges and enter: **Y <R>**

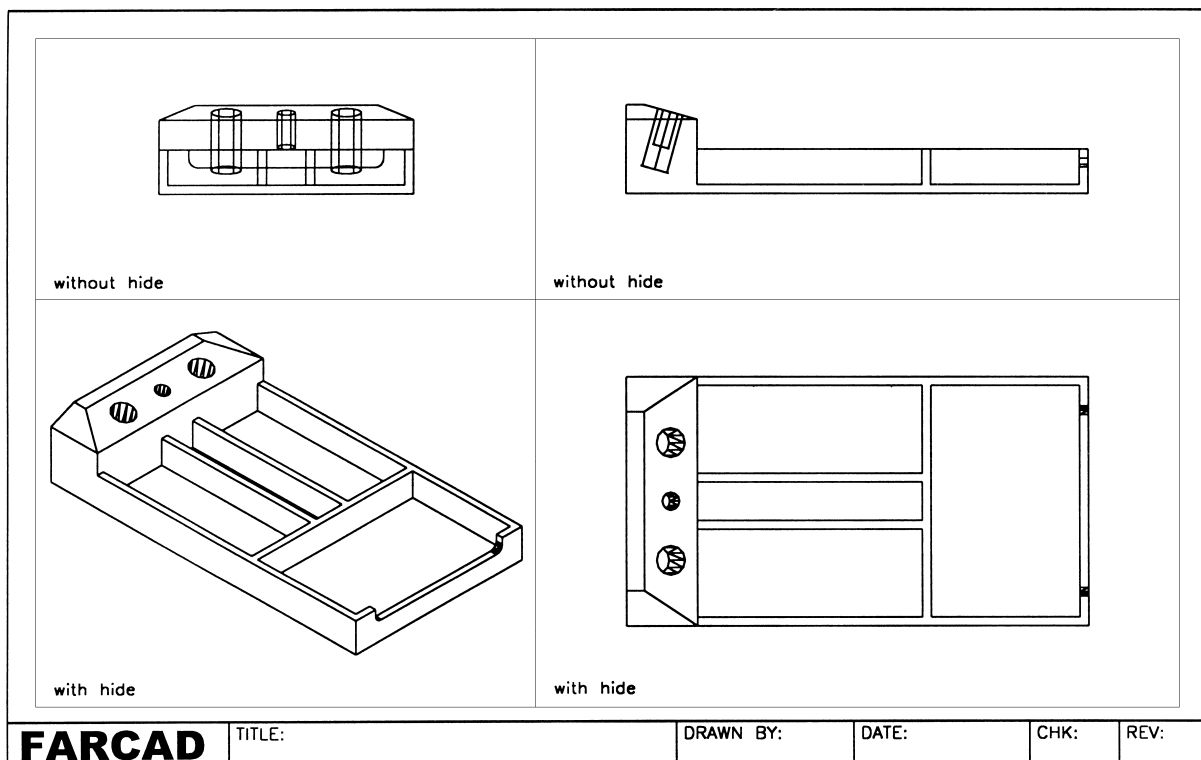


Figure 41.2 Complete desk tidy model.

- 3 The model will be displayed with black lines
- 4 Using the Layer Properties Manager dialogue box:
 - a) note new layers PH?? and PV??
 - b) load the linetype HIDDEN (if required) and set the new PH?? layer with this linetype
 - c) freeze layer MODEL in this active viewport

Extracting the section

- 1 With the top right viewport active, ensure UCS BASE and make layer SECT current
- 2 With the SECTION icon from the Solids toolbar:
 - a) pick the composite then right-click
 - b) enter ZX as the section plane
 - c) enter 0,42,5,0 as a point on the plane
 - d) a region will be displayed in all viewports
- 3
 - a) currently freeze layer MODEL in the top right viewport
 - b) currently freeze layer SECT in the other three viewports
- 4 With an appropriate UCS setting, add hatching to the region using: pattern name: ANSI32; scale: 1; angle: 0

Extracting the true shape

- 1 The layout on your screen at present will differ from Fig. 41.3 so in paper space use the MOVE command to interchange the viewports, i.e. the two lower viewports to the top of the paper sheet and the two top viewports to the bottom of the sheet. Use the endpoint icon and pick a viewport border corner
- 2 Make a new layer named TS, colour to suit and current
- 3 In the top left and the two lower viewports, freeze the new layer TS in these active viewports
- 4 With the top right viewport active, restore UCS SLOPE
- 5 Menu bar with **Modify-Solids Editing-Copy** faces and:

prompt Select faces

respond **pick any point within the sloped face indicated by pt 1 in the top left viewport then right-click/enter**

prompt Specify a base point and enter: **0,0,0 <R>**

prompt Specify a second point and enter: **@0,0,0 <R>**

prompt Enter a face editing option and enter: **X <R>** and **X <R>**
- 6 The face has been copied 'on to itself'
- 7 Now currently freeze layer MODEL in the top right viewport to display the coloured true shape of the sloped surface
- 8 Menu bar with **View-3D Views-Plan View-Current UCS** to display the copied face (true shape) in plan view. The position of this plan view may not be ideal
- 9 With the ROTATE icon from the MODIFY toolbar:
 - a) pick the shape and <R>
 - b) enter 0,0 as the base point
 - c) enter **74.476** as the rotation angle – this angle should become apparent once the dimensions have been added

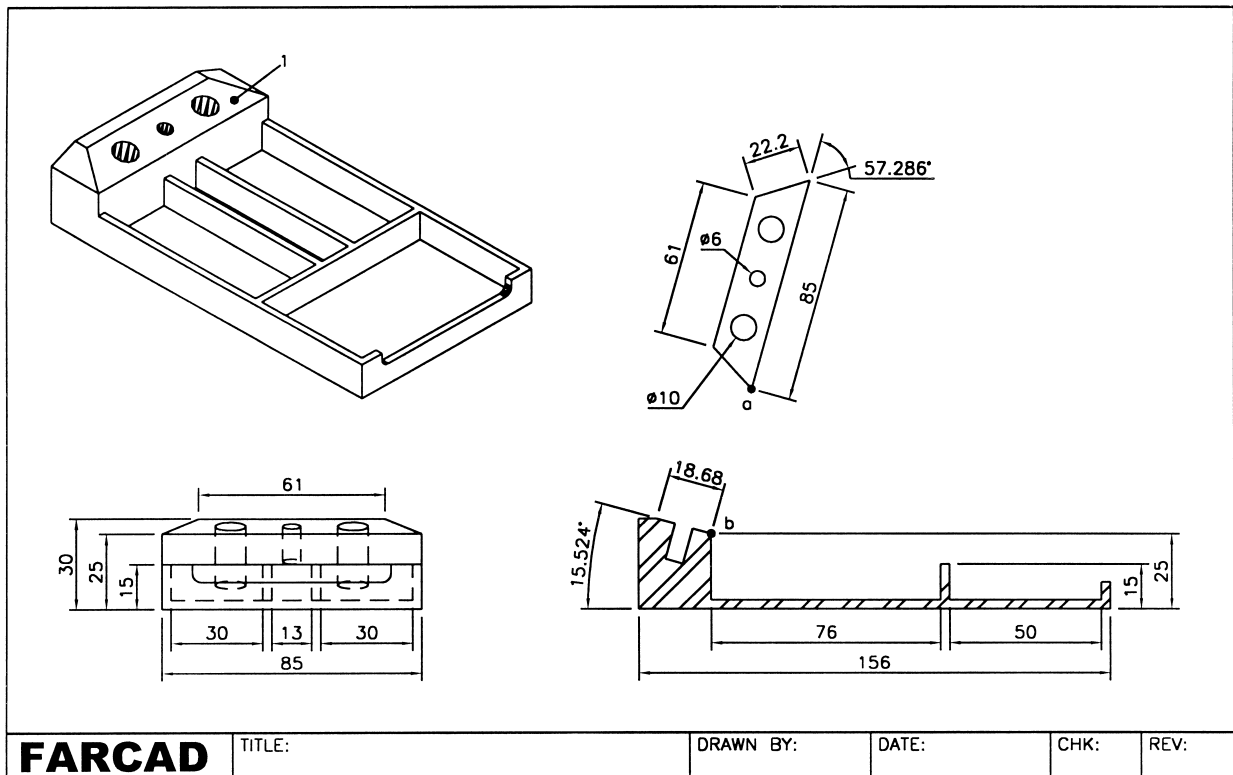


Figure 41.3 Extracting details for the desk tidy model.

- 10 Pan the rotated shape to a suitable part of the viewport
- 11 In paper space activate the MOVE command and:
 - a) objects: pick the top right viewport border then right-click
 - b) base point: endpoint icon and pick pt a
 - c) second point: endpoint icon and pick pt b
- 12 Repeat the MOVE command and:
 - a) objects: pick the same viewport border then right-click
 - b) base point: endpoint icon and pick pt b
 - c) second point: enter @50<74.476 <R>
- 13 The true shape is now positioned relative to the slope from which it was copied

Task

- 1 Inquire into the model and:
 - a) Area: 47433.58
 - b) Mass: 106339.71 – my figures
- 2 Modify the dimension style and set the decimal angle precision to 0.000
- 3 With DIMASSOC set to 2, add paper space dimensions to the layout as Fig. 41.3
- 4 The detail exercise is now complete and can be saved.
- 5 This detailed drawing has used most of the solid model concepts we have investigated, i.e. profile extraction, section extraction, paper space dimensioning etc.

Blocks, wblocks and external references

Solid model blocks and wblocks can be created and inserted into drawings like any other 2D or 3D object. In this chapter we will create two interesting (I hope) solid model assembly drawings from blocks and wblocks and then investigate solid model external references. We will also investigate the 'interference' between solids.

Example 1 – a desk tray assembly

- 1 Open your A3SOL template file with the MVLAY1 tab as normal and refer to Fig. 42.1
- 2 Zoom centre about 55,40,20 at 0.75XP in all viewports
- 3 Set the ISOLINES system variable to 8 and FACETRES to 0.5

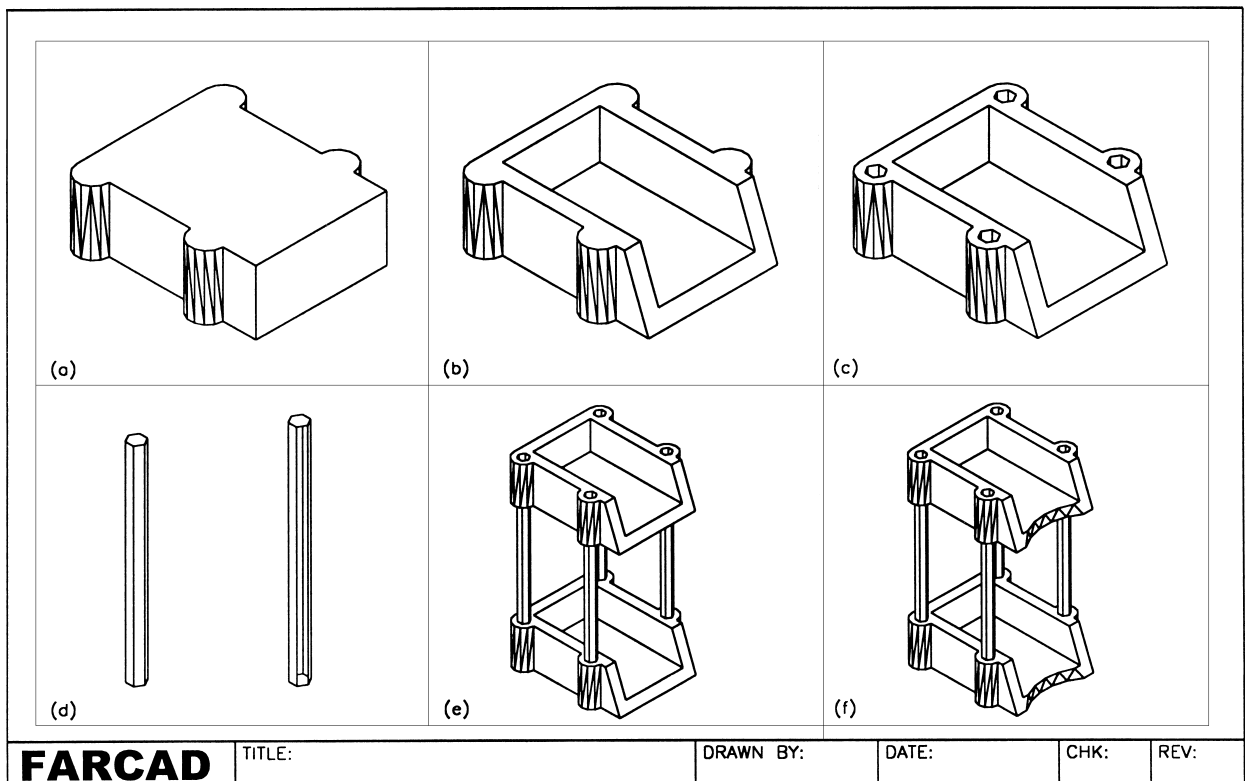


Figure 42.1 Creation of the assembled model.

The Tray

- 1 Make a new layer TRAY, colour blue and current
- 2 With UCS BASE, create two primitives from:

<i>Box</i>	<i>Cylinder</i>
corner: 0,0,0	centre: 10,0,0
length: 110	radius: 10
width: 80	height: 40
height: 40	
- 2 Rectangular array the cylinder:
 - a) for 2 rows and 2 columns
 - b) row distance: 80 and column distance: 70
- 3 Union the box and the four cylinders – fig(a)
- 4 Create another two primitives from:

<i>Box</i>	<i>Wedge</i>
corner: 10,10,10	corner: 110,0,40
length: 110	length: -15
width: 60	width: 80
height: 40	height: -40
- 5 Subtract the box and wedge from the composite – fig(b)
- 6 Draw a polygon with:
 - a) sides: 5
 - b) centre: 10,0,0
 - c) inscribed in a circle of radius 5
- 7 Solid extrude the polygon for a height of 10 with 0 taper
- 8 Menu bar with **Modify-3D Operation-3D Array** and:

<i>prompt</i>	Select objects
<i>enter</i>	L <R><R> – two returns for the cylinder (last object)
<i>prompt</i>	Enter the type of array and enter: R <R>
<i>prompt</i>	Enter the number of rows and enter: 2 <R>
<i>prompt</i>	Enter the number of columns and enter: 2 <R>
<i>prompt</i>	Enter the number of levels and enter: 2 <R>
<i>prompt</i>	Specify the distance between rows and enter: 80 <R>
<i>prompt</i>	Specify the distance between columns and enter: 70 <R>
<i>prompt</i>	Specify the distance between levels and enter: 30 <R>
- 9 Subtract the eight cylinders from the composite – fig(c). A paper space zoom may be needed to help with this, but remember to return to model space
- 10 At the command line enter **-BLOCK <R>** and:

<i>prompt</i>	Enter block name and enter: TRAY <R>
<i>prompt</i>	Specify insertion base point and enter: 0,0,0 <R>
<i>prompt</i>	Select objects
<i>respond</i>	pick the composite then right-click
- 11 The tray block may disappear. If it does not, erase it
- 12 The -BLOCK entry allows the prompts to be entered from the keyboard and not from the dialogue box

The support

- 1 Make a new layer SUPPORT, colour magenta and current
- 2 Draw two polygons then solid extrude using the following:

	<i>polygon 1</i>	<i>polygon 2</i>
a) sides:	6	6
b) centre:	0,0,0	50,50,0
c) inscribed:	rad 5	rad 5
d) height:	130	140
e) taper:	0	0
- 3 Change the colour of the 140 high cylinder to green
- 4 The two extrusions are displayed as fig(d)
- 5 Make two blocks of these supports with:
 - a) Name: SUP1
 Insertion base point: 0,0,0
 Objects: pick the red extrusion and <RETURN>
 - b) Name: SUP2
 Insertion base point: 50,50,0
 Objects: pick the green cylinder and <RETURN>

Inserting the blocks

- 1 Zoom extents the zoom 1.2 in all viewports
- 2 Make layer TRAY current
- 3 Menu bar with **Insert-Block** and:

prompt Insert dialogue box

respond 1. At Name scroll and pick TRAY
 2. ensure the on-screen prompts are active (tick)
 3. pick OK

prompt Specify insertion point and enter: **0,0,0 <R>**

prompt Enter X scale factor and enter: **1 <R>**

prompt Enter Y scale factor and enter: **1 <R>**

prompt Specify rotation angle and enter: **0 <R>**
- 4 Repeat the INSERT command and insert block TRAY:
 - a) at the point 0,0,150
 - b) full size (i.e. X=Y=1) with 0 rotation
- 5 At the command line enter **-INSERT <R>** and:

prompt Enter block name and enter: **SUP2 <R>**

prompt Specify insertion point and enter: **10,0,30 <R>**

prompt Enter X scale factor and enter: **1 <R>**

prompt Enter Y scale factor and enter: **1 <R>**

prompt Specify rotation angle and enter: **0 <R>**
- 6 Menu bar with **Modify-Solids Editing-Union** and:

prompt Select objects

respond **pick the three inserted blocks then right-click**

prompt At least 2 solids or coplaner regions must be selected
- 7 What does this prompt mean? The union operation has not been successful with the three inserted blocks. This is because blocks must be exploded before they can be used with Boolean operations.
- 8 Using the EXPLODE icon from the Modify toolbar, select the three inserted blocks

Checking for interference

- 1 Make layer 0 current
- 2 Menu bar with **Draw-Solids-Interference** and:


```

prompt  Select the first set of solids
        Select objects
respond  pick the top tray then right-click
prompt  Select the second set of solids
        Select objects
respond  pick the support then right-click
prompt  Comparing 1 solid against 1 solid
        Interfering solids (first set)   : 1
                                   (second set) : 1
        Interfering pairs                : 1
        Create interference solids>[Yes/No]<N>
enter   Y <R>

```
- 3 The model will be displayed with a black solid (layer 0) where interference occurs between the tray and the support leg. This interference is due to the support leg (SUP2) being too long, or the TRAY having been inserted into the drawing at the wrong insertion point. We deliberately created the support leg (SUP2) with a height of 140 to obtain interference.
- 4 Erase the support leg and the black interference cylinder will still be displayed. Erase this interference cylinder.

Inserting the correct support

- 1 Still with the two (exploded) inserted trays displayed?
- 2 UCS BASE, lower left viewport active and layer SUPPORT current
- 3 INSERT the support block SUP1 with:
 - a) insertion point: 10,0,30
 - b) full size with 0 rotation
 - c) explode this inserted block
- 4 Rectangular array the inserted support:
 - a) for 2 rows and 2 columns
 - b) row distance: 80 and column distance: 70
- 5 Make layer MODEL current
- 6 Menu bar with **Modify-Solids Editing-Union** and pick the six exploded blocks then right-click – fig(e)
- 7 The composite will be displayed blue or magenta. Why is this when layer MODEL (red) is current? The colour of the composite depends on the order you selected for the union operation. If a tray was the first object selected, then the composite will be blue, and if a support was first selected, then the composite will be magenta
- 8 Using the CHPROP command, pick the composite and change the layer to MODEL. The composite should be displayed in red.

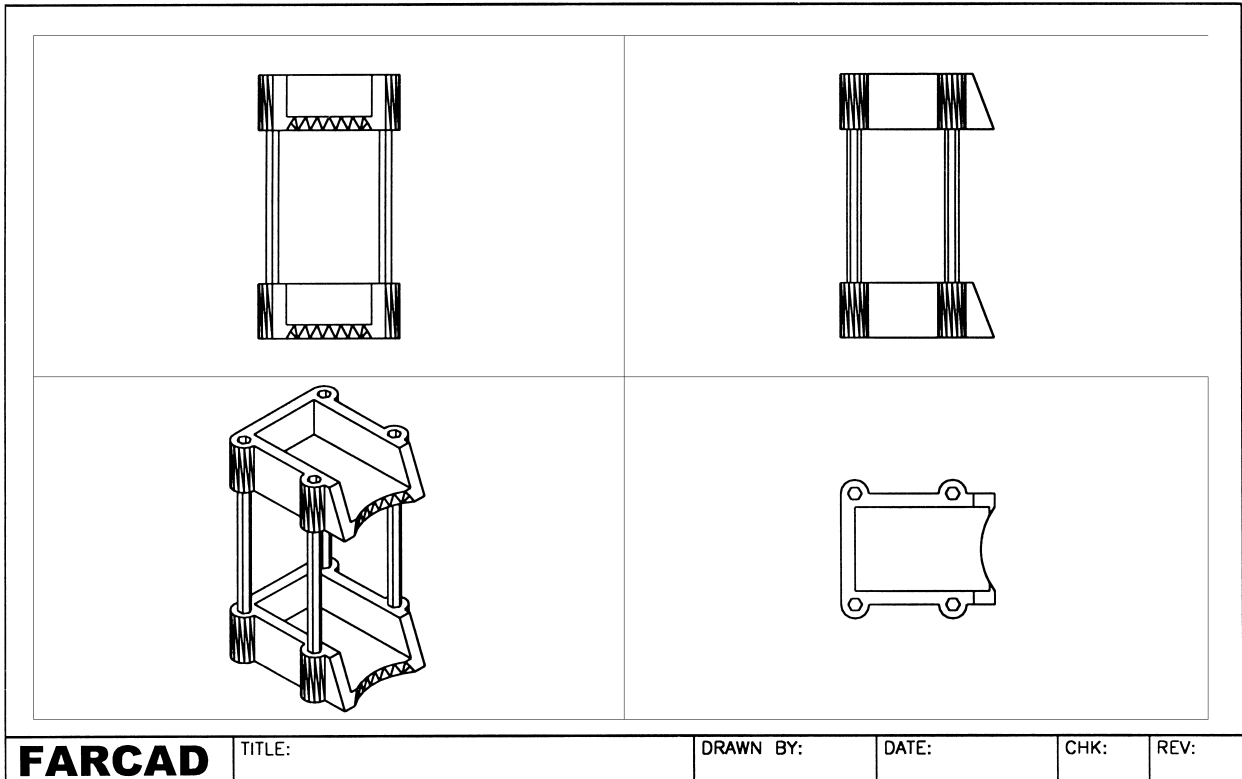


Figure 42.2 Compared desk tray assembly model.

Completing the model

- 1 Create a cylinder with:
 - a) centre: 150,40,0
 - b) radius: 50 and height: 200
- 2 Subtract this cylinder from the composite – fig(f)
- 3 The model is now complete and can be saved as **MODR2002\DESKTRAY**
- 4 Figure 42.2 displays the four viewport configuration of the completed model assembly.

Example 2 – a wall clock

In this example, seven different coloured blocks will be created and used for an assembly drawing. The assembly will then be used for profile and section extraction.

- 1 Open your A3SOL template file with MVLAY1 tab as normal
- 2 Refer to Fig. 42.3 which displays the blocks to be created.
- 3 Restore UCS FRONT, layer MODEL and zoom centre about 100,75,0 at 250 mag in all viewports.

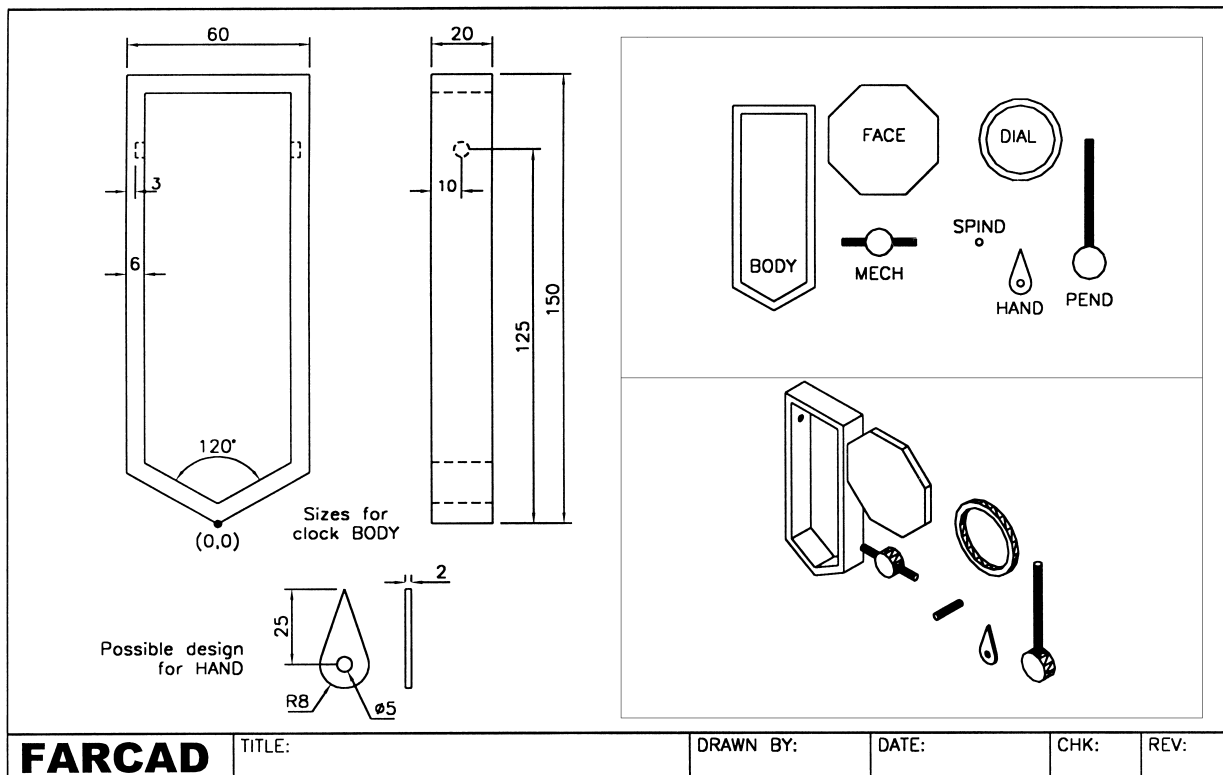


Figure 42.3 Wall clock solids for block creation.

Creating the blocks

Body (red)

- 1 Create the clock body from two polylines using the sizes given. The start point should be at 0,0 and your discretion is needed for any sizes not given. (I created the body from lines then used the polyline edit command to join them into one polyline. I then offset the polyline by 6)
- 2 Solid extrude the two polylines for a height of -20 with 0 taper
- 3 Subtract the inside extrusion from the outside extrusion
- 4 Set ISOLINES to 6 and FACETRES to 0.5
- 5 Create a cylinder with:
 - a) centre: -27,125,-10 and diameter: 5
 - b) centre of other end option: @54,0,0
- 6 Subtract the cylinder from the extruded composite
- 7 Make a block of the composite body with:
 - a) block name: BODY
 - b) insertion base point: 0,0,0

Face (blue)

- 1 Draw a blue octagon with:
 - a) centre: 80,0,125
 - b) circumscribed in a circle of radius: 40
- 2 Solid extrude the octagon for a height of 6 with 0 taper
- 3 Make a block of the extruded octagon with:
 - a) block name: FACE
 - b) insertion base point: 80,125,0

Dial (green)

- 1 Create two green cylinders:
 - a) centre: 180,125,0; diameter: 60; height: 5
 - b) centre: 180,125,0; diameter: 50; height: 5
- 2 Subtract the smaller cylinder from the larger cylinder
- 3 Block the composite as DIAL with insertion base point: 180,125,0

Mechanism (magenta)

- 1 Create two magenta cylinders:
 - a) centre: 50,50,0; diameter: 5; centre of other end: @54,0,0
 - b) centre: 77,50,-5; diameter: 20; height: 10
- 2 Union the two cylinders
- 3 Block the composite as MECH, insertion base point: 77,50,0

Pendulum (cyan)

- 1 Create two cyan cylinders:
 - a) centre: 230,125,0; diameter: 6; centre other end: @0,-90,0
 - b) centre: 230,35,-5; diameter: 24; height: 10
- 2 Union the two cylinders
- 3 Block the composite as PEND with insertion base point: 230,125,0

Spindle (colour number 124)

- 1 Create a cylinder (colour number 20) with:
 - a) centre: 150,50,0
 - b) diameter: 5 and height: 25
- 2 Block as SPIND with insertion base point: 150,50,0

Hand (colour number 124)

- 1 Create a cylinder with centre: 180,20,0, diameter: 5 and height: 2
- 2 Create a polyline outline using sizes as a guide – own design
- 3 Solid extrude the polyline for a height of 2 with 0 taper

- 4 Subtract the cylinder from the extrusion
- 5 Block as HAND with insertion base point: 120,20,0

Inserting the blocks

- 1 Still with UCS FRONT, layer MODEL and lower left viewport
- 2 Zoom centre about 0,75,0 at 200 magnification – all viewports
- 3 At the command line enter **-INSERT <R>** and:
 - prompt* Enter block name and enter: **BODY**
 - prompt* Specify insertion point and enter: **0,0,0**
 - prompt* Enter X scale factor and enter: **1**
 - prompt* Enter Y scale factor and enter: **1**
 - prompt* Specify rotation angle and enter: **0**
- 4 Insert the other blocks using the following information:

<i>name</i>	<i>insertion pt</i>	<i>scale</i>	<i>rot</i>
FACE	0,125,0	X=Y=1	0
DIAL	0,125,6	X=Y=1	0
MECH	0,125,-10	X=Y=1	0
PEND	0,125,-10	X=Y=1	0
SPIND	0,125,-5	X=Y=1	0
HAND	0,125,15	X=Y=1	0
HAND	0,125,18	X=Y=0.75	-150
- 5 Shade the 3D viewport (Gouraud) the use the 3D orbit command. The result is very impressive
- 6 **DO NOT** union the inserted blocks but save as **MODR2002\CLOCK** for the rendering exercises.

Tasks

- 1 Union the eight inserted blocks – explode needed? The model will be displayed with the block inserted colours. Why is this different from the desk tidy example when the union command resulted in the model being displayed in one colour?
Think layers!
- 2 In the lower right viewport extract a profile of the model to display hidden detail and remember:
 - a) hidden linetype must be loaded and set to the pH layer
 - b) currently freeze layer MODEL in the viewport
 - c) optimise the LTSCALE variable
- 3 In the top left viewport extract a vertical section through the model and remember:
 - a) use the layer SECT with the required section plane
 - b) add hatching to the region – UCS is important
 - c) currently freeze layer MODEL in this viewport
 - d) currently freeze layer SECT in other viewports
- 4 Add an additional viewport to display the assembled model from below
- 5 The final layout should be as Fig. 42.4 and can be saved.

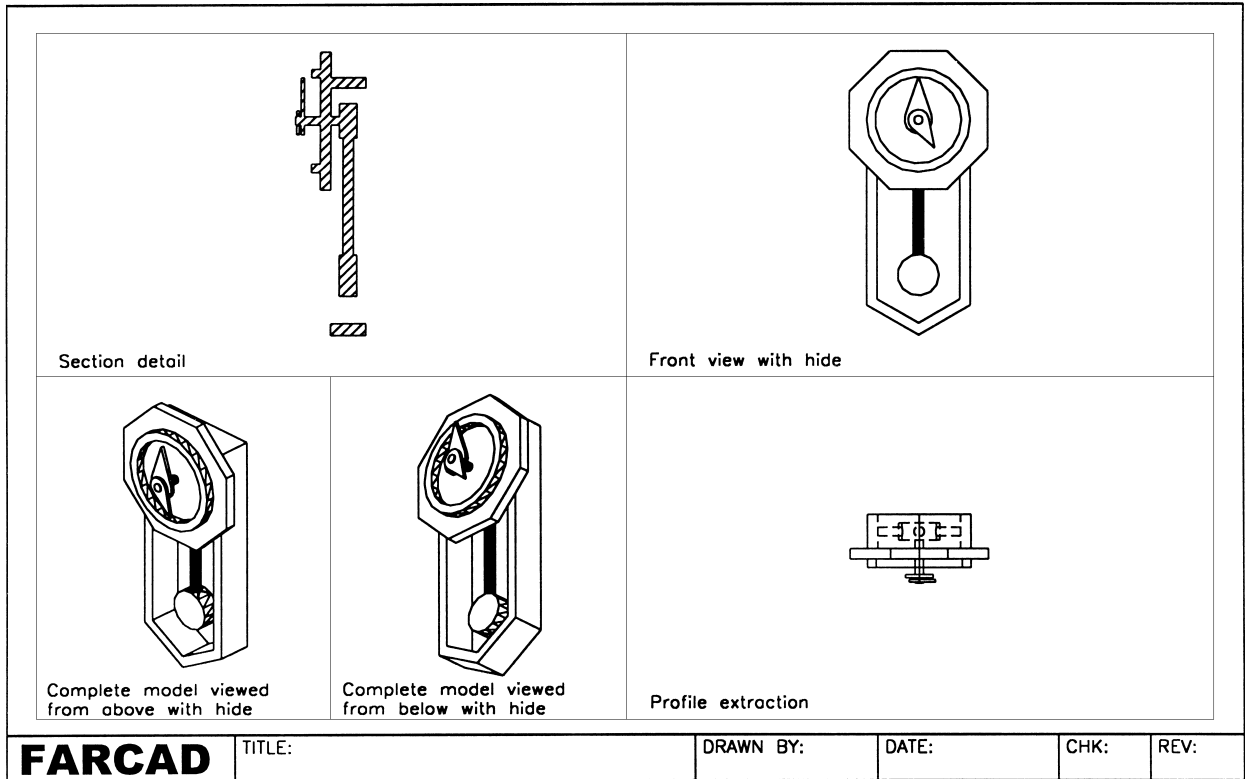


Figure 42.4 Assembled wall clock with profile and section extraction..

External references with solid models

To demonstrate how solid models can be used as wblocks and external references, a relatively simple layout will be created to represent a workshop, the machines being large machining centres. Solid model composite will be used for these machining centres.

- 1 Open your A3SOL template file with MVLAY1 as normal
- 2 Restore the WCS with the command line entry **UCS <R>** then **W <R>**
- 3 Create the following primitives:

	<i>box</i>	<i>wedge</i>	<i>wedge</i>
corner:	0,0,0	100,0,70	100,80,70
length:	100	-50	-50
width:	80	15	-25
height:	70	-50	-50
colour:	red	blue	green
- 4 Create a cylinder, centre at 20,0,70 and radius 20 with centre of other end at @0,80,0. This cylinder is to be magenta
- 5 Union the box and cylinder and subtract the two wedges from the composite. This model represents the machining centre.

- 6 At the command line enter **WBLOCK <R>** and:
- prompt* Write Block dialogue box
respond 1. Source: objects
 2. Base point with X: 0, Y: 0, Z: 0
 3. Objects: pick Select objects and:
- prompt* Select objects
respond **pick the composite then right-click**
prompt Write Block dialogue box
respond 1. File name: **MACH1**
 2. Location: c:\MODR2002 or named your folder
 3. Insert units: Millimeters
 4. pick OK
- prompt* Write block preview displayed at top left of screen
- 7 Erase the composite then create another box and cylinder with:
- a) box: corner at 0,-20,0 with cube option of length 40
 b) cylinder: centre at 40,0,20, radius 15 with centre of other end at @40,0
- 8 Change the colour of the box and cylinder to colour number 24 then union the two primitives. This model represents the robot feeder for the machining centres
- 9 Make a second wblock with:
- a) source: objects
 b) base point: X 0, Y 0, Z 0
 c) objects: select objects, pick composite, right-click
 d) file name: MACH2
 e) location: C:\MODR2002 or named your folder
 f) insert units: millimeters
 g) pick OK
- 10 Close the A3SOL file (no save) then re-open it again
- 11 Restore the WCS (UCS-W) then zoom centre about 0,100,50 at 300 magnification in all viewports
- 12 With the 3D viewport active, menu bar with **Insert-External Reference** and:
- prompt* Select Reference File dialogue box
respond 1. scroll and pick C:\MODR2002 or named your folder
 2. scroll and pick MACH1
 3. pick Open
- prompt* External Reference dialogue box
with Name: MACH1
respond 1. ensure all on-screen prompts active (tick)
 2. reference type: Attachment
 3. pick OK
- prompt* Specify insertion point and enter: **-200,-40,0 <R>**
prompt Enter X scale factor and enter: **1 <R>**
prompt Enter Y scale factor and enter: **1 <R>**
prompt Specify rotation angle and enter: **0 <R>**
- 13 Polar array the insertion external reference about the point 0,0 for four items with rotation, the angle to fill being -180

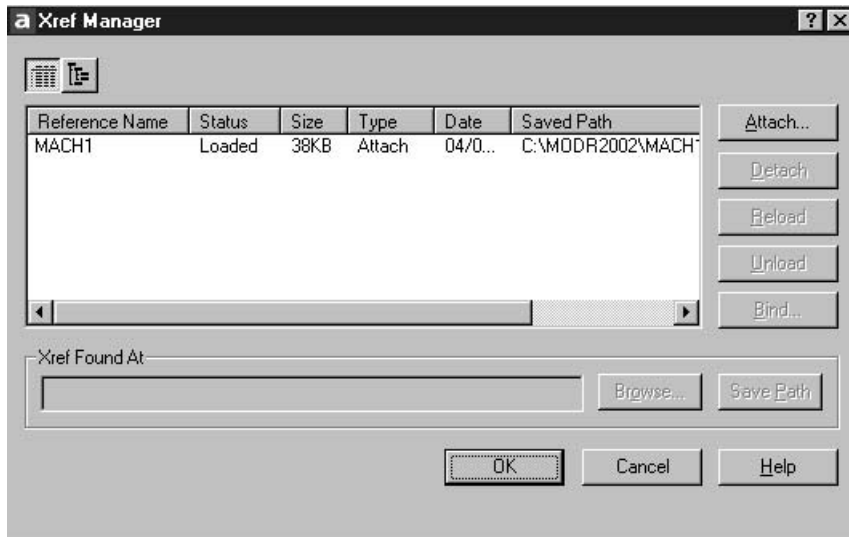


Figure 42.5 The Xref Manager dialogue box for MACH1.

- 14 At the command line enter **XREF <R>** and:
- prompt* Xref Manager dialogue box
- with* MACH1 listed (Fig. 42.5)
- respond* **pick Attach**
- prompt* Select Reference File dialogue box
- respond* 1. ensure MODR2002 active folder
2. scroll and pick MACH2
3. pick Open
- prompt* External Reference dialogue box
- with* Name: MACH2
- respond* 1. ensure all on-screen prompts active
2. reference type: Attachment
3. pick OK
- prompt* Specify insertion point and enter: **0,-20,0 <R>**
- prompt* Enter X scale factor and enter: **1 <R>**
- prompt* Enter Y scale factor and enter: **1 <R>**
- prompt* Specify rotation angle and enter: **120 <R>**
- 15 Now save as **MODR2002\SHOPLAY**
- 16 Figure 42.6(a) displays this layout as a 3D and top view only.
- 17 Close this file then open drawing file MACH1. If a plan view is displayed, set a SE Isometric viewpoint
- 18 Restore the WCS
- 19 Create the following primitives:
- a) box with corner: 40,30,70
Length: 30, Width: 20, Height: 40
colour: red
- b) cylinder with centre: 100,0,70
radius: 20
centre of other end at @0,80
colour: magenta

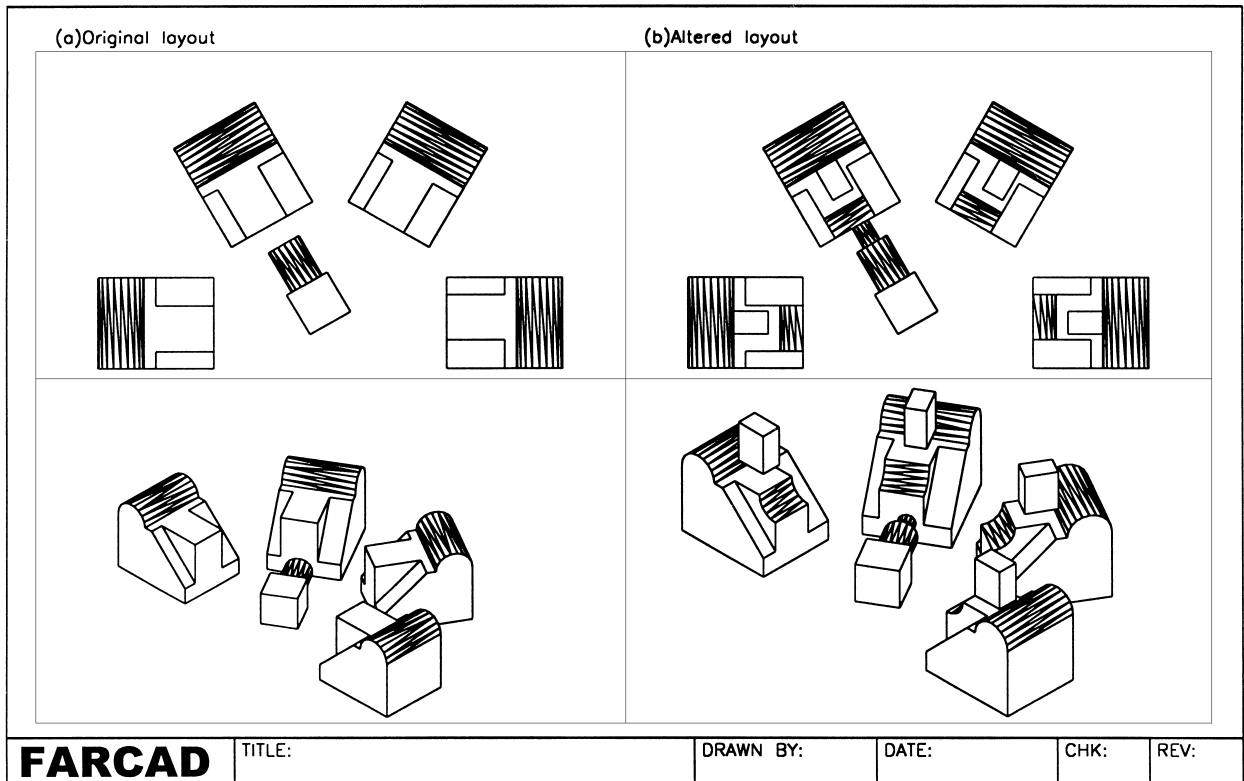


Figure 42.6 SHOPLAY created from external references.

- 20 Union the box and subtract the cylinder from the composite
- 21 Menu bar with **File-Save** to automatically update MACH1 then close the file and open MACH2
- 22 Set a SE Isometric viewpoint and restore the WCS
- 23 Create a cylinder with:
 - a) centre at 80,0,20
 - b) radius: 8
 - c) centre of other end: @20,0
 - d) colour: red
- 24 Menu bar with File-Save to automatically update MACH2 then close the drawing file
- 25 Open the SHOPLAY drawing file and the prompt line will display:


```
Resolve Xref "MACH1": C:\modr2002\MACH1.dwg
"MACH1" loaded
Resolve Xref "MACH2": C:\modr2002\MACH2.dwg
"MACH2" loaded
```
- 26 The SHOPLAY will be displayed with the modified solid composites as Fig. 42.6(b).
- 27 Save the layout, but what is the name? Is it SHOPLAY or a different name? This is your decision.

This exercise completes the chapter.

Summary

Blocks, wblocks and external references can be used with solid modelling as with any other type of drawing layout.

Assignment

In activity 23, the garden block of MACFARAMUS was created. This object was used for many building projects in various parts of the city of CADOPOLIS. Part of a garden project was unearthed which contained a wall, gate and path. The original garden block has to be saved as a block (no pun intended) and used to create the wall, gate and path.

ACTIVITY 24: Using the garden block of MACFARAMUS

- 1 Open the activity 22 drawing and create a block of the garden block with:
 - a) name: GRADBL
 - b) base point: pick the suit – suggest 0,0
 - c) select the object
 - d) ensure insert units are millimetres
- 2 Wall: with UCS RIGHT
 - a) insert the GRADBL at a suitable point
 - b) the scales are full size
 - c) create a suitable wall layout
- 3 Gate: with UCS RIGHT
 - a) insert the GARDBL at the correct point relative to the wall
 - b) scales are X: 2, Y: 3, Z: 0.5
- 4 Path: with UCS BASE
 - a) insert the GRADBL at the correct point relative to the gate
 - b) scales are X: 2, Y: 1, Z: 0.25
- 5 Optimise your layout and save

The setup commands

The setup commands (Drawing, View and Profile) allow drawing layouts to be created by the user. As the profile command has been discussed in a previous chapter, we will only consider the View and Drawing commands in this chapter. These two commands can be summarised as:

View: creates floating viewports using orthographic projection to lay-out multi and sectional views of 3D solid models.

Drawing: generates profiles and sections in viewports which have been created with the setup VIEW command.

Basically the two commands allow the user to create multiple viewport configurations which will display top, front, end and auxiliary views as well as extracting profile and sections of the model. In other words the two commands will create the same type of layout that has been achieved with our A3SOL template file, the 3D viewpoint command and with the SECTION and PROFILE commands.

We will demonstrate the two commands using previously created models.

Example 1 – the backing plate

- 1 Open the drawing file MODR2002\BACKPLT from chapter 32 and:
 - a) MVLAY1 tab active
 - b) restore UCS BASE with layer MODEL current
 - c) in paper space erase the viewports except the 3D view
 - d) scale the 3D viewport about the lower left corner by 0.3
 - e) in paper space ensure the paper space icon is relative to World by command line entry **UCS <R>** then **W <R>**
 - f) in model space, zoom extents
- 2 Refer to Fig. 43.1

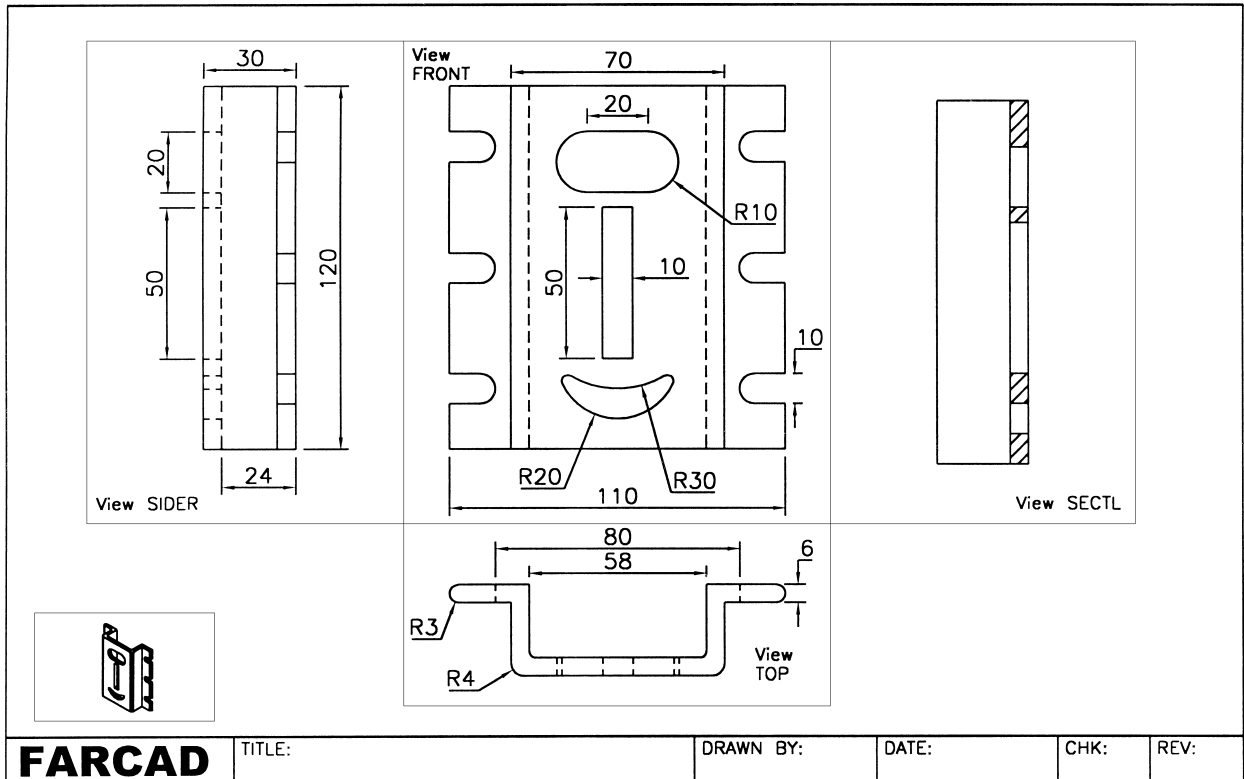



Figure 43.1 The View and Drawing example 1 – the backing plate.

The Setup View command

- 1 Select the SETUP VIEW icon from the Solids toolbar and:

<i>prompt</i>	Enter an option [Ucs/Ortho/Auxiliary/Section]	
<i>enter</i>	U <R> – the Ucs option	
<i>prompt</i>	Enter an option [Named/World/?/Current]<Current>	
<i>enter</i>	N <R> – the named option	
<i>prompt</i>	Enter name of UCS to restore	
<i>enter</i>	BASE <R>	
<i>prompt</i>	Enter view scale<1>	
<i>enter</i>	I <R>	
<i>and</i>	paper space entered with icon displayed	
<i>prompt</i>	Specify view center	
<i>enter</i>	200,50 <R>	
<i>and</i>	UCS icon positioned at the entered point	
<i>prompt</i>	Specify view center, i.e. repositioned the view center?	
<i>respond</i>	<RETURN> i.e. accept entered value	
<i>prompt</i>	Specify first corner of viewport	
<i>enter</i>	130,25 <R>	
<i>prompt</i>	Specify opposite corner of viewport	
<i>enter</i>	270,85 <R>	
<i>prompt</i>	Enter view name	
<i>enter</i>	TOP <R>	
<i>prompt</i>	Enter an option [Ucs/Ortho/Auxiliary/Section]	
<i>enter</i>	X <R> – to end the command	
<i>and</i>	a top view of the model is displayed in a viewport	

- 2 Menu bar with **Draw-Solids-Setup-View** and:
 - prompt* Enter an option [Ucs/Ortho/Auxiliary/Section]
 - enter* **O <R>** – the ortho option
 - prompt* Specify side of viewport to project
 - respond* pick bottom horizontal line of the new viewport and note the midpoint snap effect
 - prompt* Specify view center and enter: **200,170 <R>**
 - prompt* Specify view center and <RETURN>
 - prompt* Specify first corner of viewport and enter: **130,85 <R>**
 - prompt* Specify opposite corner of viewport and enter: **270,245 <R>**
 - prompt* Enter view name and enter: **FRONT <R>**
 - prompt* Enter an option [Ucs/Ortho/Auxiliary/Section] and enter: **X <R>**
- 3 A front view of the model is displayed in a viewport
- 4 At the command line enter **SOLVIEW <R>** and:
 - prompt* Enter an option [Ucs/Ortho/Auxiliary/Section] and enter: **O <R>**
 - prompt* Specify side of viewport to project
 - respond* pick right vertical line of the new (front) viewport
 - prompt* Specify view center and enter: **80,165 <R>**
 - prompt* Specify view center and <RETURN>
 - prompt* Specify first corner of viewport and enter: **130,85 <R>**
 - prompt* Specify opposite corner of viewport and enter: **25,245 <R>**
 - prompt* Enter view name and enter: **SIDER <R>**
 - prompt* Enter an option [Ucs/Ortho/Auxiliary/Section] and enter: **X <R>**
- 5 A right view of the model is displayed
- 6 Activate the setup View command and:
 - prompt* Enter an option [Ucs/Ortho/Auxiliary/Section]
 - enter* **S <R>** – the section option
 - prompt* Specify first point of cutting plane
 - respond* 1. pick the new 'front' viewport
2. enter **0,0,0 <R>**
 - prompt* Specify second point of cutting plane
 - enter* **0,120 <R>**
 - and* a dashed 'section' line is displayed
 - prompt* Specify side to view from
 - respond* **pick a point to left of section line**
 - prompt* Enter view scale and enter: **1 <R>**
 - prompt* Specify view center and enter: **320,165 <R>**
 - prompt* Specify view center and <RETURN>
 - prompt* Specify first corner of viewport and enter: **270,85 <R>**
 - prompt* Specify opposite corner of viewport and enter: **370,245 <R>**
 - prompt* Enter view name and enter: **SECTL <R>**
 - prompt* Enter an option [Ucs/Ortho/Auxiliary/Section]
 - enter* **X <R>**
- 7 A left view of the model is displayed
- 8 *Note:*
 - a) In the example I have given coordinates for the view centre point, the viewport corners, the section line etc. It is usual to 'pick these points' on the screen rather than enter coordinates. The only reason for the coordinate entry was that this was our first example and it should help the user understand the responses to the various prompts
 - b) The view names can be entered as V1, V2, etc. or as TOP, FRONT. This is dependent on the user. Just ensure that you know what view has been allocated to the name.

Investigating the layers

- 1 Menu bar with Format-Layer and note the new layer names:
 - a) FRONT-DIM, FRONT-HID, FRONT-VIS
 - b) SECTL-DIM, SECTL-HAT, SECTL-HID, SECTL-VIS
 - c) SIDER-DIM, SIDER-HID, SECTR-VIS
 - d) TOP-DIM, TOP-HID, TOP-VIS
 - e) VPORTS
- 2 Each new viewport has a dimension, hidden and visible layer and the section viewport has a hatch layer
- 3 In model space make the TOP viewport active and activate the Layer Properties Manager dialogue box. The three FRONT, four SECTL and three SIDER layers are all frozen in this current viewport.
- 4 Using the Layer Properties Manger dialogue box, load the HIDDEN linetype and set the four -HID layers to this linetype. The reason for this will become obvious.
- 5 The new layers are automatically created with the SETUP VIEW command and are viewport specific as follows:
 - a) -DIM: for dimensions
 - b) -HID: for hidden detail
 - c) -VIS: for visible lines
 - d) -HAT: for hatching detail if the section option has been used

The Setup Draw command

- 1 In model space with any viewport active, set the following system variables from the command line:

<i>command line</i>	<i>enter</i>
HPNAME	ANSI31
HPANG	0
HPSCALE	1
- 2 Select the SETUP DRAW icon from the Solids toolbar and:

<i>and</i>	paper space entered
<i>prompt</i>	Select viewports to draw
<i>then</i>	Select objects
<i>respond</i>	pick the four new viewports borders then right-click
- 3 The four new viewports will display the model:
 - a) with hidden line removal: top, front, right
 - b) as a section: left – with no hidden detail
 - c) you should now understand why the HIDDEN linetype was loaded and set to certain layers, and why the three HP system variables were set
- 4 Optimise the LTSCALE system variable for the hidden line detail
- 5 *Note:*
 - a) The three command line entries of HPNAME, HPANG and HPSCALE have set the hatch pattern name, angle and scale. This could have been achieved after the hatching was added to the section view with the Modify-Hatch command.
 - b) Note the model space icon in each viewport has the same orientation. This is because when new viewports are created, the UCSVP system variable defaults to 1, i.e. the UCS icon will have the same configuration in each new viewport. We will leave it as it is.



6 Task:

Add dimensions to the new viewports remembering:

- a) the -DIM layers are viewport specific but need to be current in the appropriate viewport
- b) is a named UCS required for this?
- c) Fig. 43.1 displays the complete layout with dimensions
These dimensions have been added without altering the UCS orientation. They are also viewport specific dimensions and not paper space dimensions. Thus this is another method of adding dimensions to a drawing layout
- d) save the model with your own entered file name

Example 2 – the slip block

- 1 Open model MODR2002\SLIPBL from chapter 37 and:
 - a) in paper space erase three viewports to leave the 3D view
 - b) in model space restore UCS BASE and layer MODEL
 - c) refer to Fig. 43.2
- 2 Use the Setup View command with:
 - a) options: U
 - b) UCS options: N then BASE
 - c) view scale: 0.5
 - d) view center: 190,90 then <RETURN>
 - e) first corner of viewport: 130,50
 - f) opposite corner of viewport: 250,130
 - g) view name: V1
 - h) options: X

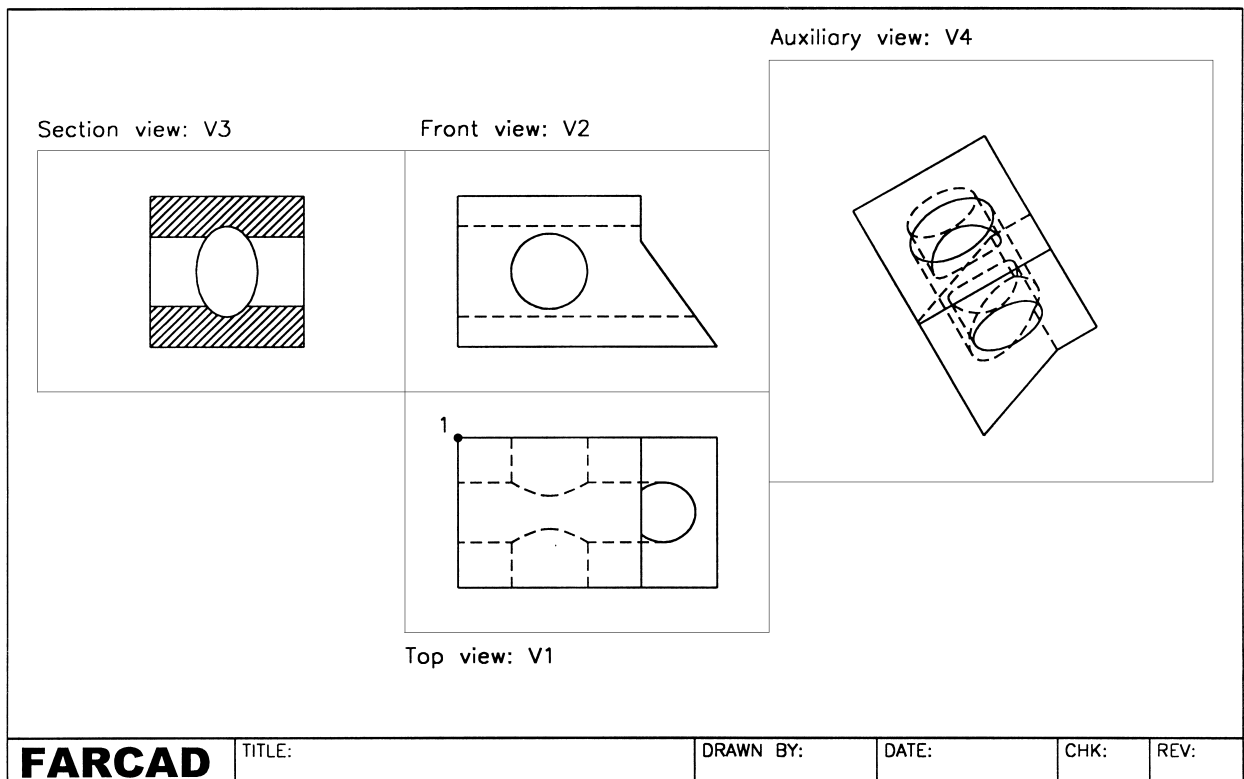


Figure 43.2 The setup View and Drawing example 2 – the slip block.

- 3 In paper space erase the original 3D view then return to model space
- 4 Use the SOLVIEW command with:
 - a) options: U
 - b) UCS options: N then FRONT
 - c) view scale: 0.5
 - d) view center: 190,170 then <RETURN>
 - e) first corner of viewport: 250,130
 - f) opposite corner of viewport: 130,210
 - g) view name: V2
 - h) options: X
- 5 In model space make the new viewport active – should be
- 6 With the SOLVIEW command use the following:
 - a) options: S – section
 - b) first point of cutting plane: 50,0
 - c) second point of cutting plane: 50,100
 - d) side to view from: pick to right of the section line
 - e) view scale: 0.5
 - f) view centre: 0,100 (note icon effect) then <RETURN>
 - g) first corner of viewport: 130,210
 - h) opposite corner of viewport: 10,130
 - i) view name: V3
 - j) options: X
- 7 Activate the Setup View command and:

prompt Enter an option [Ucs/Ortho/Auxiliary/Section]

enter **A <R>** – the auxiliary option

prompt Specify first point of inclined plane

respond **make first new viewport active**

and **ENDpoint icon and pick pt1**

prompt Specify second point of inclined plane

enter **@100<-60 <R>**

prompt Side to view from

respond **pick 'below' the inclined plane**

prompt View centre

enter **0,175 <R>** then <RETURN>

prompt Specify first corner of viewport and enter: **250,100 <R>**

prompt Specify opposite corner of viewport and enter: **390,240 <R>**

prompt View name and enter: **V4 <R>**

prompt Enter an option [Ucs/Ortho/Auxiliary/Section] and enter: **X <R>**
- 8 Select the Setup Drawing icon and pick the four new viewport borders and the display will be:
 - a) as a section: the left viewport
 - b) with hidden line removal: the other three viewports after the HIDDEN linetype has been loaded and set to the -HID layers and the LTSCALE system variable has been optimised
- 9 Note the hatch effect is the ANGLE hatch pattern. This is the default setting
- 10 To alter the hatch pattern, enter model space and make the left(section)viewport active. Restore UCS RIGHT

- 11 Menu bar with **Modify-Object-Hatch** and:
prompt Select associative hatch object
respond **pick the hatching**
prompt Hatch Edit dialogue box
respond 1. Type: User-defined
 2. Angle: 45
 3. Spacing: 4
 4. pick OK
- 12 The model layout is now as Fig. 43.2 and can be saved
- 13 Note: the view centre entry with the section and auxiliary options is perpendicular to the paper space icon. This icon is orientated relative to the:
 a) section plane
 b) auxiliary inclined plane

Example 3 – the pipe and flange model

- 1 Open the drawing file MODR2002\FLLPIP created in chapter 33 and:
 a) paper space to erase the viewports leaving the 3D view
 b) model space with UCS BASE and layer MODEL current
 c) load the linetype HIDDEN
 d) set the LTSCALE variable value to a suitable value
 e) set the following hatch variables:
 HPNAME: ANSI32; HPANG: 0; HPSCALE: 1
 f) paper space scale the 3D viewport by 0.5 and move to the top left of the drawing sheet
 g) model space and zoom the model in the 3D viewport to a suitable size
 h) refer to Fig. 43.3 for the viewport layouts

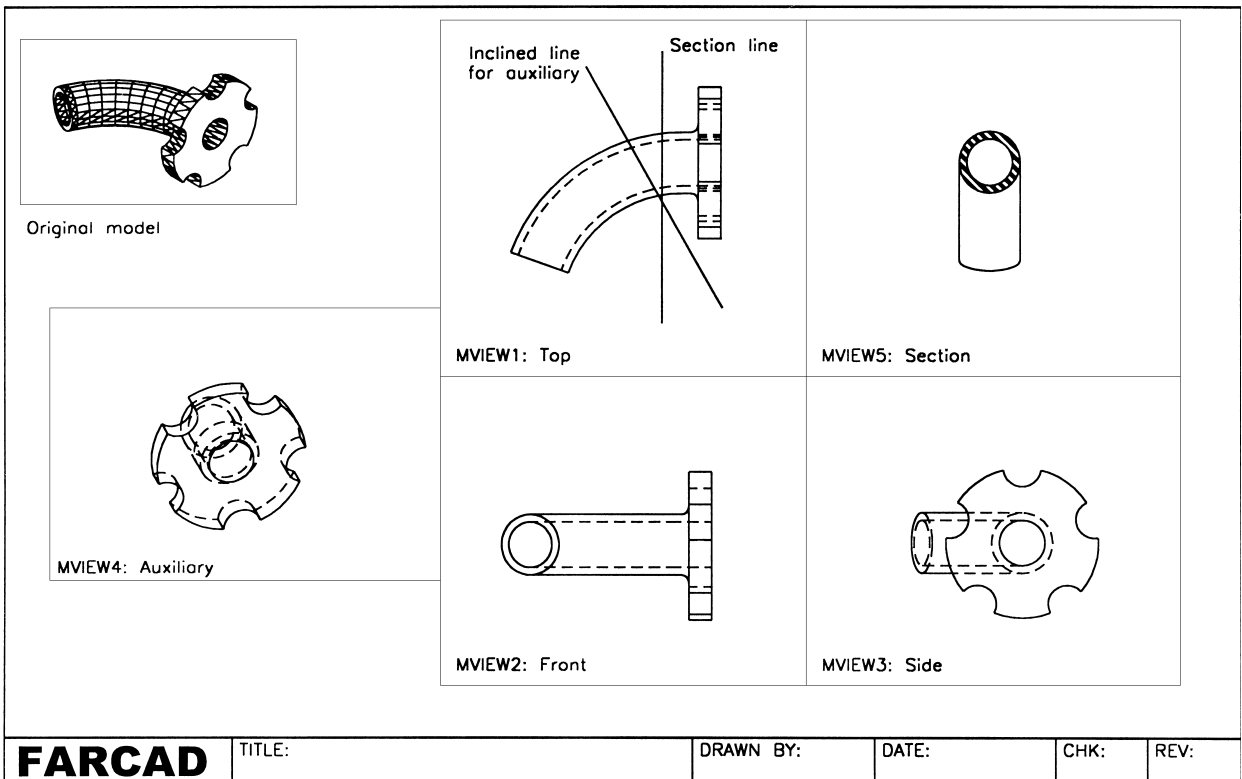


Figure 43.3 Setup view and drawing example 3 – the flange/pipe model.

- 2 Activate the setup View command and:
 - a) UCS option with UCS BASE
 - b) view scale: 0.25
 - c) view centre: 200,200
 - d) position viewport: pick to suit yourself
 - e) name: MYVIEW1
 - f) options: X

- 3 Repeat the setup View command:

prompt Enter an option [Ucs/Ortho/Auxiliary/Section]
enter **O <R>** – the ortho option
prompt Specify side of viewport to project
respond **pick lower horizontal line of viewport**
prompt Specify view center
enter **200,100 <R>**
prompt Specify view centre
and **pick to suit yourself**
prompt Specify corners of viewport
respond **pick to suit your layout**
prompt Enter view name and enter: **MYVIEW2 <R>**
prompt Enter an option and continue with next part of exercise

- 4 Setup view command still active with options:
 - a) select the Ortho option
 - b) pick right vertical line of second viewport
 - c) view center: pick a point to right to suit
 - d) viewport corners: pick points to suit
 - e) view name: MYVIEW3
 - f) options
 - g) select the Auxiliary option
 - h) Inclined plane points: pick to suit – see Fig. 43.3
 - i) Side to view from: pick 'below' the inclined line
 - j) view center: pick a point to suit
 - k) viewport corners: position to suit
 - l) view name: MYVIEW4
 - m) options
 - n) select the section option
 - o) cutting plane points: pick points as indicated in Fig. 43.3
 - p) side to view from: pick to right of the section line
 - q) view scale: 0.25
 - r) view center: pick to suit
 - s) viewport corners: pick to suit the layout
 - t) name: MYVIEW5
 - u) options: X to end command

- 5 Linetype HIDDEN and three HP variables set?

- 6 Activate the Setup Drawing command and pick the five viewports to display the layout with hidden line removal and section detail as Fig. 43.3

- 7 Freeze layer VPORTS if required

- 8 *Question*
 In our three setup examples the type of layout has not been mentioned, i.e. first or third angle. I am sure that you are aware of the projections obtained, the first two examples being in first angle, and the third example being in third angle. This is the 'power' of the setup commands. First and third angle detail layouts can be obtained for models by simply picking the relevant view centre points

- 9 Save the exercise as it is now complete.

Summary

- 1 The set View and Drawing commands allow the user to layout multi-view drawings without the need to create viewports and set viewpoints
- 2 Both commands can be activated:
 - a) by selecting the icon from the Solids toolbar
 - b) from the menu bar with Draw-Solids-Setup
 - c) from the command line with SOLVIEW and SOLDRAW
- 3 The setup View command has options which allow views to be created:
 - a) relative to a named UCS
 - b) as an orthographic view relative to a selected viewport
 - c) as an auxiliary view relative to an inclined plane
 - d) as a section view relative to a cutting plane
- 4 When used, the View command creates viewport specific layers, these being relative to the viewport handle number with the following names:
 - VIS for visible lines
 - HID for hidden lines
 - DIM for dimensions
 - HAT for hatching but only if the section option is used
- 5 The View command requires the user to:
 - a) enter the view scale
 - b) position the viewport centre point
 - c) position the actual viewport corners
- 6 With the Ortho option, both First and Third angle projections can be obtained dependent on which side the new viewport is to be placed
- 7 The section option requires that the system variables HPNAME, HPANG and HPSCALE are set. It is usual to use the ANSI31 hatch pattern name, but this is not essential. AutoCAD defaults the ANGLE hatch pattern
- 8 The Drawing command will display models which have been created with the View command:
 - a) with visible and hidden detail
 - b) as a section if the section option has been used
- 9 It is recommended that the linetype HIDDEN be loaded before the Drawing command is used.
- 10 The hidden linetype appearance is controlled by the LTSCALE system variable.
- 11 The user now has two different methods for creating multi-view layouts of solid models:
 - a) using the A3SOL template file idea which sets the viewports and viewpoints prior to creating the model. Profiles can then be extracted to display hidden detail
 - b) using the VIEW and DRAWING commands with a solid composite to layout the drawing in First or Third angle projection with sections and auxiliary views as required.
 - c) it is now the user's preference as to which method is used
- 12 Dimensions can now be added to models:
 - a) using viewport specific layers
 - b) using paper space dimensioning
 - c) using the setup commands

The final composite

We have now covered virtually every concept of solid modelling within the AutoCAD draughting package. The next chapter will introduce the user to rendering, but before that we will make a final solid model using the various techniques that have been discussed. The model is quite involved, so try not to miss out any of the steps, especially those which set a new UCS position.

The three examples selected to demonstrate the View and Drawing commands have used previously created models. This example will create a new model 'from scratch'.

- 1 Open your A3SOL template/drawing file with layer MODEL, UCS BASE and make the model tab active. We will use this tab to create the model and then set up our drawing layout with the MVLAY1 tab
- 2 Set ISOLINES to 6 and refer to Fig. 44.1
- 3 The new model will be created from five primitives, each requiring a new UCS position.

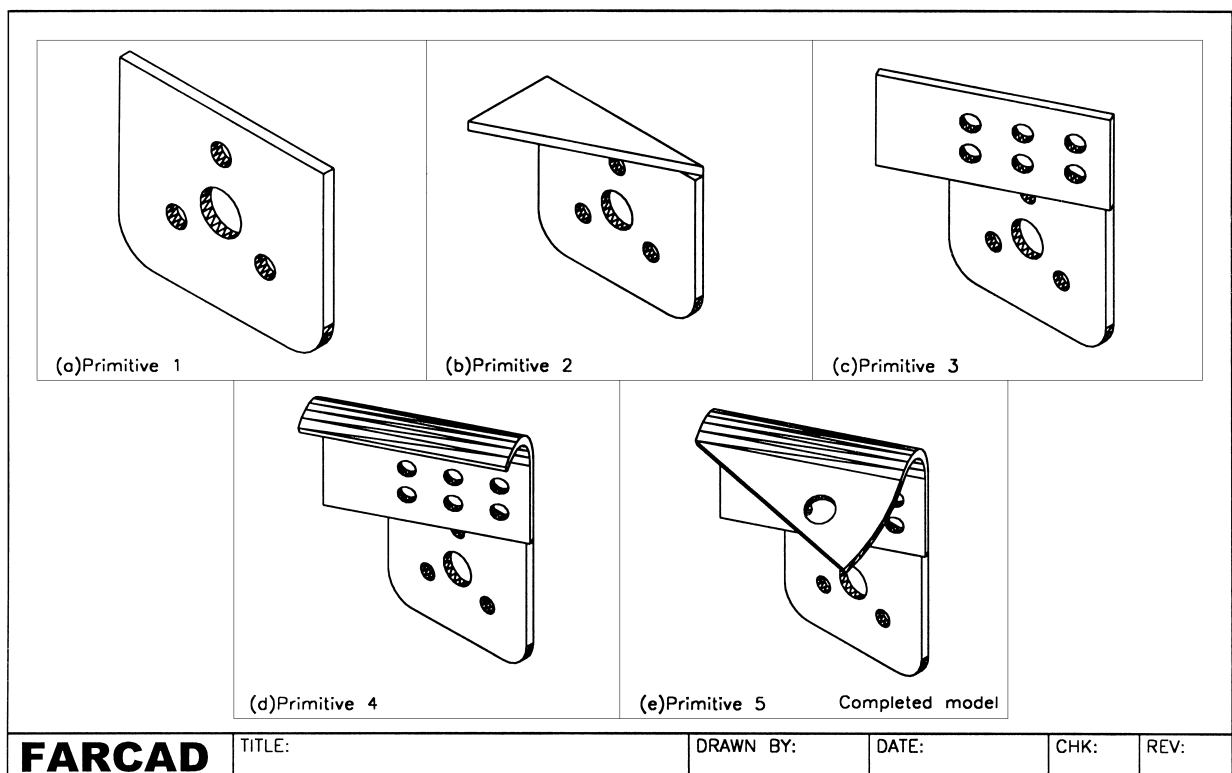


Figure 44.1 Construction of the computer link model.

Primitive 1: the base

- 1 Rotate the UCS about the X axis by 90 and save as PRIM1
- 2 Draw a polyline:
 - a) start point: 0,50
 - b) next point: @60,0
 - c) next point: @0,-40
 - d) arc option with endpoint: @-10,-10
 - e) line option to: @-40,0
 - f) arc option with endpoint: @-10,10
 - g) line option to: close
- 3 Zoom extents then zoom to a scale of 3
- 4 Solid extrude the polyline for a height of 3 with 0 taper
- 5 Create two cylinders:
 - a) centre: 30,25,0; radius: 6; height: 3
 - b) centre: 30,40,0; radius: 3; height: 3
- 6 Polar array the smaller cylinder about the point 30,25 for 3 items with full circle rotation
- 7 Subtract the four cylinders from the extruded polyline – fig(a)

Primitive 2: wedge on top of first primitive

- 1 UCS PRIM1 current
- 2 Set a new 3 point UCS position with:
 - a) origin: 0,50,0
 - b) x-axis: 60,50,0
 - c) y-axis: 0,50,3
 - d) save as: PRIM2
- 3 Create a wedge with:
 - a) corner: 0,0,0
 - b) length: 60; width: -3; height: -30
- 4 Rotate 3D this wedge:
 - a) about the X axis
 - b) with 0,0,0 as a point on the axis
 - c) for 90 degrees
- 5 Union the wedge and the extruded polyline – fig(b)

Primitive 3: box on top of wedge

- 1 UCS PRIM2 current
- 2 Set a new 3 point UCS position with:
 - a) origin: 60,0,0
 - b) x-axis: 0,30,0
 - c) y-axis: 60,0,-3
 - d) save as: PRIM3
- 3 Create a solid box with
 - a) corner: 0,0,0
 - b) length: 67.08; width: 30; height: -3. Why 67.08?

- 4 Create a cylinder with:
 - a) centre: 10,10,0 and:
 - b) radius: 3; height: -3
- 5 Rectangular array the cylinder:
 - a) for 2 rows and 3 columns
 - b) row offset: 10; column offset: 15
- 6
 - a) subtract the six cylinders from the box
 - b) union the box and the composite – fig(c)

Primitive 4: curved extension on top of box

- 1 Pan the model to lower part of screen
- 2 UCS PRIM3 current
- 3 Set a new 3 point UCS position with:
 - a) origin: 0,30,-3
 - b) x-axis: 67.08,30,-3
 - c) y-axis: 0,30,0
 - d) save as: PRIM4
- 4 Zoom in on the 'free edge' of the box
- 5 Draw two line segments with:

start point: 0,0 next point: @0,-15; next point: @50,0
- 6 Draw a polyline about the 'top rim' of the box using the ENDpoint snap and the close option
- 7 With the solid revolve command:
 - a) objects: enter L <R><R> – to select the polyline
 - b) options: enter O <R> – object option
 - c) object: pick the left end of long construction line
 - d) angle: enter 120
- 8 Erase the two line segments
- 9 Zoom previous to restore original view
- 10 Union the revolved component and the composite – fig(d)

Primitive 5: final curved component

- 1 UCS PRIM4 current
- 2 Set a new 3 point UCS position with:
 - a) origin: 67.08,-22.5,-12.99
 - b) x-axis: 0,-22.5,-12.99
 - c) y-axis: 67.08,-24,-15.59
 - d) save as: PRIM5
 - e) can you work out the three sets of coordinates?
- 3 Zoom-in on the 'free end' of the curved component
- 4 Draw a polyline about the free end of the curved component using the ENDpoint snap and the close option

- 5 With the Solid revolve command:
 - a) objects: enter L <R><R> – to select the polyline
 - b) options: enter Y <R> – the Y axis
 - c) angle: enter -30
- 6 Create a cylinder with:
 - a) centre: 45,0,15
 - b) radius: 5
 - c) centre of other end: @0,10,0
- 7 Subtract the cylinder from the revolved component, then union the revolved component with the cylinder – fig(e)
- 8 Zoom previous to restore the original view
- 9 The model is now complete, so:
 - a) Gouraud shade and 3D orbit – impressive?
 - b) restore 2D wireframe representation at the original viewpoint
- 10 a) restore UCS BASE
 - b) save as MODR2002\COMPLINK

Laying out the viewports

This part of the exercise will use the MVLAY1 tab with all options of setup View command.

- 1 Pick the MVLAY1 tab name
- 2 In paper space:
 - a) erase three viewports but leave the 3D viewport
 - b) stretch (crossing option) the two vertical edges of the 3D viewport by @50,0 (left side) and @-50,0 (right side)
 - c) move the 3D viewport as far left as possible
 - d) in model space, UCS BASE, layer Model current and zoom the model to suit
- 3 a) load linetype HIDDEN
 - b) set the following variables:
HPNAME: ANSI31; HPANG: 0; HPSCALE: 0.5
 - c) set the LTSCALE value to suit which may change after the layout has been created
- 4 Activate the setup View command with:
 - a) UCS option with BASE
 - b) view scale: 1
 - c) view center: 175,75
 - d) viewport corners: pick to suit
 - e) view name: TOP
 - f) options: X
- 5 Using the SOLVIEW command:
 - a) UCS option with PRIM1 as the named UCS
 - b) view scale: 1
 - c) view center: 175,200
 - d) viewport corners: pick to suit
 - e) view name: FRONT
 - f) exit command or continue with command

- 6 SOLVIEW command with:
 - a) Ortho option
 - b) side: pick right vertical side of the second viewport
 - c) view center: 50,200
 - d) viewport corners: pick to suit
 - e) view name: RIGHT
 - f) exit command or continue with command

- 7 With the setup View command:
 - a) activate the Section option
 - b) pick any point in the second viewport
 - c) cutting plane points: 30,0 and 30,120
 - d) side to view from: pick a point to left of section line
 - e) view scale: 1
 - f) view center: 0,-110
 - g) viewport corners: pick to suit
 - h) view name: SECTION
 - i) exit or continue with command

- 8 The final SOLVIEW command is with:
 - a) the Auxiliary option
 - b) first point of inclined plane and with the first viewport active, pick ENDpoint of pt1 (see Fig. 44.2)
 - c) second point of inclined plane: PERP to line 23
 - d) side to view from: pick to left of inclined line
 - e) view center: 0,200
 - f) viewport corners: pick to suit
 - g) view name: AUXILIARY
 - h) end the command

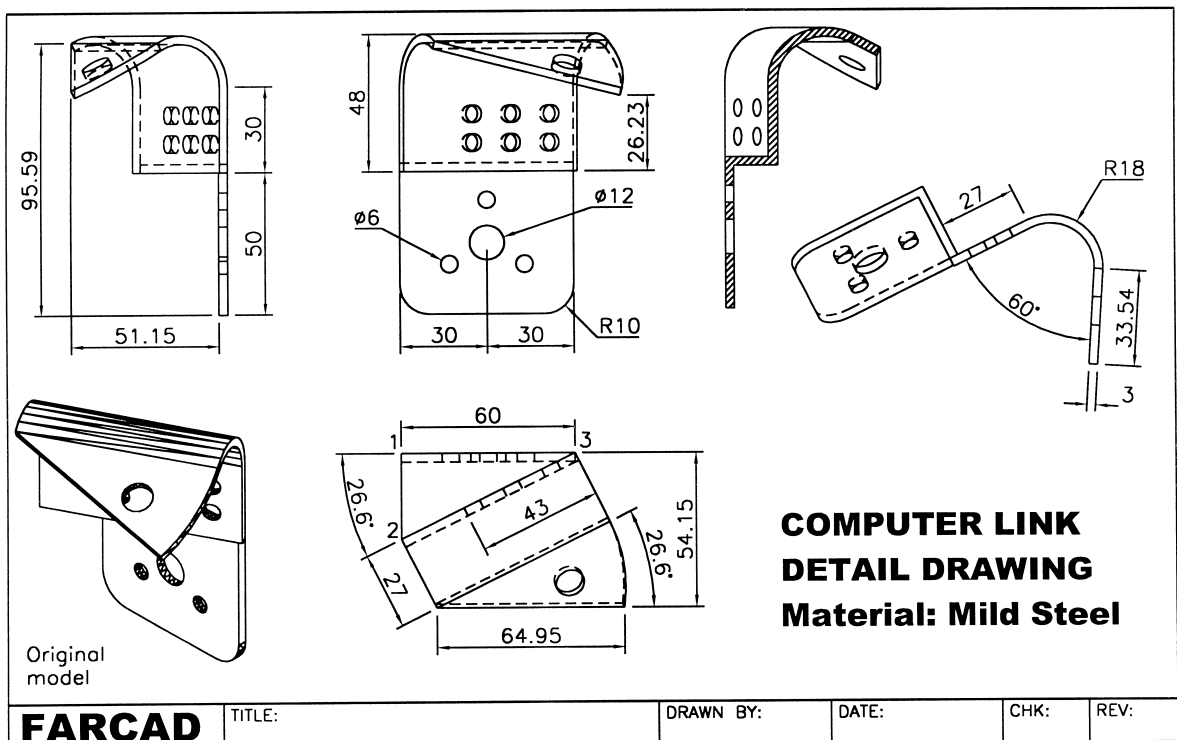


Figure 44.2 Computer link detail drawing using the setup commands.

- 9 Using the setup Drawing command, pick the five viewports to display hidden detail and a section view – linetype HIDDEN loaded?
- 10 Now optimise the LTSCALE system variable
- 11 *Tasks*
 - a) Interrogate the model in the 3D viewport:
Area: 20196.01
Mass: 26758.64
 - b) Using the viewport specific -DIM layers, add the dimensions displayed in Fig. 44.2. The UCS in the new created viewports should be 'set' to allow this as the UCSVP system variable always defaults to 1 when a new viewport is created. Note that a paper space zoom of the viewport being dimensioned will assist with the dimensions.
 - c) Freeze the VP and VPORTS layers
 - d) In paper space, optimise your drawing with suitable text
 - e) Save the completed exercise – worth the effort?

Assignment

To give some additional practice with the View and Drawing commands I have included another new model to be created.

Activity 25 – dispenser of MACFARAMUS

One of the discoveries in the city of CADOPOLIS was a container which was thought to be a dispenser belonging to MACFARAMUS. It is this container which has to be created as a solid model and then displayed with the setup commands.

- 1 Use your template file with the Model tab active and when the model is complete use the MVLAY1 tab as the chapter example
- 2 Make two new layers BODY blue and TOP green
- 3 With UCS FRONT, draw the two shapes using the reference sizes in Fig. 44.3. Use the start points given. Draw as lines/arcs then use the join option of the modify polyline command to convert the segments into single polylines.

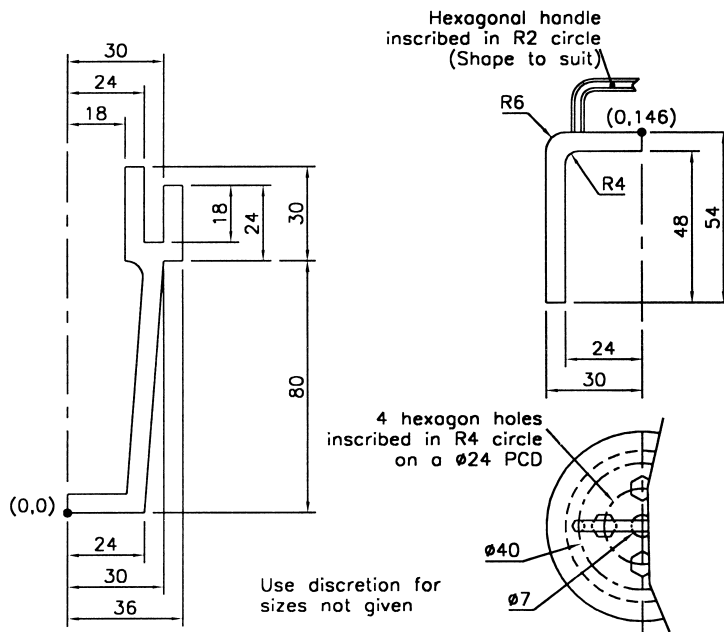


Figure 44.3 Reference details for activity 25.

- 4 Solid revolve the two polylines for a full circle
- 5 Create the holes in the top
- 6 Create a handle from a hexagon, the actual shape being at your discretion – I extruded a hexagon along a polyline path
- 7 Use the VIEW and DRAWING commands to create a multi-view layout to display:
 - a) top and front views with hidden detail
 - b) two section views through the planes indicated
 - c) an auxiliary view through an inclined plane at 45 degrees
 - d) a 3D view
 - e) I used a view scale of 0.6
- 8 This activity should highlight a problem when the setup commands are used with a model containing more than one part. The dispenser has a top and a body, but when the section option is used, the same hatching is added to both parts. How would different hatching be added to the two parts?

Rendering

Rendering is a topic with its own terminology and we will discuss this terminology by rendering previously created models. The reader should realise that this chapter is only a brief introduction to rendering. As we have created several interesting models, it seems reasonable that we investigate the next step in the modelling process, i.e. the production of rendered images.

What is render?

Rendering is a process which creates an image (usually in colour) of a 3D surface or solid model. This image is created from *a scene using a view with lights*.

How is render activated?

AutoCAD render is automatically loaded into memory when the RENDER command (or any render option) is selected. The RENDER command can be activated with:

- a) the menu bar selection **View-Render-Render**
- b) the RENDER icon from the Render toolbar
- c) entering **RENDER <R>** at the command line



The three methods give the Render dialogue box as Fig. 45.1, which (at present) has three main areas:

1. the rendering types
2. the scene which is to be rendered
3. the rendering destinations

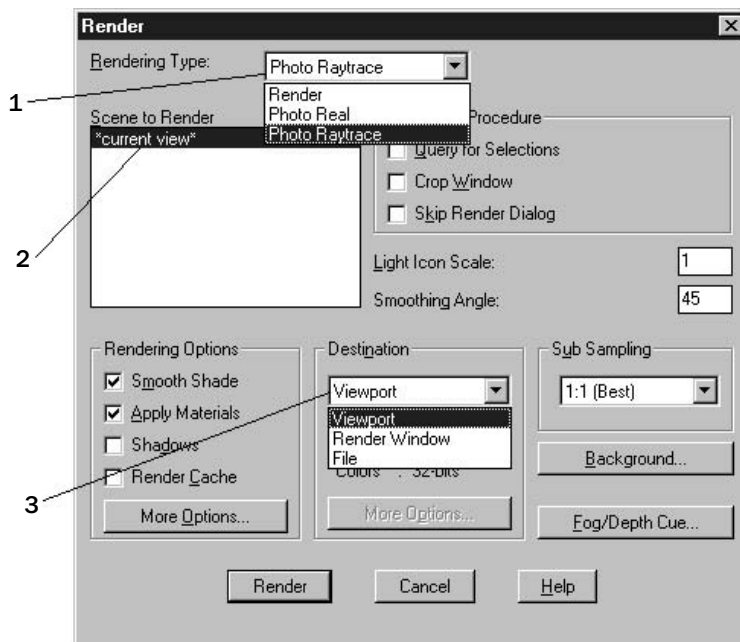


Figure 45.1 The Render dialogue box.

Rendering types

AutoCAD has three 'types' of rendering, these being:

1. Render: the basic AutoCAD render option which allows models to be rendered without the need for scenes, lights or materials. This is the default 'setting'
2. Photo Real: is a photo-realistic renderer which can display bitmapped and transparent materials. Volumetric and mapped shadows can also be generated.
3. Photo Raytrace: is a photo-realistic raytrace renderer which can generate reflections, refraction and precise shadows.

Rendering destination

The rendered image of the model can be:

1. displayed in the current viewport of a multi-screen layout
2. displayed in the render window
3. saved to a file for future recall

Scene to render

Allows the user to select a named scene of the model to render. The user can 'make' several scenes using different views of the model with various lights and materials added. AutoCAD provides a 'default' scene for rendering purposes. This default scene uses a distant light which cannot be modified by the user. At all times it should be remembered that:

***** A scene is a view with lights added *****

The AutoCAD lights

Adding lights to a model layout immediately improves the render appearance and lights can be used to illuminate a complete model or to highlight specific parts of the layout. AutoCAD 2002 has four 'types' of light available, these being ambient, distant, point and spot.

Ambient light

- 1 Provides a constant illumination to all surfaces of a model.
- 2 It is always 'there' and does not originate from any particular source.
- 3 The user has control of the intensity of the ambient light
- 4 Generally the ambient light intensity should be a low value or the model will be displayed 'too bright'
- 5 The default ambient intensity value is 0.3
- 6 Ambient light on its own does not produce good rendered images.

Distant light

- 1 Gives a parallel beam in a particular direction
- 2 The user specifies the target point and the light source location
- 3 Think of a torch shining at an object
- 4 Distant light rays extend to infinity on either side of the light source
- 5 The distant light intensity is **not** affected by the distance of the source from the target
- 6 It is recommended that distant lights are positioned at the extents of the drawing
- 7 Distant lights are used to give a 'uniform lighting' facility
- 8 A single distant light simulates the sun

Point light

- 1 A point light emits light in all directions from its position
- 2 The user specifies the point light location
- 3 Think of a light bulb
- 4 The intensity of a point light is affected by the distance from the model
- 5 Point lights are used for general lighting effects
- 6 Point lights are used with spot lights for lighting effects

Spot light

- 1 Gives a directional cone of light
- 2 The user specifies the direction of the light and the size of the cone
- 3 The intensity of a spot light diminishes with the distance from the model
- 4 Spot lights have 'hot-spots' and 'fall-off angles' that determine how the light diminishes at the edge of the cone
- 5 Spot lights can be used to highlight specific features on a model.

Point lights, distant lights and spot lights are represented in a drawing with symbols, the light name being displayed within the light symbol – Fig. 45.2

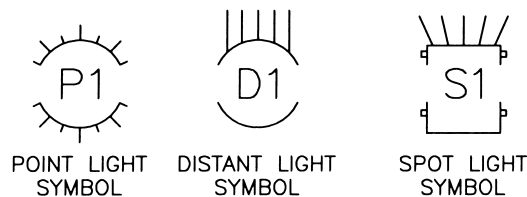


Figure 45.2 Light symbols with 'names' added.

Note

- 1 Lights are essential for rendering and their position in relation to the model is very important. AutoCAD will position a light in the centre of the active viewport, irrespective of the model position. The user must know exactly where the light is to be position relative to the model. This can be achieved using coordinates and object snaps. It is also important for the user to know the basic sizes of the model and its where the model is situated on the drawing screen. This is one of the main reasons that I use the 0,0,0 origin point when creating models.
- 2 The basic 'order' with rendering is:
 - a) create the model
 - b) make a view for a particular model 3D viewpoint
 - c) position lights – with or without shadows
 - d) add materials to the model parts
 - e) make a scene from a view and lights
 - f) render with a type and to a destination

Models to be rendered

Two previously created solid models will be used to demonstrate how lights, scenes, materials, shadows, etc. can be added to a layout and produce a rendered coloured image. The models selected are the extruded backing plate and the wall clock.

Render example 1 – the solid model backing plate

When rendering a model, the user will activate several dialogue boxes. It is not my intention to display these dialogue boxes (other than Render) in the exercise. The necessary steps to activate parts of these dialogue boxes will be given, and the user can investigate other ‘parts’ of these dialogue boxes at their leisure.

- 1 Open your saved drawing MODR2002\BACKPLT from chapter 32
- 2 Make the model tab active and set a SE Isometric viewpoint
- 3
 - a) Ensure UCS BASE and layer Model current
 - b) Ensure that the model is positioned with the mid-point of the front bottom edge at the origin. It should be, but if it is not, move the model so that it is. This is important for positioning the lights
 - c) Display the Render toolbar
- 4 For maximum effect we want the lights to ‘cast shadows’ and we will create a base for the model to stand on and a back wall, so from the menu bar select Draw-Solids-Box and create two box primitives with:

	<i>box 1</i>	<i>box 2</i>
a) corner:	-70,-20,0	-70,78,0
b) length:	200	200
c) width:	100	2
d) height:	-2	100
e) colour:	number 41	number 41
- 5 Union the two box primitives then zoom extents. The complete model layout should be displayed
- 6 *The view*

As stated earlier, once the model has been created a view should be saved, and we will use our existing screen layout for this, so at the command line enter **VIEW <R>** and:

```
prompt View dialogue box
respond 1. pick Named View tab
        2. pick New
prompt New View dialogue box
respond 1. View name: enter V1
        2. Current display and Save UCS with view active
        3. UCS name: BASE
        4. pick OK
prompt View dialogue box
with V1 listed with details
respond pick OK
```
- 7 Note that -VIEW <R> will allow command line entry

8 The lights

Select the LIGHTS icon from the render toolbar and:



```

prompt Lights dialogue box
respond 1. set Ambient Light Intensity to 0.3
        2. scroll at New and pick Point Light
        3. pick New

prompt New Point Light dialogue box
respond 1. Light name: enter P1
        2. Intensity: alter to 70
        3. pick Modify

prompt Drawing screen returned with rubber band effect to light
        position which is at the 'centre of the viewport' i.e.
        the screen in this case

and Enter light location<current>
enter 0,-50,100 <R> i.e. in front of the model

prompt New Point Light dialogue box
respond 1. activate Shadows on – tick
        2. pick Shadows Options

prompt Shadows Options dialogue box
respond 1. ensure Shadow Volumes/Ray Traced Shadows active
        2. pick OK

prompt New Point Light dialogue box
respond pick OK

prompt Lights dialogue box with P1 listed
respond 1. scroll at New and pick Distant Light
        2. pick New

prompt New Distant Light dialogue box
respond 1. Light name: enter D1
        2. Intensity: alter to 0.9
        3. pick Modify

prompt Drawing screen returned with rubber band effect to light
        position at centre of screen

and Enter light direction TO
enter 50,0,30 <R>

prompt Enter light direction FROM
enter @20,-30,30 <R>

prompt New Distant Light dialogue box
respond 1. activate Shadows on
        2. pick Shadows Options

prompt Shadows Options dialogue box
respond 1. ensure Shadow Volumes/Ray Traced Shadows active
        2. pick OK

prompt Distant Light dialogue box
respond pick OK

prompt Lights dialogue box with D1 and P1 listed
respond pick OK

```

9 The scene

Menu bar with **View-Render-Scene** and:

```

prompt Scenes dialogue box
respond pick New

prompt New Scenes dialogue box
respond 1. enter Scene name: SC1
        2. pick Views: V1
        3. pick Lights: *ALL*
        4. pick OK

prompt Scenes dialogue box with SC1 listed
respond pick OK

```

10 *The first render*

At the command line enter **RENDER <R>** and:

prompt Render dialogue box

respond 1. Rendering Type: scroll and pick Photo Real
 2. Scene to Render: pick SC1
 3. Destination: scroll and pick Render Window
 4. Rendering Options: ensure Smooth Shade, Apply Materials and Shadows are active (tick)
 5. pick Render

prompt Render window will be displayed with a coloured image of the backing plate

and The rendered image should be reasonably impressive and the user should observe:

- a) two shadow effects
- b) a white background
- c) the light effect on the model being 'too bright'

respond pick AutoCAD from the Windows taskbar to return to the drawing screen

11 The user has now to decide whether the lights are in the correct position, is the intensity set correctly etc. We will leave the light settings as they are.

12 *The background*

We now want to add a background to enhance the display and AutoCAD allows rendered images to be displayed with four 'types' of background, these being:

- a) the default white background
- b) a one coloured background
- c) a gradient background of three colours
- d) a 'picture' background of an already saved image

13 Select the BACKGROUND icon from the Render toolbar and:

prompt Background dialogue box

respond 1. Select Gradient
 2. Accept the RGB colour settings
 3. Alter Horizon: 0.7; Height: 0.5; Rotation: -20
 4. pick Preview then OK

14 Render scene SC1 with Photo Real to the Render Window and the model image will be displayed with the set gradient background

15 Return to the AutoCAD screen

16 *Attaching a material*

The rendered image of the model is displayed with the colours of the primitives from which it was created. AutoCAD has a library which allows the user to attach different materials to surface and solid models, so select the Materials Library icon from the Render toolbar and:

prompt Materials Library dialogue box

with a) Materials in Current Drawing – Global
 b) Current Library list

respond 1. scroll at Current Library list
 2. pick WOOD-WHITE ASH
 3. scroll at Preview, pick Cube then pick Preview
 4. pick <-Import

and WOOD-WHITE ASH added to Current Drawing list

respond pick OK

17 *Note*

1. The current library list will display either a few material names or a large number of names. If only a few names are displayed then it will be necessary to open an **MLI** file. This is achieved by:
 - a) picking Open from the Materials Library dialogue box
 - b) scrolling and opening the render.mli files from Support
2. Although WHITE ASH has been selected, any material can be used for this exercise.

18 Menu bar with **View-Render-Materials** and:

prompt Materials dialogue box

respond 1. pick WOOD-WHITE ASH
2. pick Attach<

prompt Select objects to attach "WOOD-WHITE ASH" to:

respond pick the composite and right-click

prompt Materials dialogue box

respond pick OK

- 19 Now render scene SC1 to the render window to display the model with a material and a background. The result should be quite impressive?
- 20 The rendered display is a bit-map image and can be saved for future recall into AutoCAD or to other graphics type packages, so from the render menu bar select **File-Save** and:

prompt Save BMP dialogue box

respond 1. named folder should be current
2. enter file name: **BACKPLT**
3. pick OK
- 21 Return to AutoCAD and save the screen layout with a suitable name. This will save the model and the lights, scene, materials etc
- 22 A screen dump of the model is displayed in Fig. 45.3
- 23 This first render exercise is now complete.

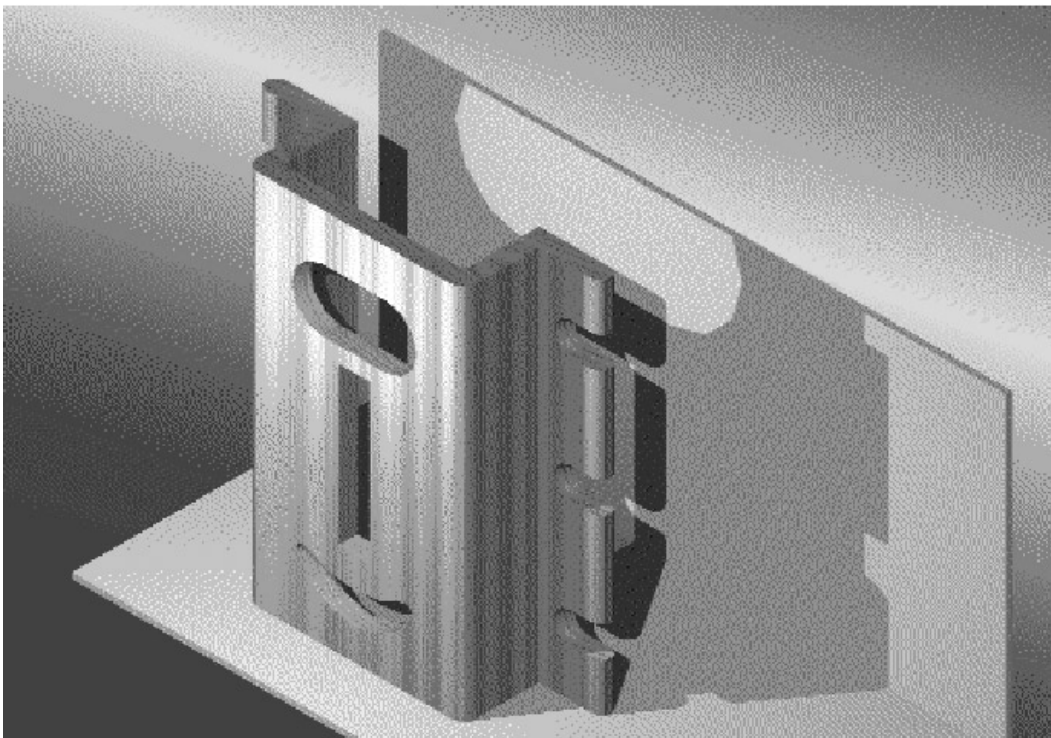


Figure 45.3 The rendered BACKING PLATE with materials attached.

Render example 2 – the wall clock

Preparing a model for rendering follows the basic procedure used with example 1, i.e. save a view, position lights, create a scene, attach materials, render the scene. For our second example we will use the wall clock model and attach several materials to the various parts of the model.

- 1 Open model CLOCK of the wall clock created in chapter 41
- 2 Make the model tab active
- 3 Restore UCS BASE and check that the front lower vertex of the clock body is positioned at 0,0. If it is not, then move it.
- 4 Create a box primitive with:
 - a) corner: -90,40,-50
 - b) length: 180; width: 2; height: 240
 - c) colour: number 201
- 5 Now zoom extents and at the command line enter **-VIEW <R>** and:


```
prompt Enter an option
enter   S <R> – the save option
prompt Enter view name to save
enter   V1 <R>
```
- 6 Position the following three lights:
 1. Point light with:
 - a) name: P1
 - b) intensity: 100
 - c) location: 0,20,300
 - d) no shadow effect
 2. Point light with:
 - a) name: P2
 - b) intensity: 50
 - c) location: 0,-90,150
 - d) shadows on: shadow volumes/ray trace shadows
 3. Distant light with:
 - a) name: D1
 - b) intensity: 0.75
 - c) direction TO: -50,-20,150
 - d) direction FROM: @-50,-50,50
 - e) shadows (volumes/ray traced) on
- 7 Make a scene named SC1 with view V1 and all three lights
- 8 Using the materials library icon, import the materials COPPER, MARBLE GREEN, PINK MARBLE, WOOD-WHITE ASH, BLUE PLASTIC and CHECKER TEXTURE
- 9 With the Materials icon, attach the imported materials to the following parts of the model:
 - a) COPPER: pendulum, spindle and mechanism
 - b) MARBLE GREEN: octagonal face
 - c) PINK MARBLE: circular dial
 - d) WOOD WHITE ASH: body
 - e) BLUE PLASTIC: one hand
 - f) CHECKER TEXTURE: other hand

- 10 Set a gradient background of your choice
- 11 Render scene SC1 with:
 - a) photo real
 - b) render window
 - c) shadow options on
- 12 Save the rendered image to your named folder
- 13 The rendered image of the clock is displayed in Fig. 45.4

This chapter has introduced the reader to how models can be rendered. Hopefully you will investigate the topic in more detail as rendering is a fascinating subject and the results really enhance the final appearance of models.

Note.

AutoCAD 2002 allows the 3D orbit command to be used with materials attached. If you have tried the 3D orbit command it not have 'retained' the materials during the real-time rotation. To achieve this:

- a) a good quality-high memory graphics card is required
- b) certain additional settings must be activated

To demonstrate the effect:

- 1 Open the saved backing plate model which has a wood material attached
- 2 Activate the model tab

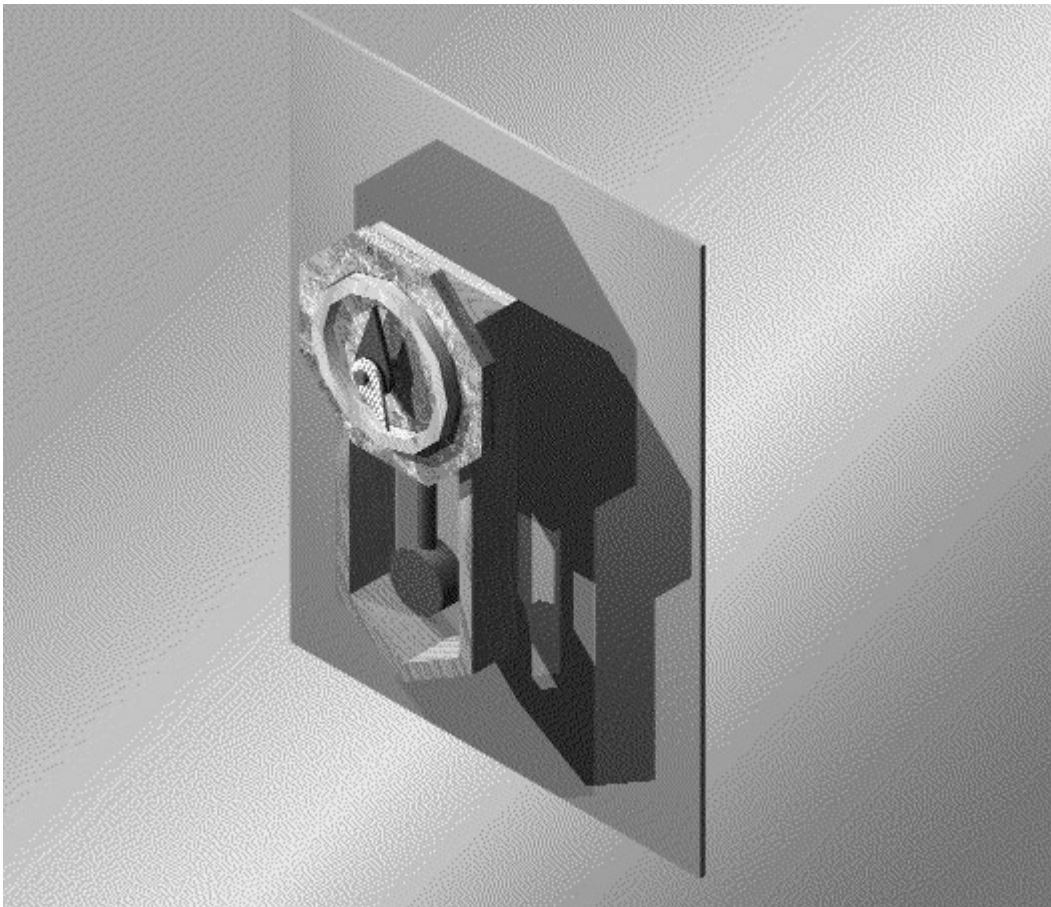


Figure 45.4 The rendered WALL CLOCK with materials attached.

- 3 Menu bar with **Tools-Options** and:
 - prompt* Options dialogue box
 - respond* pick the System tab
 - prompt* System selections
 - respond* pick Properties at the Current 3D Graphics Display
 - prompt* 3D Graphics System Configuration dialogue box
 - respond*
 - 1) Render options active
 - 2) Enable lights and materials active
 - 3) Enable textures with High Quality/Slower active
 - 4) Acceleration: Hardware active
 - 5) at Driver, select Browse and:
 - prompt* Select file (drivers) dialogue box
 - respond*
 - 1) scroll and pick **wopengl7**
 - 2) pick Select
 - prompt* 3D Graphics Configuration dialogue box returned
 - respond*
 - 1) refer to Fig. 45.5 and activate other settings
 - 2) pick Apply & Close
 - prompt* Options dialogue box
 - respond* pick OK
- 4 Now:
 - a) render your model with the Viewport as the destination
 - b) activate 3D orbit and rotate the model
 - c) materials kept with rotation?
 - d) if they do not, you may need a 'better quality' graphics card, although I would **not recommend** you rush out and buy one.

This chapter has introduced the reader to how models can be rendered. Hopefully you will investigate the topic in more detail as rendering is a fascinating subject and the results really enhance the final appearance of models.

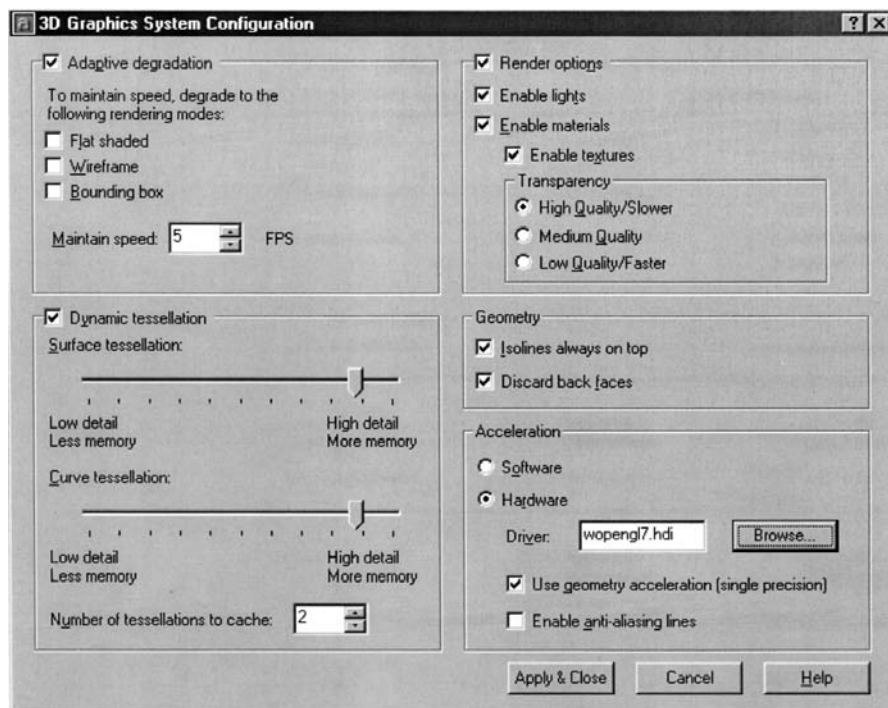


Figure 45.5 The 3D Graphics Systems Configuration dialogue box.

Publishing to the web

AutoCAD 2002 allows the user to create Web pages of existing drawings. To demonstrate the concept:

- 1 Open any model (3D or solid) and:
 - a) activate the Model tab
 - b) view the model in 3D
 - c) save with a suitable name and note the name
- 2 Repeat step 1 for some other models (3 or 4)
- 3 Start a new metric drawing from scratch to display the typical AutoCAD 2002 blank screen
- 4 Menu bar with **File-Publish to Web** and:
 - prompt* Publish to Web (Begin) dialogue box (as Layout dialogue box)
 - respond* 1. pick Create New Web Page
 - 2. pick New
 - prompt* AutoCAD message similar to Fig. 46.1
 - respond* 1. read the message
 - 2. pick Cancel

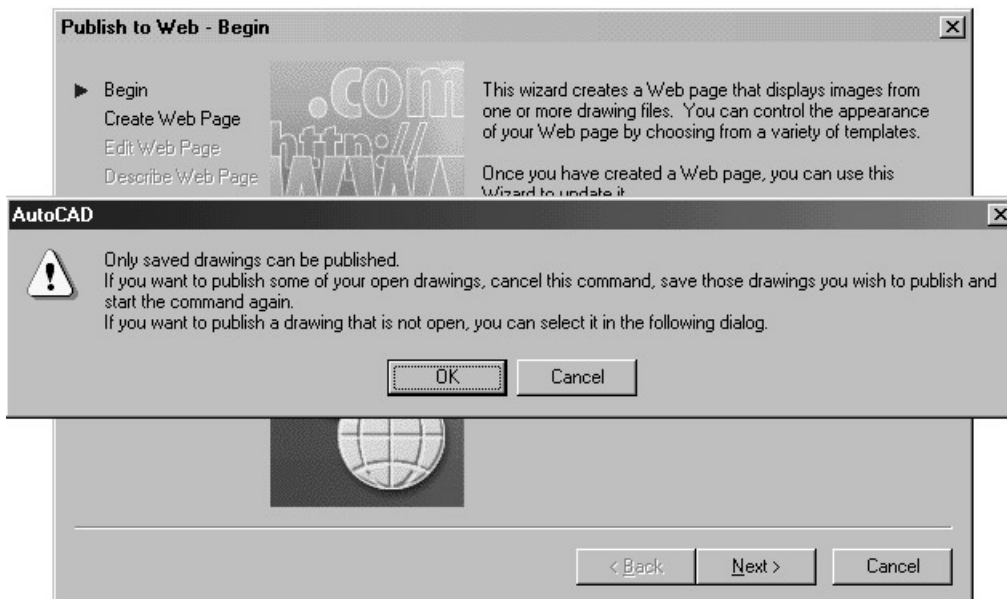


Figure 46.1 The AutoCAD message with Publish to Web.

- 5 Now open one of the previously opened/saved drawings, then menu bar with **File-Publish to Web** and:

prompt Publish to Web (Begin) dialogue box
respond 1. pick Create New Web Page
 2. pick New

prompt Publish to Web (Create Web Page) dialogue box
respond 1. Web page name: enter MYSOLID
 2. Note parent directory
 3. Add any suitable description – Fig. 46.2
 4. pick Next

prompt Publish to Web (Select Image Type) dialogue box
respond 1. select type from list : DWF
 2. pick Next

prompt Publish to Web (Select Template) dialogue box
respond 1. select a template type, e.g. Array plus Summary
 2. pick Next

prompt Publish to Web (Apply Theme) dialogue box
respond 1. scroll and select an element, e.g. Ocean Waves – Fig. 46.3
 2. pick Next

prompt Publish to Web (Enable i-drop) dialogue box
respond activate enable i-drop then pick Next

prompt Publish to Web (Select Drawings) dialogue box
with the opened drawing listed by name
respond 1. Layout: select Model (probably active)
 2. Label: alter to some suitable name
 3. Description: enter to suit
 4. pick Add->

and entered label added to Image list
respond pick the (...) at Drawing

prompt Publish to Web dialogue box with *.dwg file type
respond 1. scroll to your named folder if not active
 2. open another drawing/activity
 3. Layout: Model
 4. Label: alter to suit
 5. Description: enter to suit
 6. pick Add->

then select (...) another 2/3 drawings and repeat the above six responses
and dialogue box similar to Fig. 46.4
then pick Next

prompt Publish to Web (Generate Images) dialogue box
respond 1. Regenerate images for drawing (etc) active
 2. pick Next

prompt Plot progress information displayed
then Publish to Web (Preview and Post) dialogue box
respond pick Preview

and Internet Explorer with Images of drawings – Fig. 46.5
respond 1. view your images
 2. close the Internet to return to AutoCAD

and select Finish

- 6 You have now create a web page which can be:

a) edited to your requirements
 b) posted to the Internet

- 7 I hope that in this chapter the user has realised that AutoCAD 2002 has uses other than drawing. The web page creation is very useful and relatively simple to create



Figure 46.2 The Publish to Web (Create Web Page) dialogue box.



Figure 46.3 The Publish to Web (Apply Theme) dialogue box.

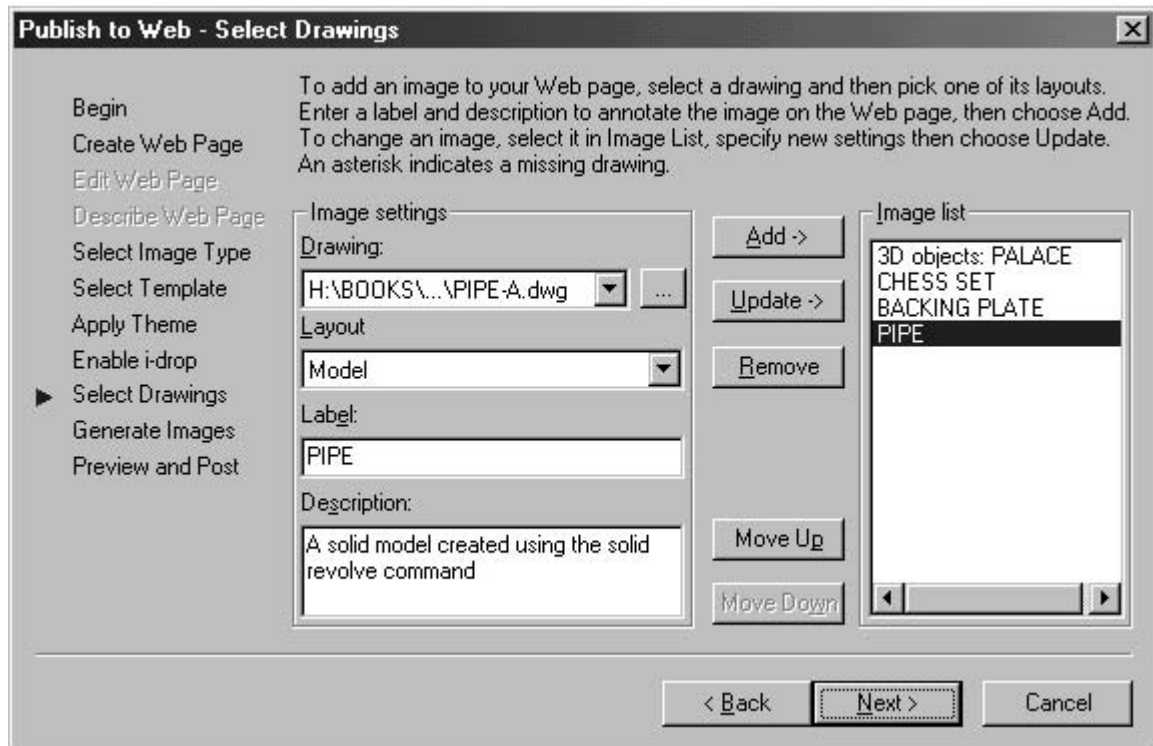


Figure 46.4 The Publish to Web (Select Drawings) dialogue box.

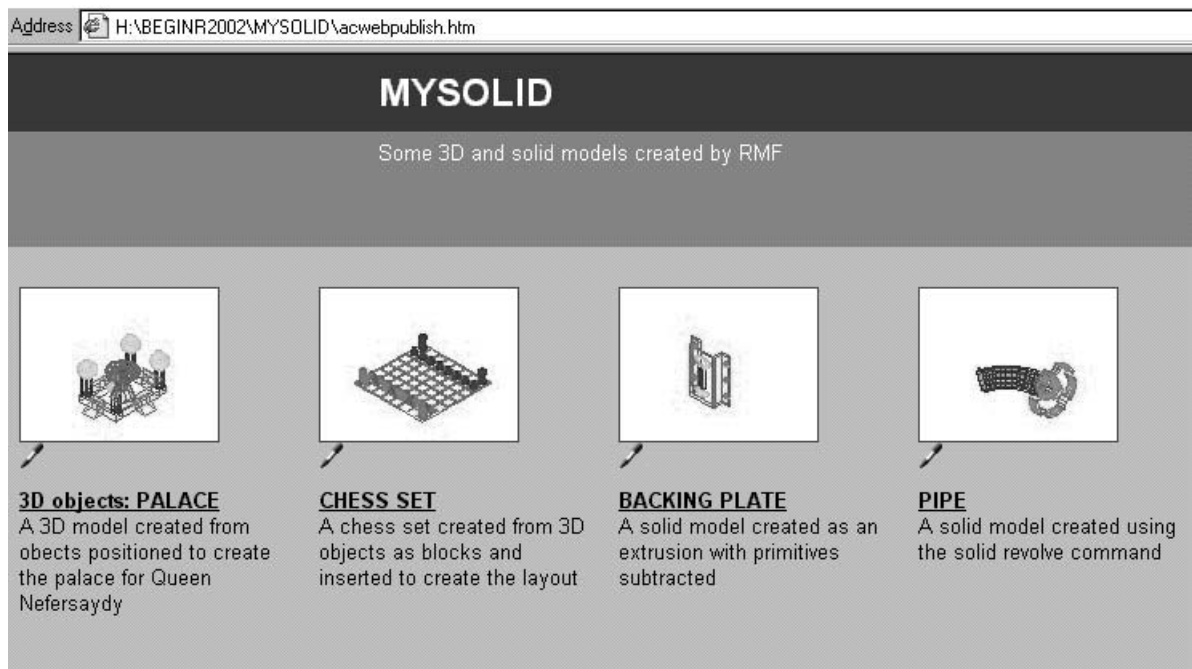


Figure 46.5 Preview of web page.

Students' models

This last chapter will show the reader some solid models which have been created by students. Motherwell College offers the HNC and HND courses in Computer Aided Draughting and Design, and one of the units on this course requires the students to complete a CADD related project. The student must 'think up' this project for themselves, produce a brief, produce a technical report and also deliver an oral presentation to their class group.

Many different and varied projects are produced including menu customisation, hatch pattern customisation, slide shows etc, but most students decide on solid modelling. The 'object' which has to be model has to have sizes and the student must obtain these and then decide on the best way to model the object, i.e. from primitives, regions etc.

To show the reader what can be obtained with solid modelling I have included 12 student projects. These are:

<i>Figure</i>	<i>Title</i>	<i>Student</i>
47.1	Anglepoise Lamp	John-Joseph McCaveny
47.2	Chieftain Tank	Joe Williams
47.3	Formula 1 Racing Car	Tom Doyle
47.4	Crash Helmet (Surface model)	Steven Anderson-Sams
47.5	Micrometer	Pamela Hettrick
47.6	Wood Plane	Michael Monaghan
47.7	Droid	Ryan Farnin
47.8	Suspension Assembly	Alan Thomas
47.9	Battle Car	Craig Matthewman
47.10	Vice	Keith Irwin
47.11	Watch	William Milne
47.12	3D Chess Set	Gary Jamieson

Further student work can be viewed from the Motherwell College website, the address being www.motherwell.co.uk

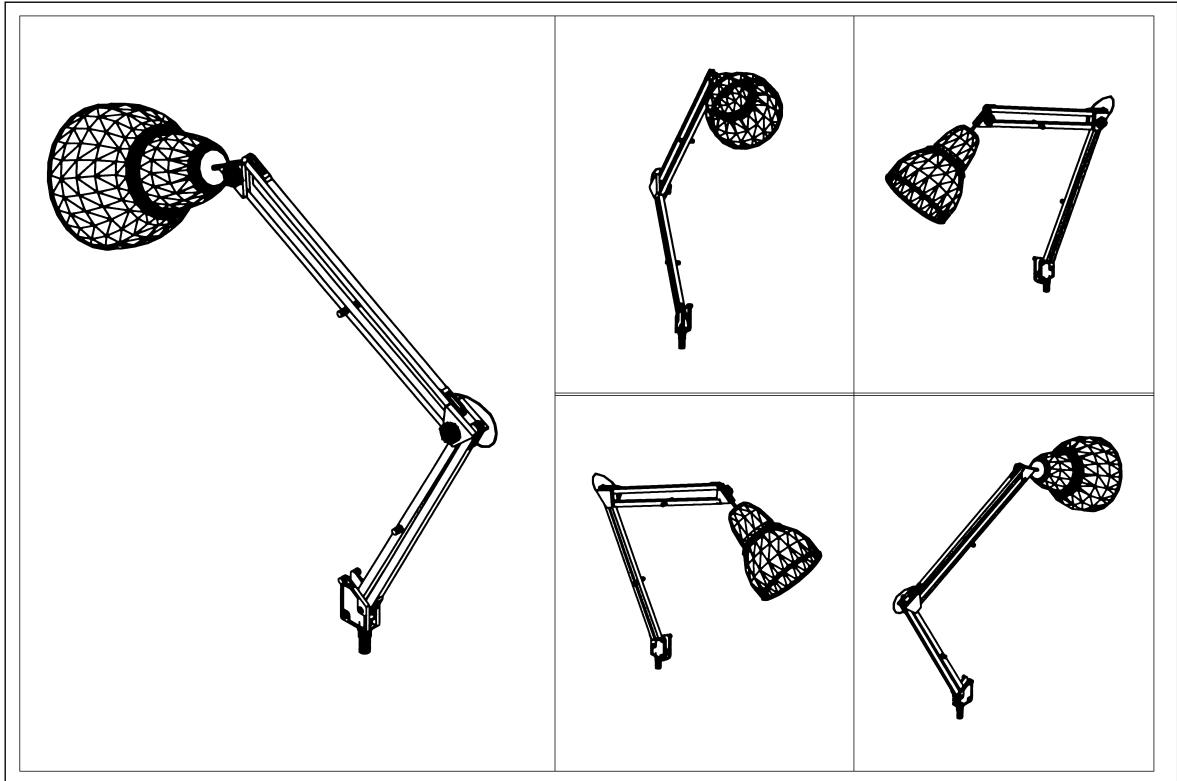


Figure 47.1 JJ McCaveny CAD Project.

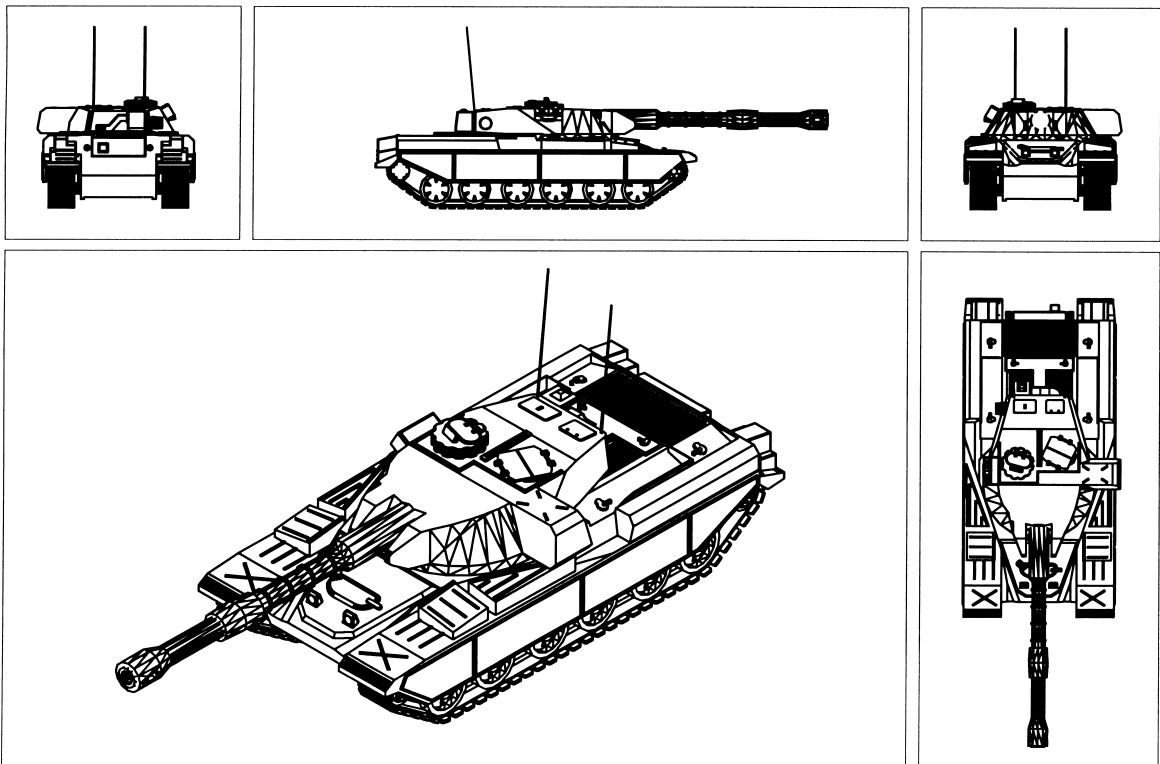


Figure 47.2 Jow Williams HNC (C) CAD Project (Chieftain tank).

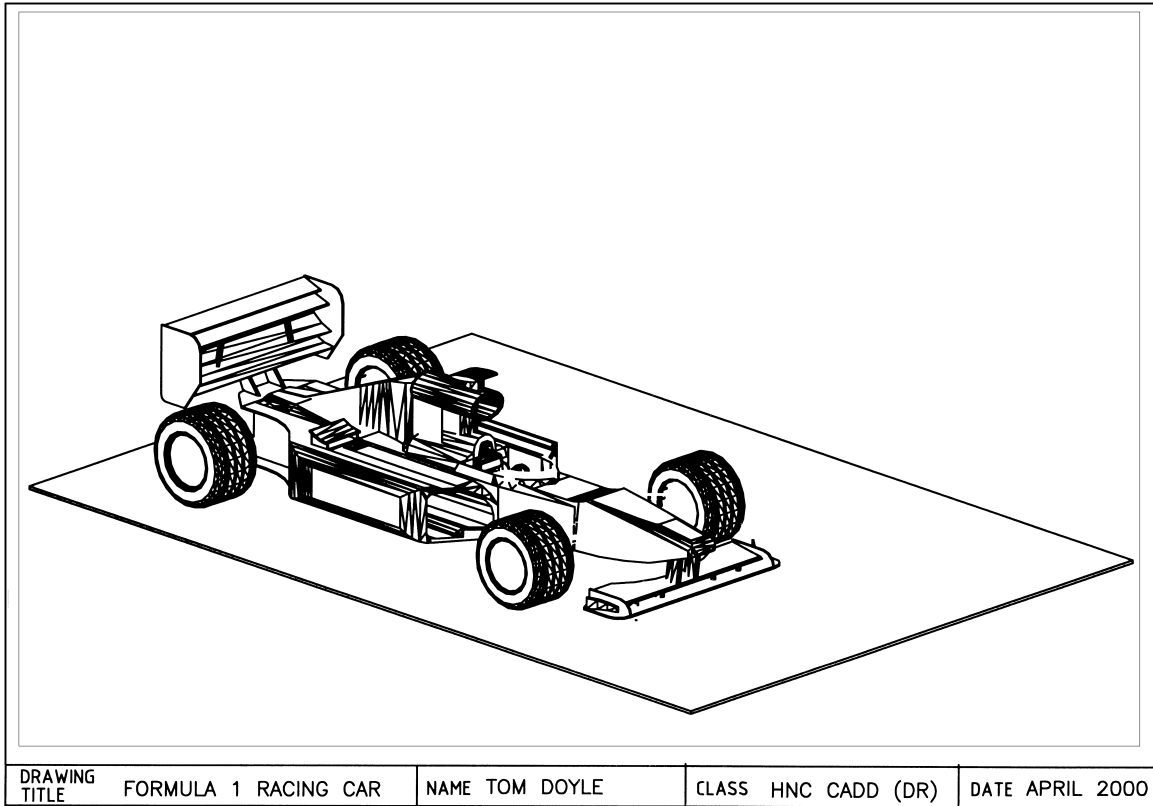


Figure 47.3

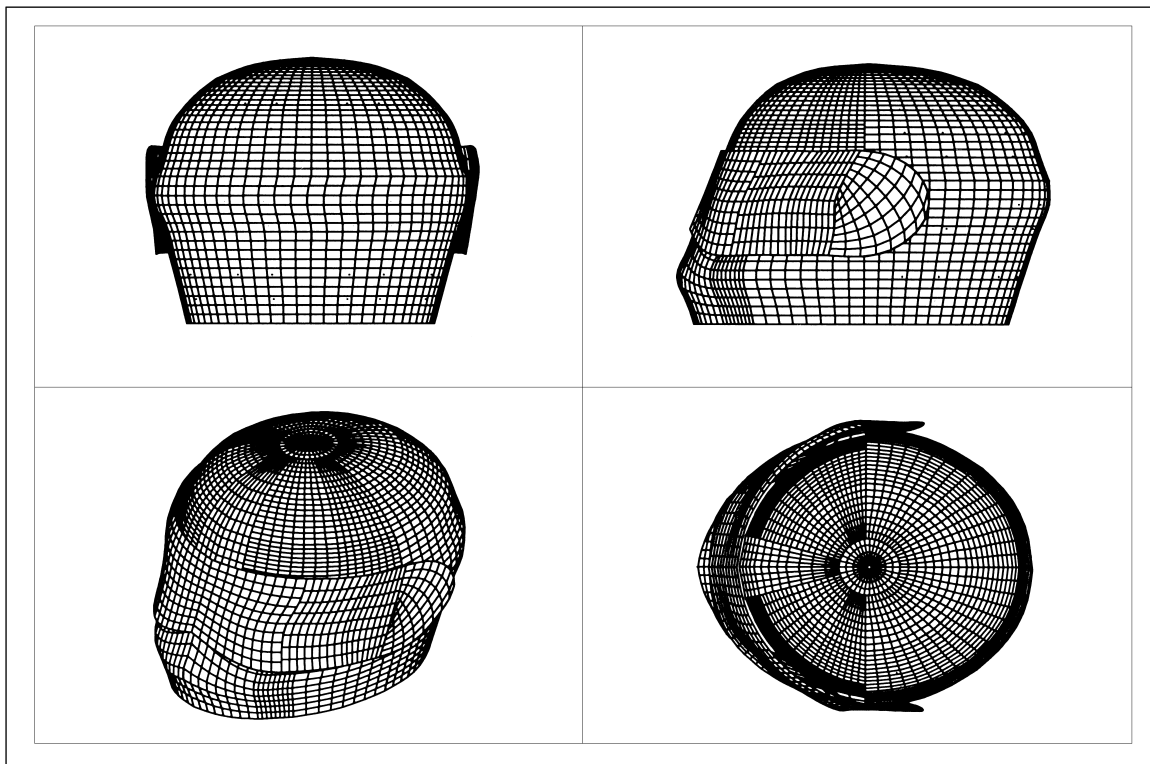


Figure 47.4 Steven Anderson-Sams HNC CADD Project.

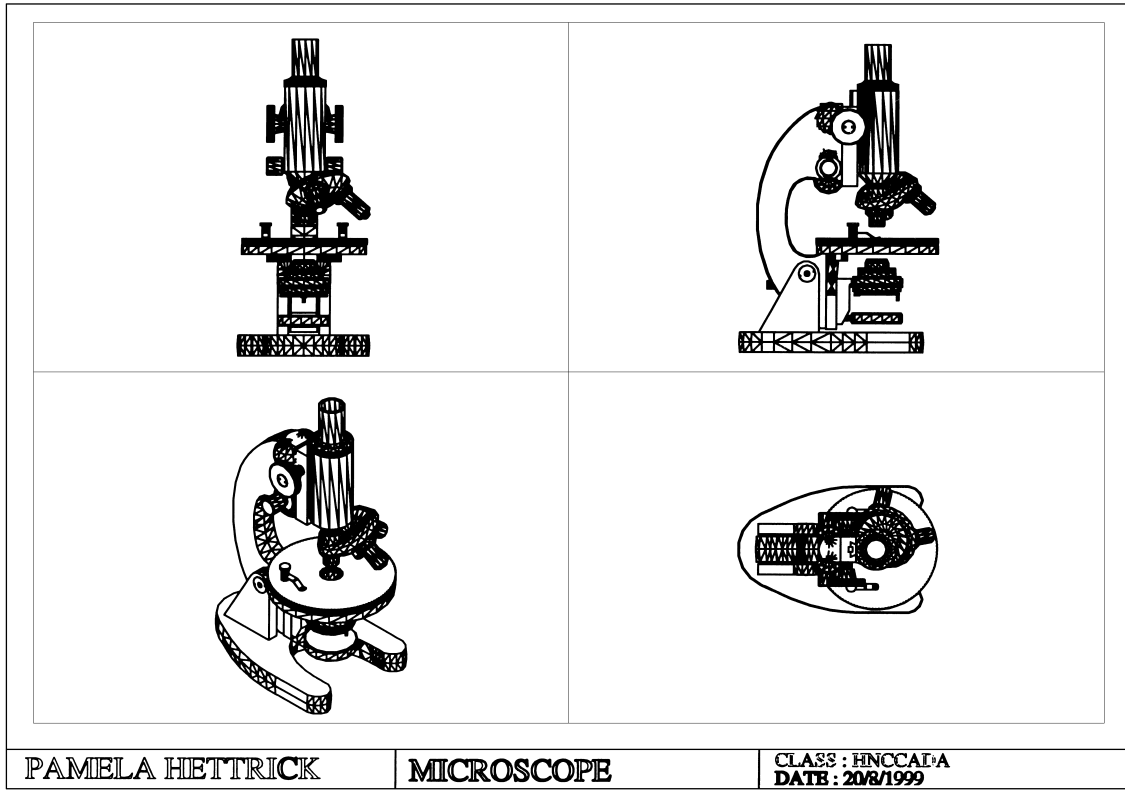


Figure 47.5

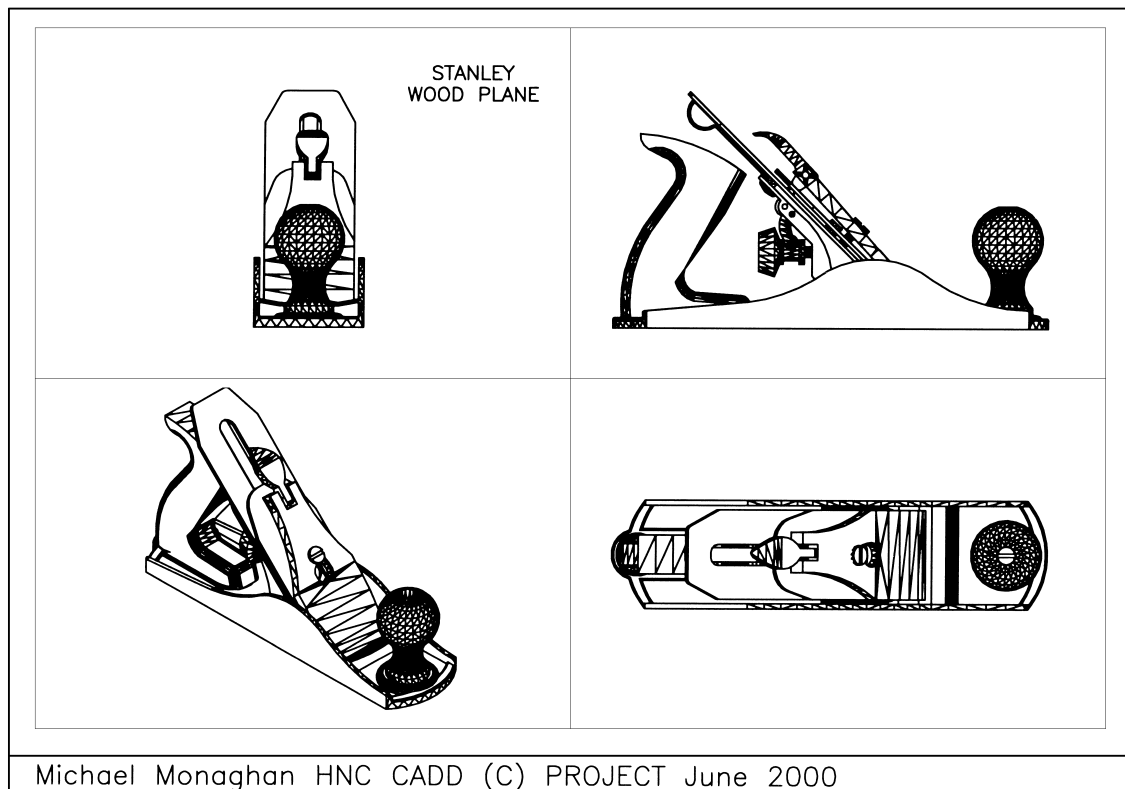


Figure 47.6

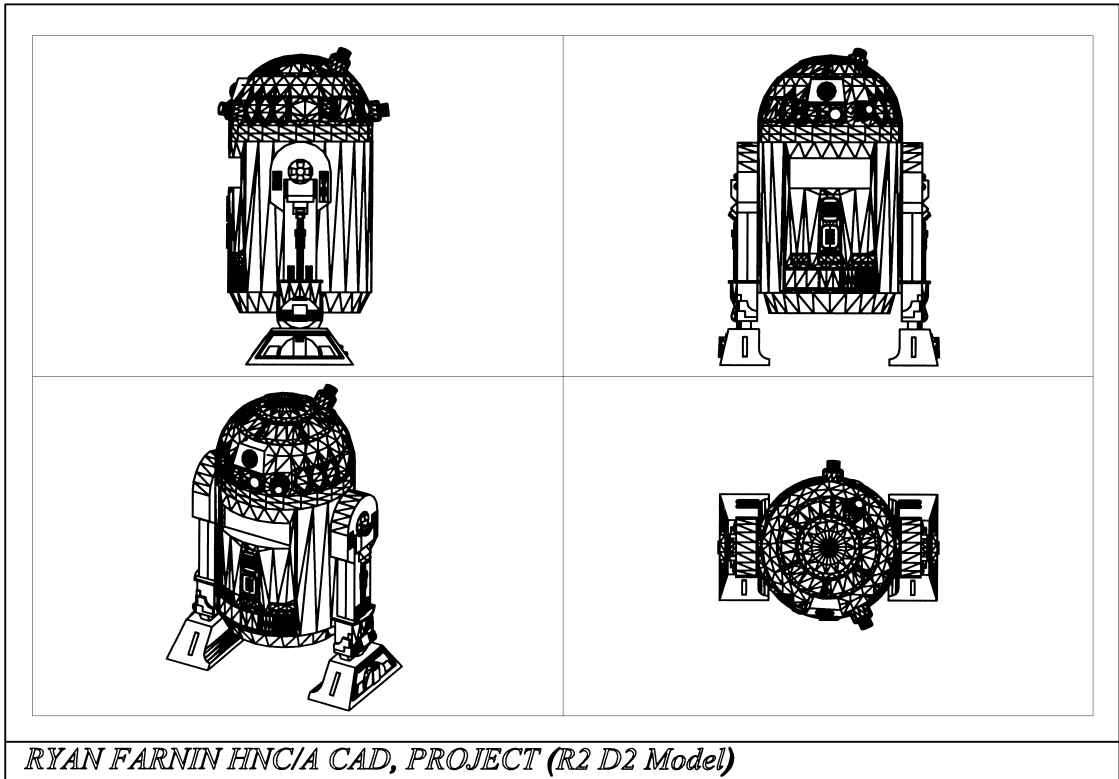


Figure 47.7

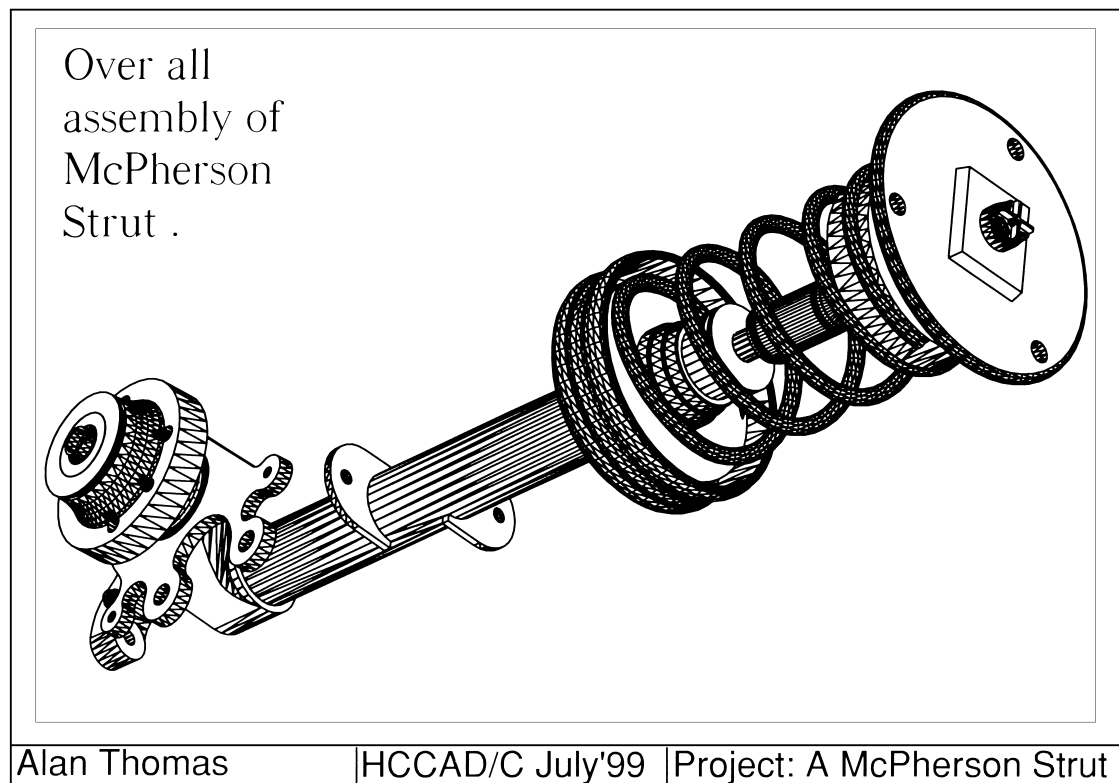


Figure 47.8

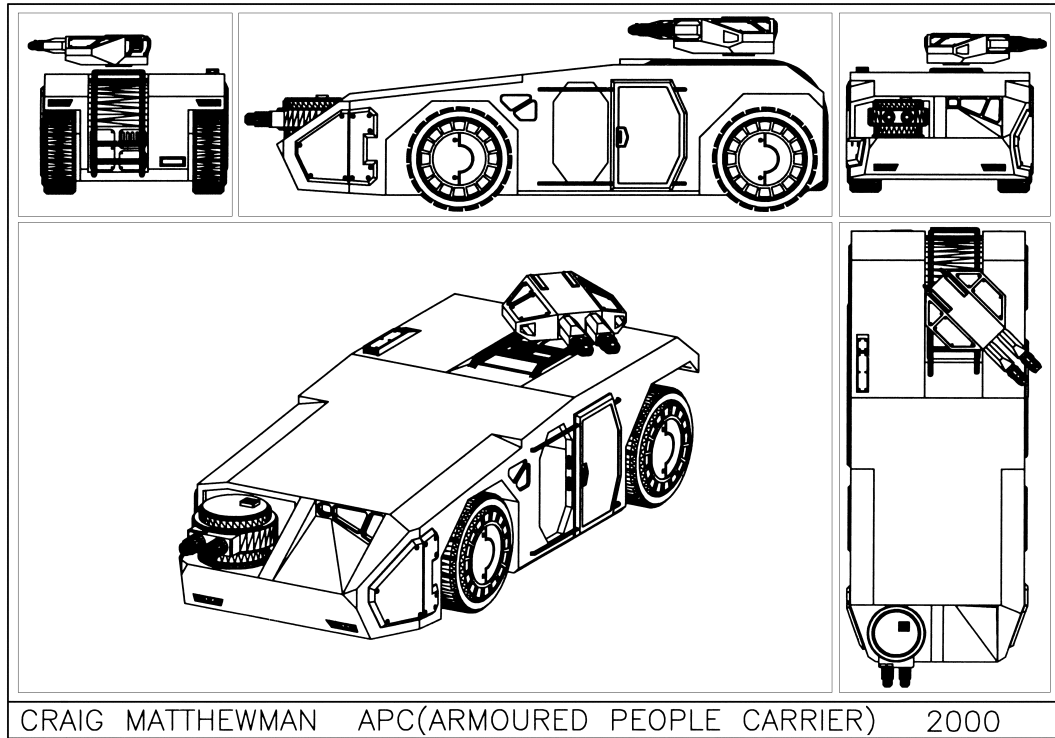


Figure 47.9

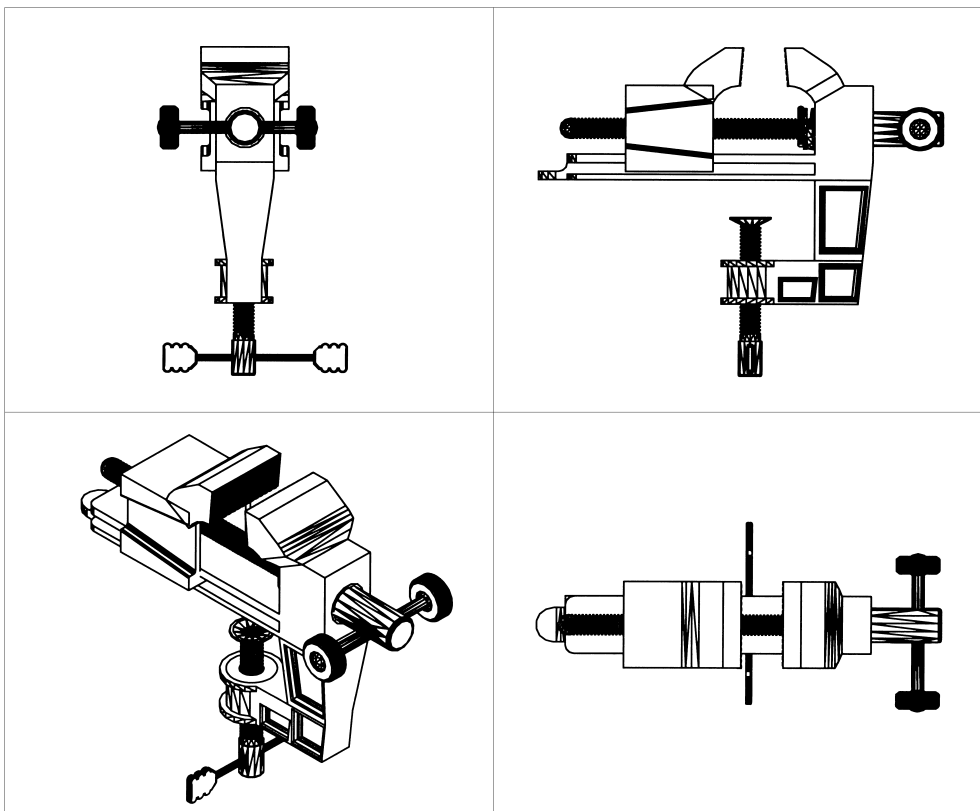


Figure 47.10 A vice by Keith Irwin.

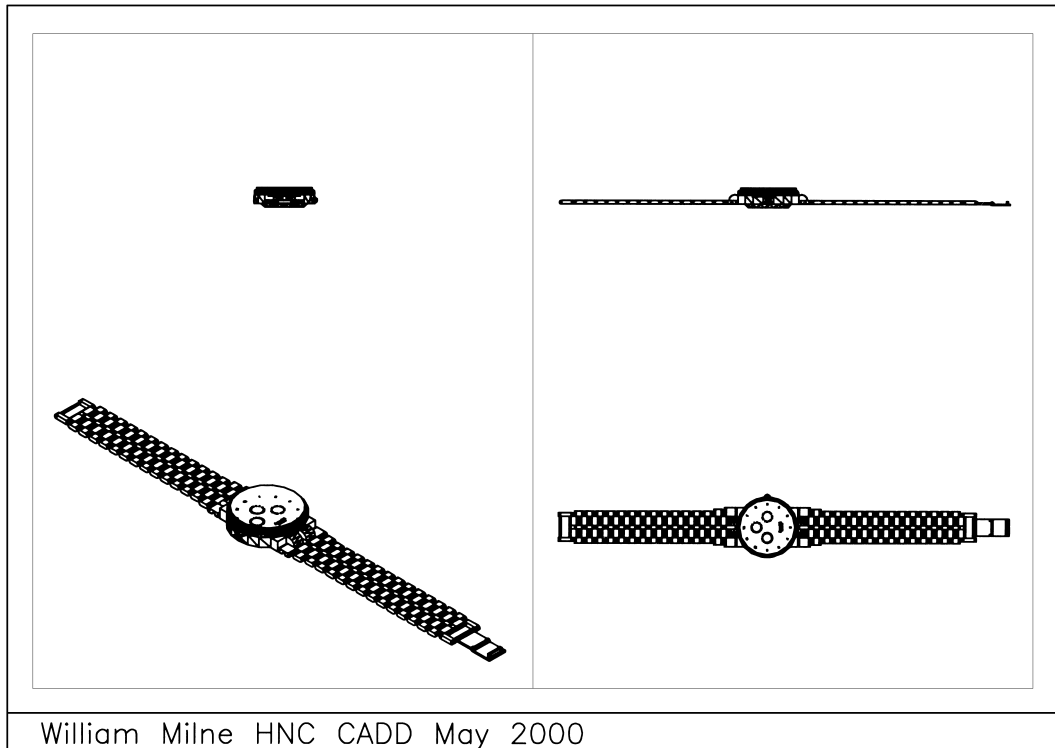


Figure 47.11

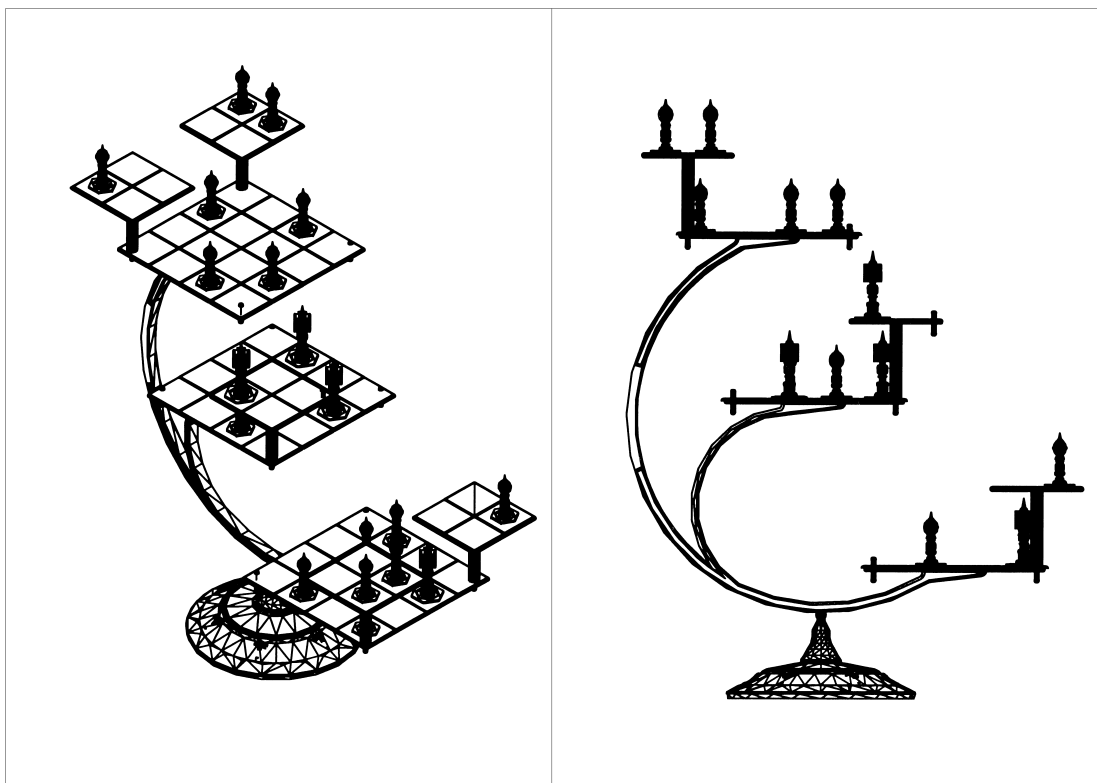
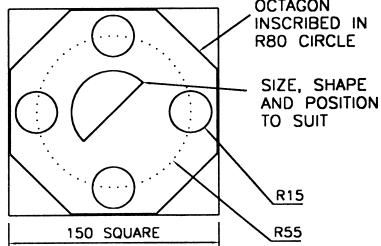
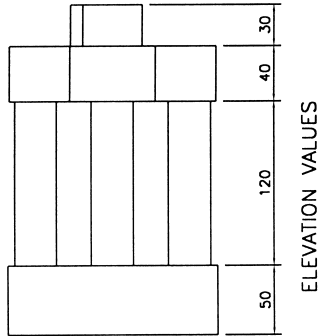


Figure 47.12 Gary Jamieson.

ACTIVITY 1

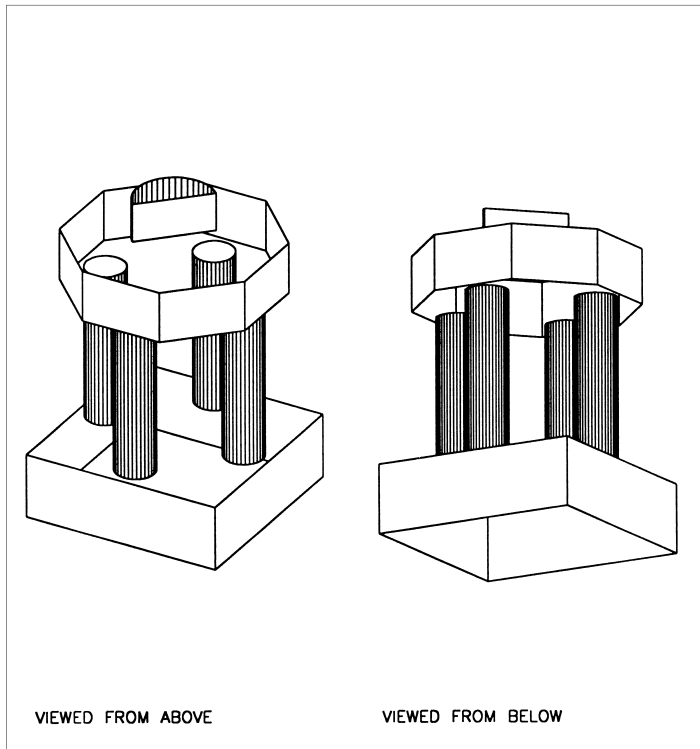
The coloured structure of MACFARAMUS

Using the extruded elevation and thickness method with the data given, create the model as a 'plan' then view in 3D from above and from below.



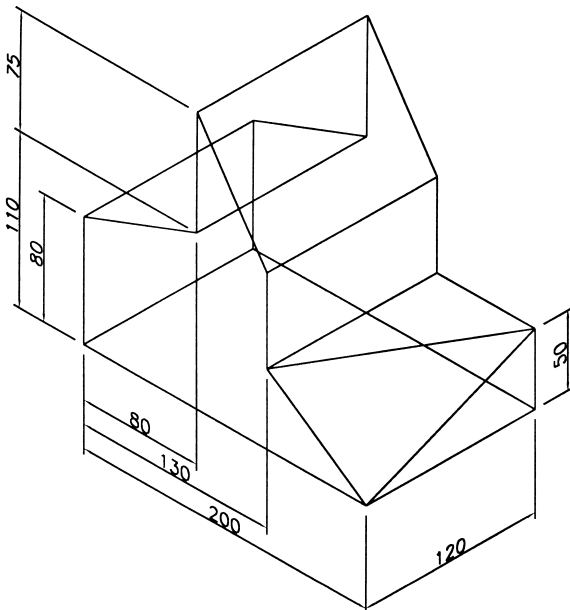
VPOINT 'R' at 300° and 30°

VPOINT 'R' at 300° and -15°



ACTIVITY 2

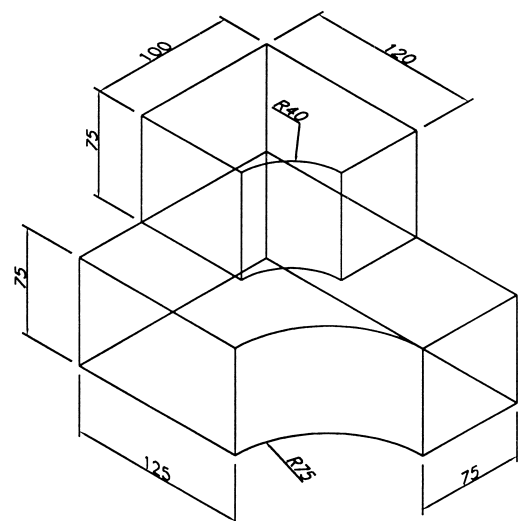
Create the 3D wire-frame model using the sizes given, and save for future recall.



MACFARAMUS's SHAPED BLOCK No.1

ACTIVITY 3

Create the 3D wire-frame model of the given shaped block and save for future recall.



MACFARAMUS's SHAPED BLOCK No.2

ACTIVITY 4
 a) open ACT-2 Wire-frame model
 b) set and save UCS positions
 c) add the given dimensions
 d) save for future recall

MACFARAMUS's shaped block 1 with added dimensions

Suggested UCS names with their positions and orientations

- BASE (1)
- FRONT (2)
- MID (3)
- RIGHT (4)
- SLOPE (5)

Dimensions shown in the model: 185, 110, 80, 20.56°, 33.69°, 35.54°, 90.14°, 30.26°, 80, 130, 200, 120, 25.51°.

ACTIVITY 5
 a) Create the wire-frame model using the sizes given
 b) Set and save a UCS for each plane on the model. Four UCS names and orientations are suggested
 c) Dimension the model
 d) Save for future recall
 e) Assist MACFARAMUS

MACFARAMUS's RECTANGULAR TOPPED PYRAMID

Suggested UCS names and orientations

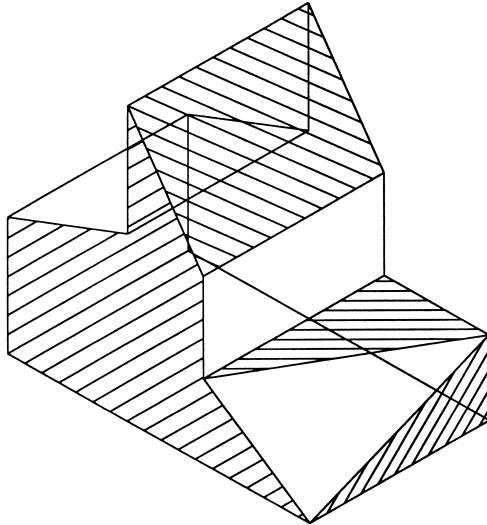
- SLOPE1
- SLOPE2
- SLOPE3
- SLOPE4

Dimensions shown in the model: 80, 50, 70, 180, 200 SQUARE.

ACTIVITY 6

Using the saved UCS settings, add hatching as shown, all hatching being user-defined at 45° and 8 spacing

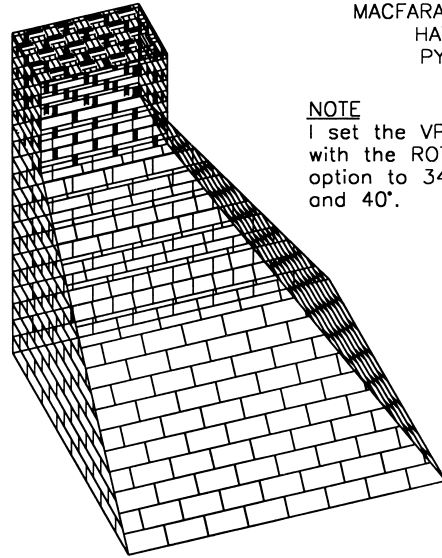
MACFARAMUS's SHAPED BLOCK with hatching added



ACTIVITY 7

Using the slope and vertical saved UCSs, add hatching to the 'planes' using the hatch information given.

MACFARAMUS's HATCHED PYRAMID



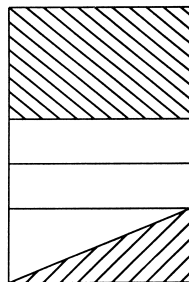
NOTE
I set the VPOINT with the ROTATE option to 340° and 40°.

Surfaces	Pattern	Angle, scale
Four slopes	BRICK	2, 0
Four verticals	BRSTONE	1, 0
One top	EARTH	1.5, 0

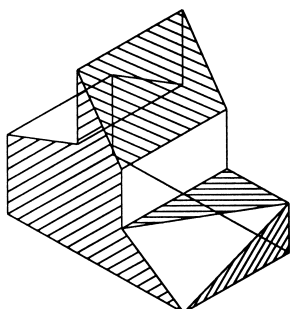
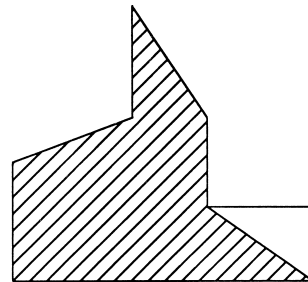
ACTIVITY 8

1. Open the hatched shaped block model
2. Create a four viewport configuration
3. Set the viewpoint to display the model as shown
4. Centre the model in each viewport about the point 100,60,90 at 225 mag (275 in 3D viewport)

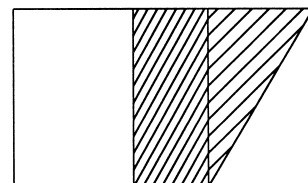
HATCHED SHAPE BLOCK of MACFARAMUS



RIGHT FRONT



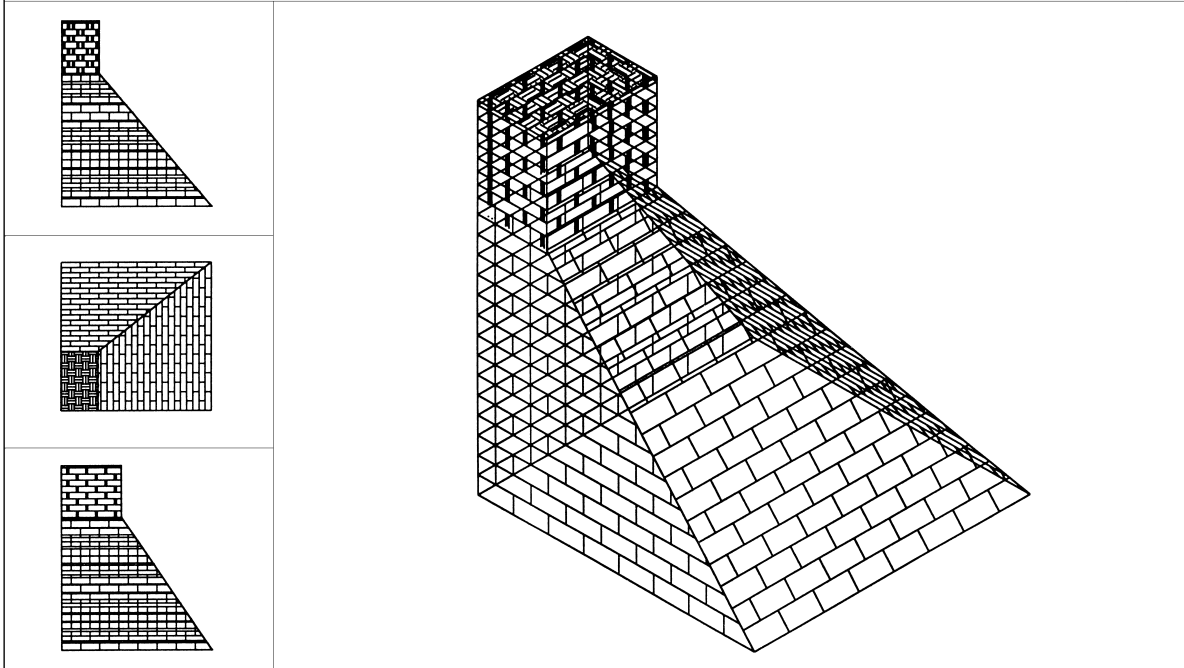
SE ISOMETRIC TOP



ACTIVITY 9

HATCHED PYRAMID of MACFARAMUS

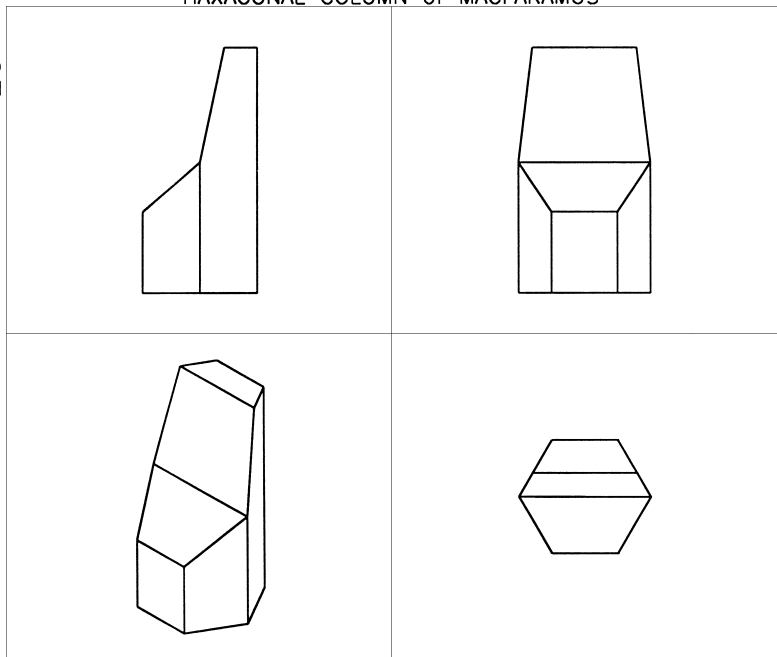
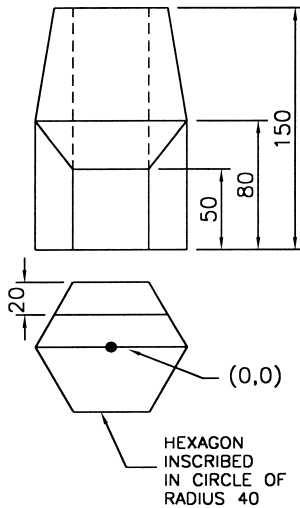
1. Open the hatched pyramid model
2. Create a four viewport configuration to displays the views stated
3. Centre the model in each viewport with either:
 - a) zoom-scale with a scale factor of 1 (3D value?)
 - b) zoom-centre about the point 100,100,125 at 300 mag (3D value?)



ACTIVITY 10

HAXAGONAL COLUMN of MACFARAMUS

1. Create a wire-frame model using the sizes given
2. Use the 3DFACE command to covert the wire-frame model into a surface model
3. Hide/shade etc.



FARCAD

TITLE:

DRAWN BY:

DATE:

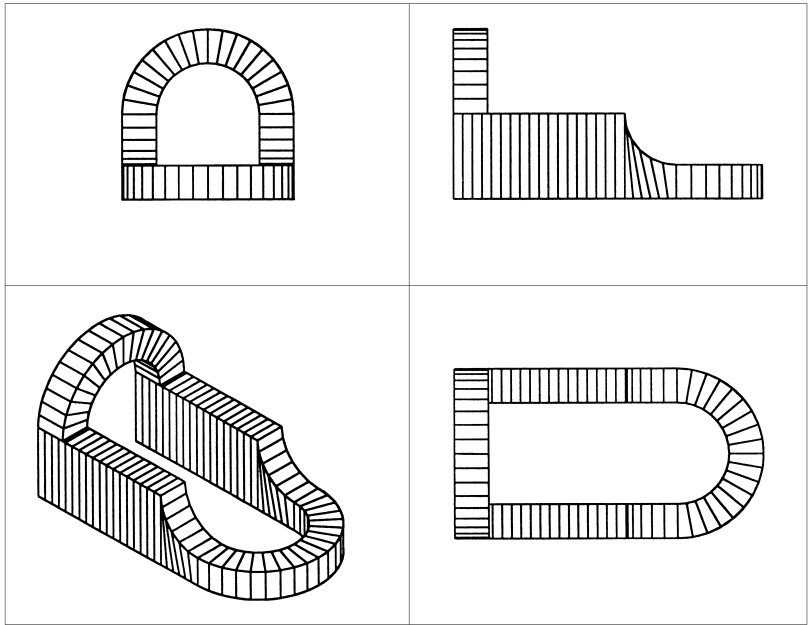
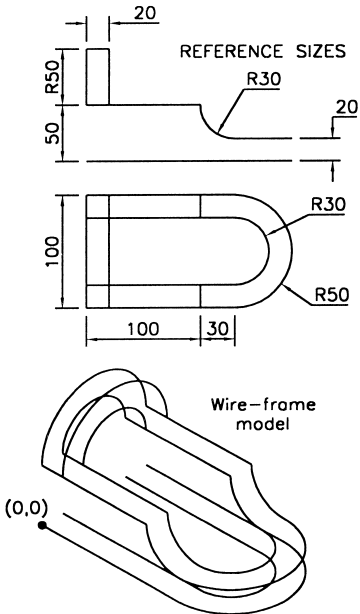
CHK:

REV:

ACTIVITY 11

Using the reference sizes create a 3D wire-frame model then add ruled surfaces using layers correctly.

ORNAMENTAL FLOWERBED
of MACFARAMUS



FARCAD

TITLE:

DRAWN BY:

DATE:

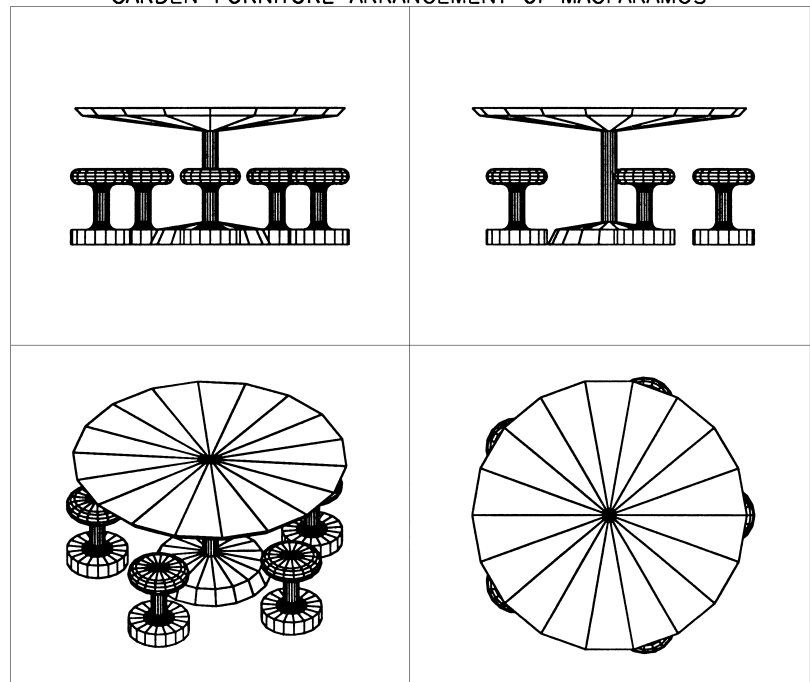
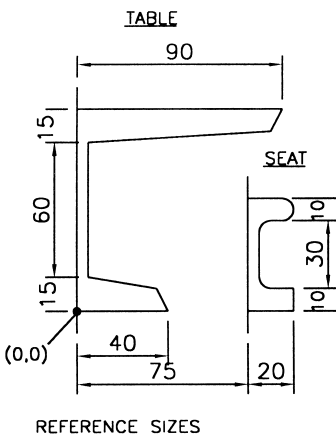
CHK:

REV:

ACTIVITY 12

Using the reference sizes as a guide, design the two path curve profiles and generate revolved surface models of the table and seat. Use discretion for design.

GARDEN FURNITURE ARRANGEMENT of MACFARAMUS



FARCAD

TITLE:

DRAWN BY:

DATE:

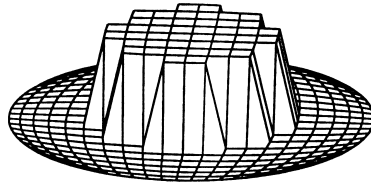
CHK:

REV:

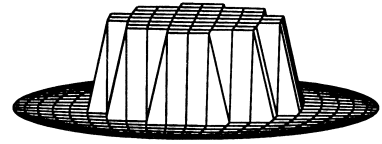
ACTIVITY 13

1. Create four touching arcs with 0,0 as their centres
2. Edge surface the four arcs with SURFTAB1 and SURFTAB2 both set to 20
3. Use Modify-Object-Polyline with Edit Vertex MOVE option, to re-position the named vertices by @0,0,100.
4. Save completed activity as MODR2002\HILL

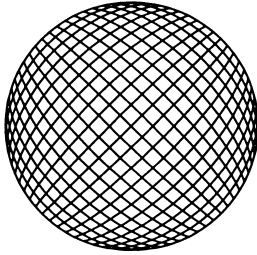
VPOINT-Rotate at 50° and 20°



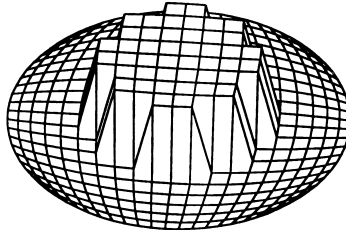
VPOINT-Rotate at 50° and 10°



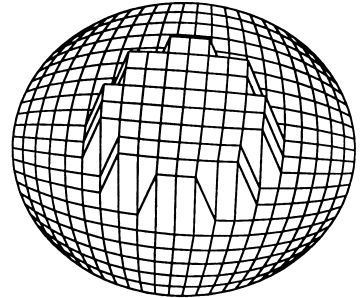
TOP VIEW



VPOINT-Rotate at 50° and 40°



VPOINT-Rotate at 50° and 60°



FARCAD

TITLE:

DRAWN BY:

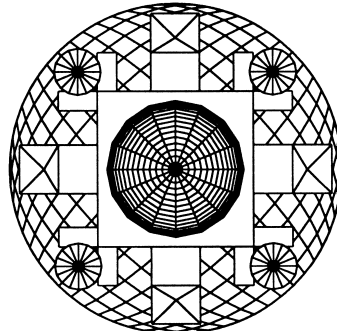
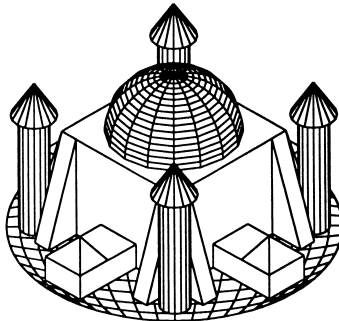
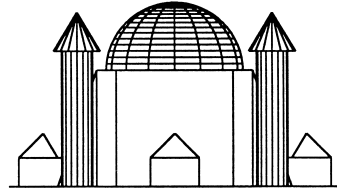
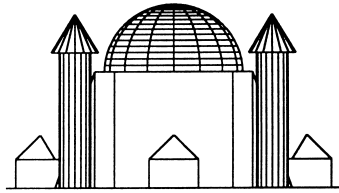
DATE:

CHK:

REV:

ACTIVITY 14: PALACE for queen NEFERSAYDY by MACFARAMUS

Create the palace from 3D objects using your imagination and initiative for the layout. When complete, save as MODR2002\PALACE.



FARCAD

TITLE:

DRAWN BY:

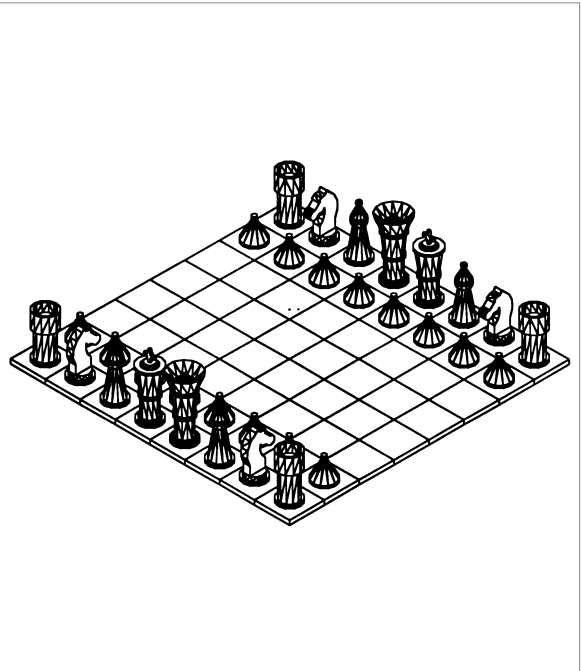
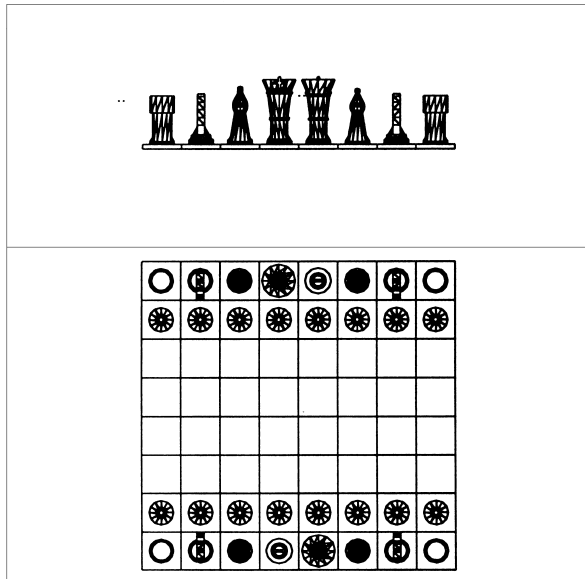
DATE:

CHK:

REV:

ACTIVITY 15

1. Recall the drawing layout MODR2002\CHESSE
2. Design the other chess pieces
3. Save the completed layout for rendering.



FARCAD

TITLE:

DRAWN BY:

DATE:

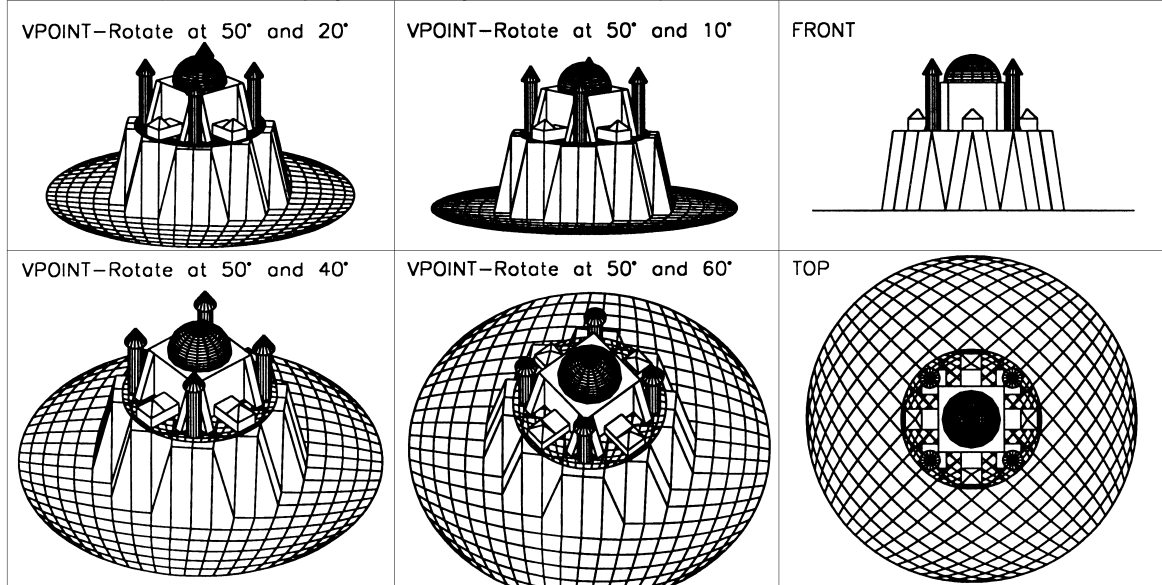
CHK:

REV:

ACTIVITY 16

1. Open the saved edge surface layout MODR2002/HILL
2. Insert the MODR2002\PALACE model, centred on the top of the hill
3. Create viewpoints to display the complete model 'to perfection'

PALACE of Queen NEFERSAYDY
by
MACFARAMUS



FARCAD

TITLE:

DRAWN BY:

DATE:

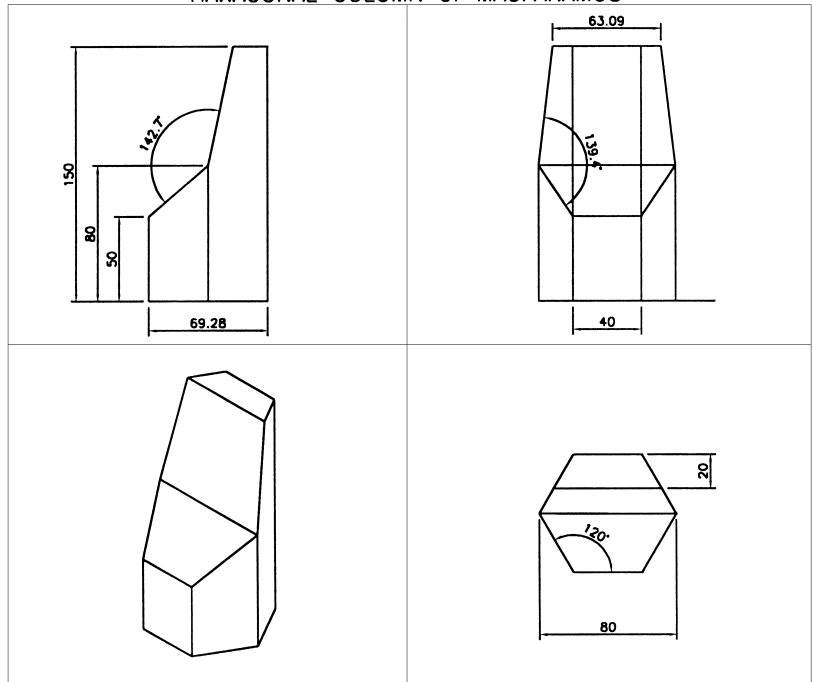
CHK:

REV:

ACTIVITY 17

- 1) Recall activity 10
- 2) Make three new viewport specific layers eg DIMTR etc
- 3) Use the new viewport specific layers to add the dimensions as shown (or your own)
- 4) Remember:
 - a) restore the correct UCS
 - b) make the correct layer current in the appropriate viewport
- 5) Save as required

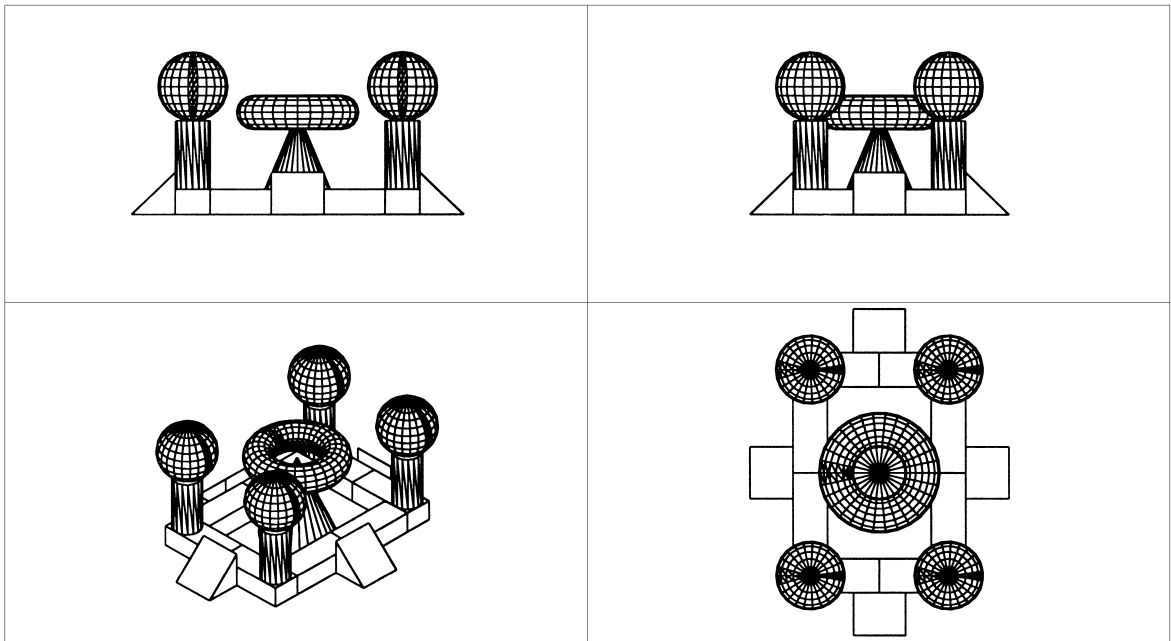
HAXAGONAL COLUMN of MACFARAMUS



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ACTIVITY 18: SOLID PRIMITIVE LAYOUT

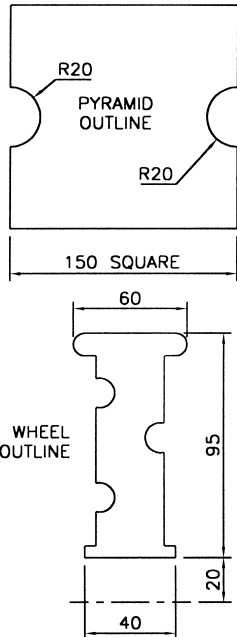
Create a layout of your own design using the six primitives.



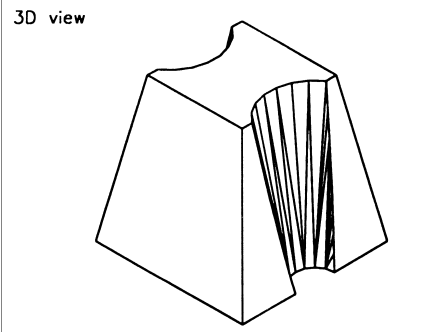
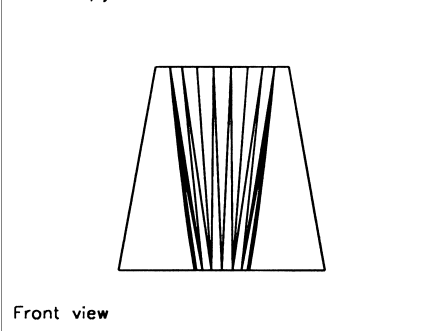
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ACTIVITY 19

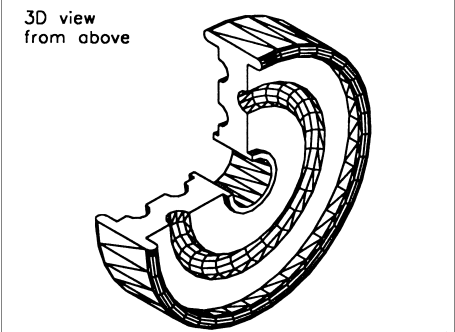
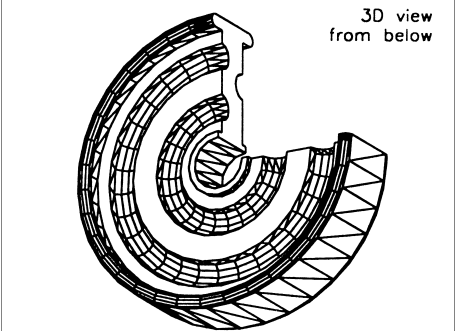
Use the reference sizes to create swept models.



Model pyramid of MACFARAMUS



Partial chariot wheel of MACFARAMUS.

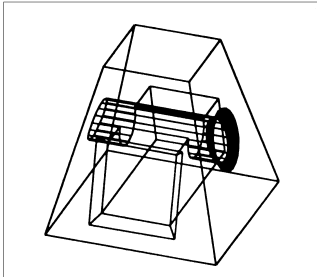


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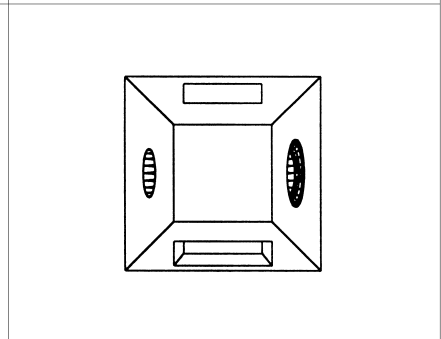
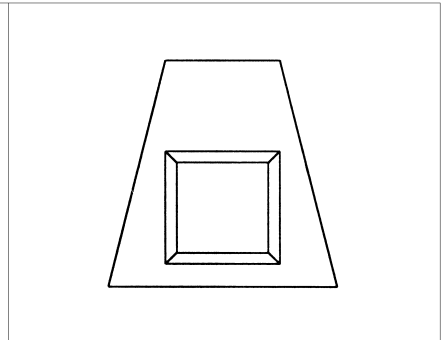
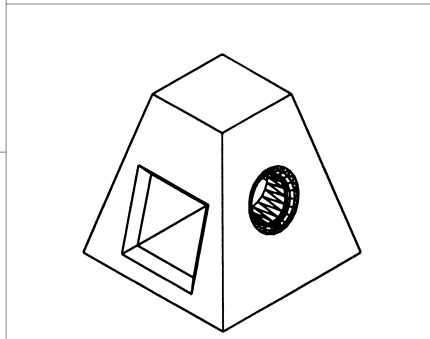
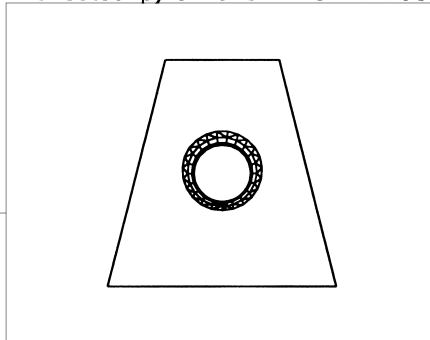
ACTIVITY 20

Create the composite from a cube, box and cylinder. The cube has to be chamfered to give a 'flat top' effect.

Display without hide.



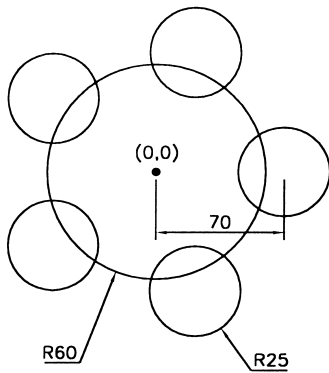
Truncated pyramid of MACFARAMUS



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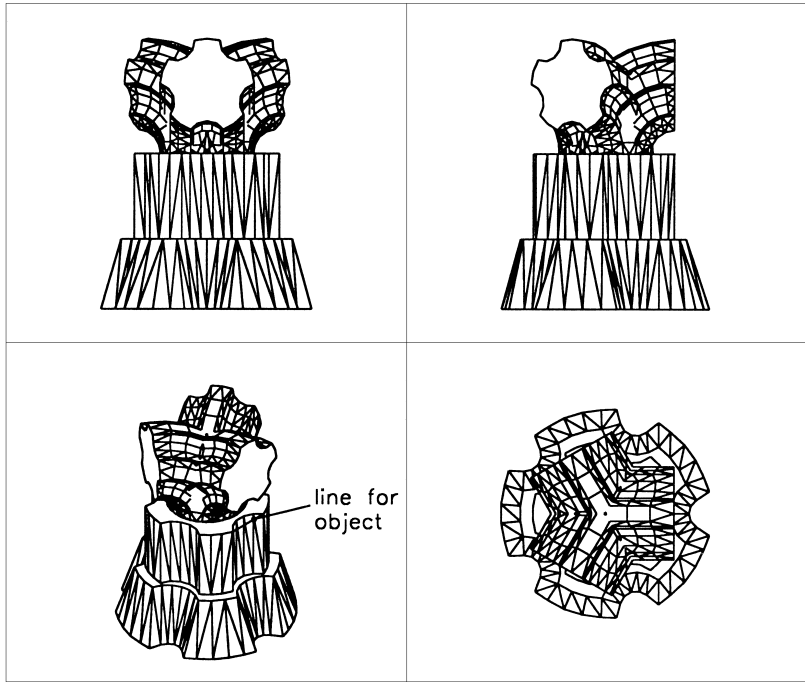
ACTIVITY 21

Create a composite from a region using the information given.



NOTE

1. The original region is copied and scaled by 0.85 and 0.55
2. The smaller regions are positioned on top of each other
3. The extruded information is:
 - a) large: height -50, taper -15°
 - b) middle: height 60, taper 0°
 - c) small: rotated thru' -90°
4. Use discretion for sizes not given.



FARCAD

TITLE:

DRAWN BY:

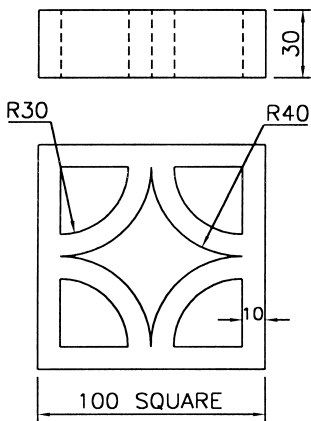
DATE:

CHK:

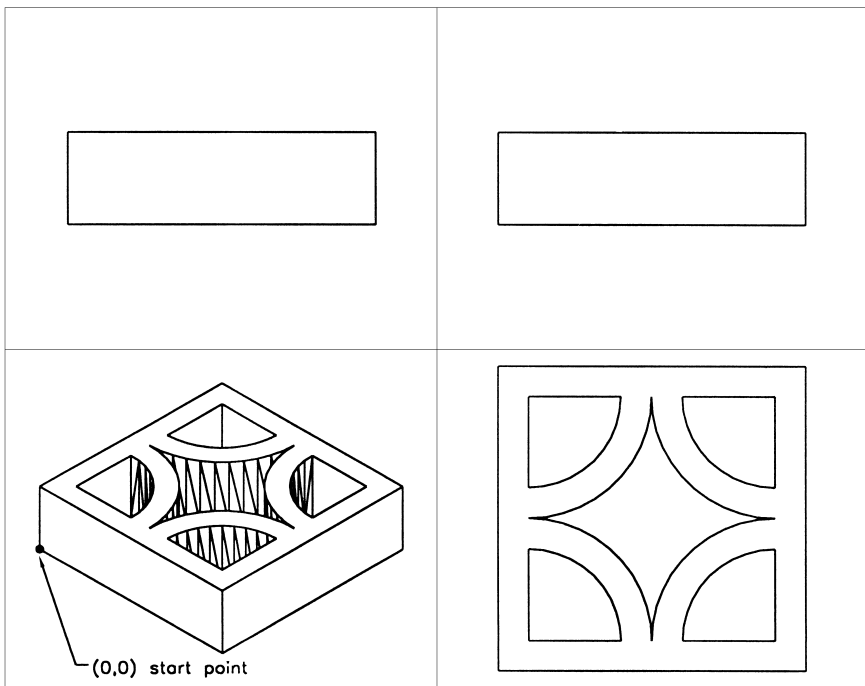
REV:

ACTIVITY 22

Create the model of the garden block using the reference sizes given.



Reference sizes



FARCAD

TITLE:

DRAWN BY:

DATE:

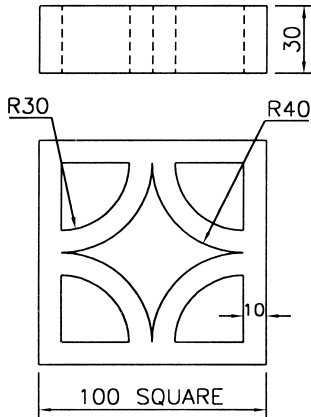
CHK:

REV:

ACTIVITY 23

Using the model from activity 22, set DIMASSOC to 2, extract profiles from the appropriate viewport and add paper space dimensions.

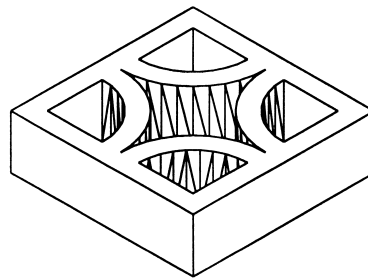
Dimensioned garden block of MACFARAMUS



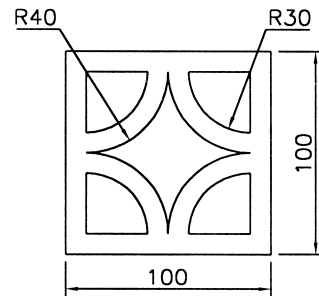
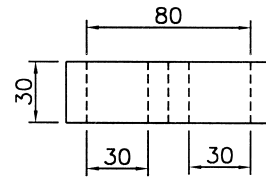
Original sizes



Diagonal section



3D view



FARCAD

TITLE:

DRAWN BY:

DATE:

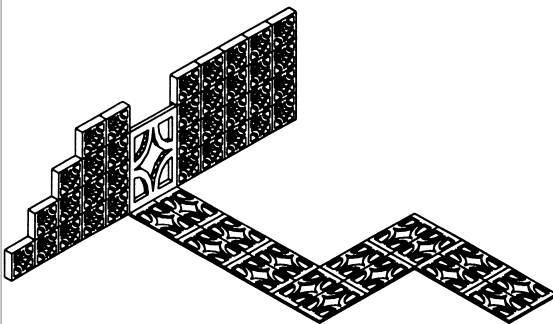
CHK:

REV:

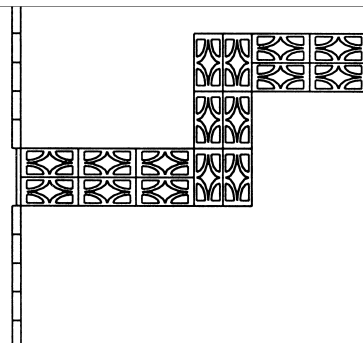
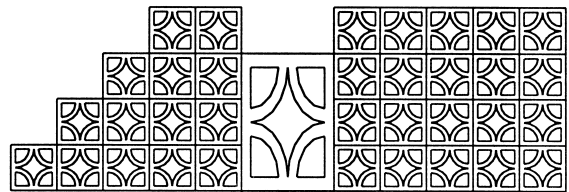
Garden layout of MACFARAMUS using block GARDBL.

ACTIVITY 24

Using the component from Activity 22, make a block, then insert to produce a layout.



NB: The views are not all to the same scale



FARCAD

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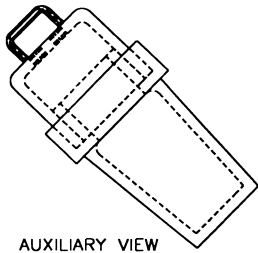
DRAWN BY:

DATE:

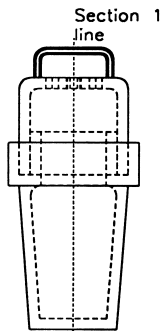
CHK:

REV:

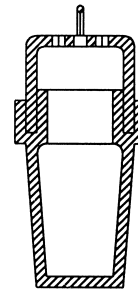
ACTIVITY 25: DISPENSER OF MACFARAMUS
 Using the information in Fig.44.3 and your discretion, create the views with the SETUP commands.



AUXILIARY VIEW

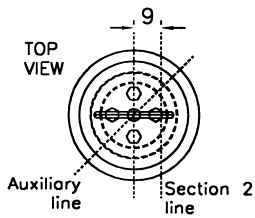
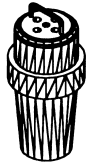


FRONT VIEW

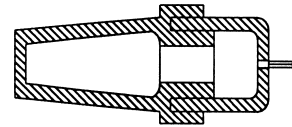


SECTION 1 VIEW

3D VIEW



TOP VIEW



SECTION 2 VIEW

FARCAD

TITLE:

DRAWN BY:

DATE:

CHK:

REV:

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