



## **LT3D Training Guide**

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# LT3D

## Training Guide

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*by <Simon Bruce>*

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*Last not least, we want to thank EC Software who wrote this great help tool called HELP & MANUAL which printed this document.*

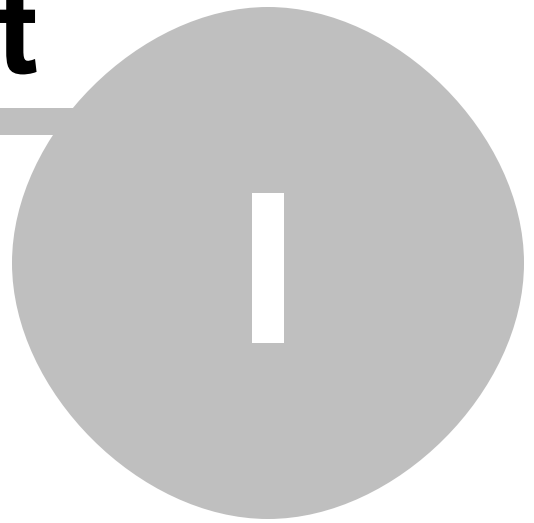
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# Foreword

This is just another title page  
placed between table of contents  
and topics

**Part**



# 1 Introduction

## 1.1 Welcome topic

Welcome to the LT3D Training Guide.

This training guide provides step by step examples and exercises to assist with learning LT3D.

## 1.2 Using This Documentation

The **LT3D** Training Guide is freely available on the **drcauto** web page at <http://www.drcauto.com/training/index.html>, and can be down loaded to be used as a self-instruction manual. On successful completion of all the exercises in the manual, the user will have achieved a suitable level of knowledge for creating 3D models in AutoCAD LT.

**drcauto** does not guarantee any level of competence when the training guide is used as a self-instruction manual. Completion of the training course at a certified **drcauto** Training Centre is recommended.

**drcauto** reserves the right to alter this manual at any time. This manual may not always reflect the exact procedures, methods or wording used in **LT3D**. **LT3D** is subject to updates and modifications without notification.

## 1.3 Typographical Conventions

### Command Line Prompts

In this training guide, the **LT3D** command line prompts appear in a special typeface, such as:

`Command:`

The command line prompts in **LT3D** will usually require a response from the user. By matching the prompts provided in this training guide, with the ones on the screen, you will be assured of performing the correct operation.

### User Input

You are required to input information on the command line or perform a specific operation as instructed by the command line. A special typeface has been used in this training guide to indicate the user's response. For Example:

### *Select the cutting edges*

## Exercises for the User

As an exercise, you will be required to experiment with some functions on your own. In these cases, a heading as follows is used:

### **Exercise:**

## Metric and Imperial

This training manual does not distinguish between imperial and metric measurements. All measurements are entered as non-specific units.

## Keys and Characters

The Enter (or Return) button is shown as: ↵

The Control Key is shown as: **Ctrl**

The Shift button is shown as: **Shift**

The Escape button is shown as: **Esc**

If one key is to be held down while you press another key, for example holding down the **Ctrl** key and pressing the **F2** key, it is shown as: **Ctrl+F2**

## 1.4 Assumptions About the User

The **LT3D** Training Guide is designed to teach basic skills needed to develop 3D models. This course does not teach drafting techniques. It is assumed that the user has some training or experience in drafting and design techniques, specific to their discipline.

This training course does not attempt to teach common 2D drawing procedures in AutoCAD, nor does it teach the use of a computer or the computer operating system. It is assumed that the user has a basic understanding of AutoCAD and understands the basic principles of 2D geometry before embarking on this course.

A user who is unfamiliar with AutoCAD, or computers and computer operating systems, will need to learn about them before attempting to learn **LT3D**.

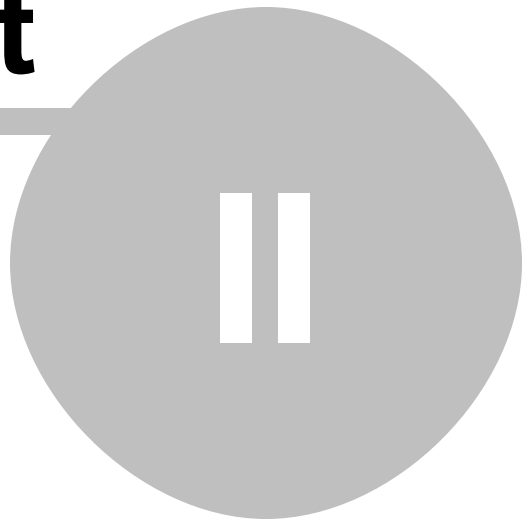
## 1.5 The User Interface

The User Interface is similar in appearance to the standard AutoCAD LT Interface. When **LT3D** is first initiated, two new options are added to the AutoCAD LT menu bar as well as a number of additional tool bars.

The LT3D menu can be loaded and unloaded using the AutoCAD LT `MENULOAD` command. The menu may also be modified using the standard procedures within AutoCAD LT.

Refer to the LT3D help for an explanation of the commands within the LT3D pull downs menus and tool bars.

**Part**



## 2 Tutorial 1

### 2.1 Overview

When developing a 3D model, it may not be necessary to always create solid objects. It can sometimes be simpler to use 3D faces or even alter the thickness and elevation of 2D objects. In this Tutorial we'll look at a number of options that can be used to expedite the drawing of 3D models, without using any 3D solid objects.

### 2.2 Exercise 1

#### Setting Thickness And Elevation

The **THICKNESS** and **ELEVATION** system variables in AutoCAD LT® can be used very effectively to create some 3-dimensional models. The AutoCAD LT help describes **THICKNESS** as "a property of objects that simulates surfaces", ie, the objects take on a specified value along the Z-axis to give the appearance of 3-dimensional entities.

Changing the **ELEVATION** system variable specifies the position above or below the XY plane at which new entities are drawn. For example, if the **ELEVATION** is set to **5**, all new objects are drawn five units above the XY plane.

Changing the **ELEVATION** or **THICKNESS** system variable settings does not alter any existing objects, however the AutoCAD LT Properties dialog box can be used to change either of these settings, for selected objects, at any time.

The **THICKNESS** variable will only alter the appearance of lines, polylines, spline-fit polylines, circles, rectangles, polygons, boundaries, and donuts.

#### 2.2.1 Changing Thickness and Elevation

Changing the Thickness and Elevation of objects to give a 3 dimensional appearance.

Before we start drawing, we need to turn the drcauto keyboard on. This will provide us with a number of shortcut keys strokes that will speed up the process.

SlingShot menu > Utilities > drcauto Keys On

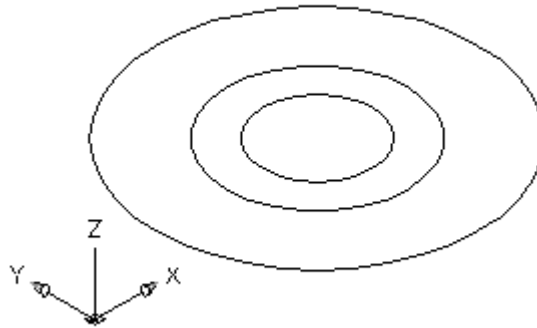
**Step 1:** Draw 3 concentric circles with a radius of **3**, **5** and **9** units, as shown in **Figure 1.1.1**.

**Step 2:** Change to a South-West isometric view.

Command: **V+Delete**

Press **V** followed by the **Delete** key to change the view.

**Note:** Refer to the LT3D Help for further information on drcauto keyboard and extended viewing functionality.



**Figure 1.1.1**

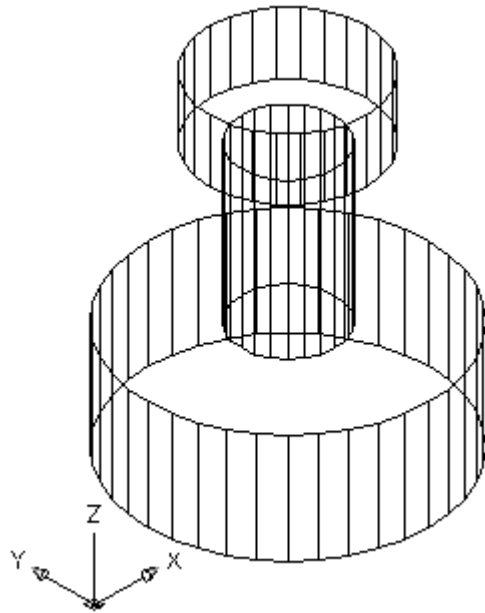
**Step 3:** Using the AutoCAD LT Properties dialog box, select and modify the following properties for each of the circles. Remember to use the **Esc** key to de-select each circle before editing the next.

**Circle with 9 unit radius**  
*Change Thickness to 7.*

**Circle with 3 unit radius**  
*Change Thickness to 10.*  
*Change Center Z to 7.*

**Circle with 5 unit radius**  
*Change Thickness to 4.*  
*Change Center Z to 17.*

**Note:** Refer to the AutoCAD LT help for further information on the **Properties** dialog box.



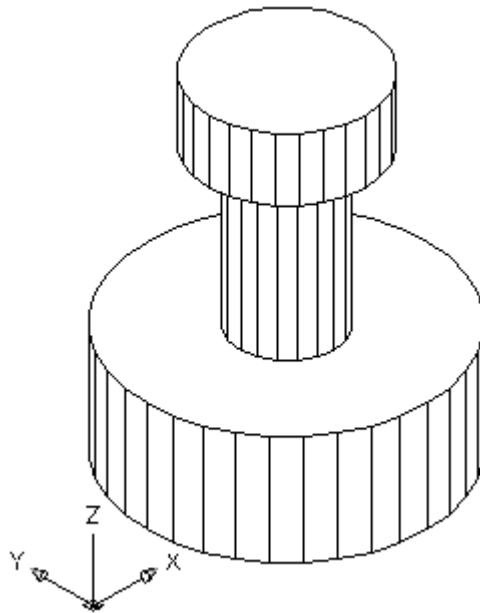
**Figure 1.1.2**

**Step 4:** Invoke a hidden line view of the objects.

Command: **hide** ↵

Removes hidden lines from the current view.

**Note:** The objects can be restored to the original 2D wire diagram by changing the current view or regenerating the drawing. Refer to the AutoCAD LT help for further information on the **HIDE** command.

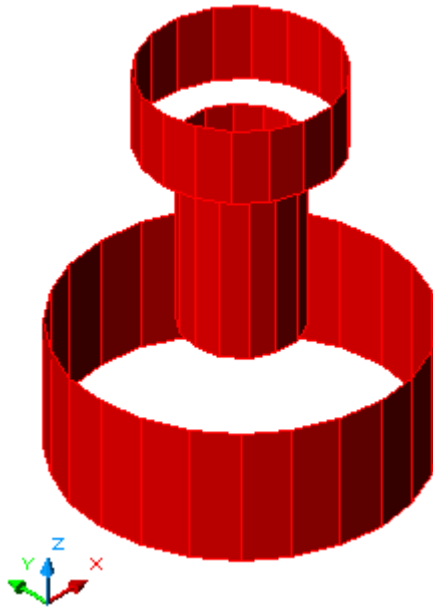


**Figure 1.1.3**

**Step 5:** Apply a shaded finish to the objects. Ensure you use a suitable colour as white on black, or black on white does not display very well.

Command: **shade** ↵

Object is shaded Flat with Edges.

**Figure 1.1.4**

**Step 6:** Change the **SHADEMODE**.

Command: **shademode** ↵

Invokes the **shademode** command.

Enter option [2D wireframe/3D wireframe/Hidden/Flat/Gouraud/Flat+edges/gouraud+edges]

<Flat+Edges>: **G** ↵

Invokes the **Gouraud** shade option.

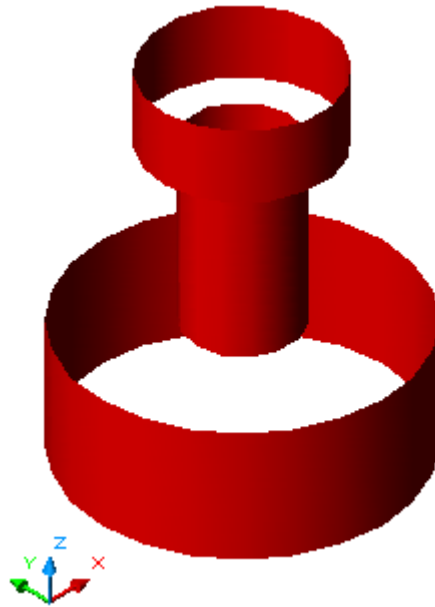


Figure 1.1.5

**Note:** Do not erase these objects, we will utilise them again in Exercise 2.

## 2.2.2 Preset Thickness and Elevation

### Creating entities with preset Thickness and Elevation.

**Step 1:** Restore the drawing to a 2-dimensional wire frame.

Command: **SHADEMODE** ↵

Invokes the **Shademode** command.

Enter option [2D wireframe/3D

wireframe/Hidden/Flat/ouraud/flat+edges/gouraud+edges]

Removes the shaded colour and restores the image to a 2D wire frame.

<2D wireframe>: **2** ↵

**Step 2:** Return to a plan view.

Command: **V+End**

Press **V** followed by the **End** key to restore the plan view.

**Step 3:** Set the **THICKNESS** system variable to **4** units.

Command: **THICKNESS** ↵

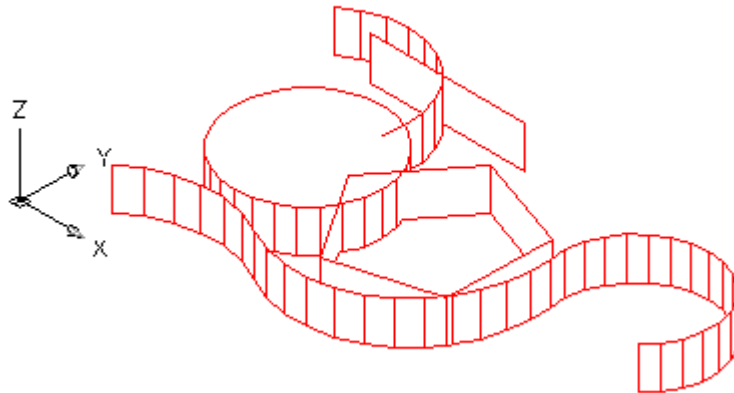
Invokes the **Thickness** command.

Enter new value for THICKNESS <0.0000>: 4 ↵

Sets the **Thickness** system variable to 4 units.

**Step 4:** Draw a circle with a radius of 7 units, an polygon of similar size and a handful of other elements, including a segmented polyline, as shown in **Figure 1.1.6**.

**Note:** Neither the **Rectangle** nor the **Ellipse** will take on the **Thickness** setting. These objects need to be edited with the **Properties** command.



**Figure 1.1.6**

**Step 5:** Set the **ELEVATION** system variable to 4 units.

Command: **ELEVATION** ↵

Invokes the **Elevation** command.

Enter new value for ELEVATION <0.0000>: 4 ↵

Sets the **Elevation** system variable to 4 units.

**Step 6:** Draw a few more objects on the screen then view from a front elevation to distinguish the different position of objects in relation to the XY plane.

Command: **V+Down Arrow**

Views the drawing in front elevation.

**Exercise:** Use the fillet command on the lines/polylines and note that the command disregards objects that are not coplanar (in the same plane).

## 2.3 Exercise 2

In Figure 1.1.4 of the previous exercise, when the SHADE command had been applied, it is apparent that each of our cylinders has no top or bottom, they are hollow tubes. We can use the 2D Solid or Solid Hatch to overcome these problems.

### 2.3.1 Solid Hatch

Using the Solid Hatch command to create surfaces.

**Step 1:** Pan the objects created in Exercise 1 (Figure 1.1.4) into the centre of your screen. Ensure Shademode is set to 2D Wireframe.

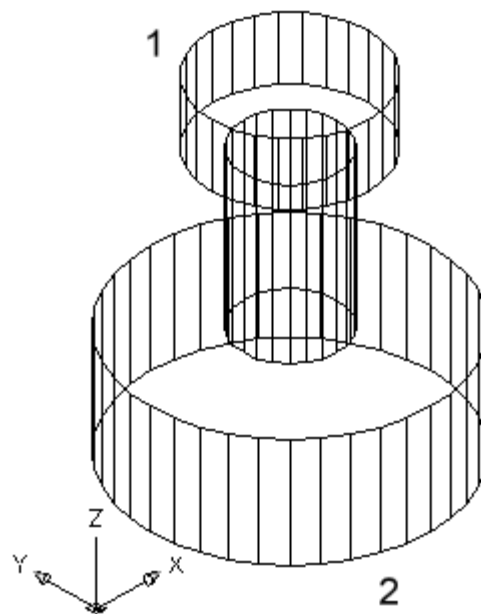
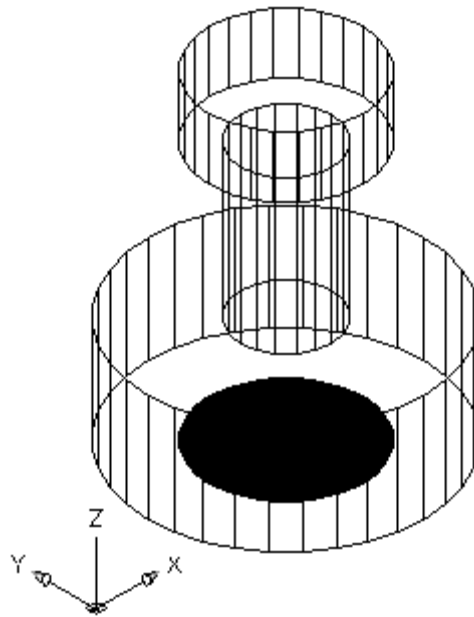


Figure 1.2.1

**Step 2:** Place a solid hatch into the thickened circle 1, at the top of the object in Figure 1.2.1.

*Invoke the BHATCH command.  
Select Solid as the hatch pattern.  
Use the Select Objects button and select circle 1.  
Apply the hatch pattern.*



**Figure 1.2.2**

**Note:** The solid hatch pattern is placed in the drawing with a **Z** coordinate value of **0** (zero).

**Step 3:** Move the hatch pattern to the top surface of the upper most circle.

Command: **M** ↵

Invokes the AutoCAD LT **Move** command.

Select objects: **Select the Hatch pattern** ↵

Specify base point or displacement: **Pick any point on the screen**

Specify second point of displacement or <use first point as displacement>:

**@0,0,21** ↵

Moves the hatch pattern **21** units along the **Z** axis.

**Exercise:** Now do the same thing for Circle 2 (**Figure 1.2.1**), creating a solid hatch pattern and moving it **7** units up the **Z** axis. Once this is complete, invoke the **Shade** command.

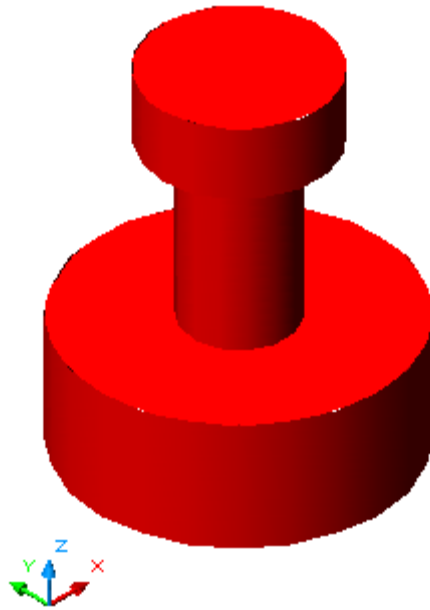


Figure 1.2.3

**Note:** The solid hatch is subject to the **FILLMODE** setting and will not display in a 2D Wireframe if the setting is turned off. Refer to the AutoCAD LT help for further information about **FILLMODE**.

### 2.3.2 2D Solid

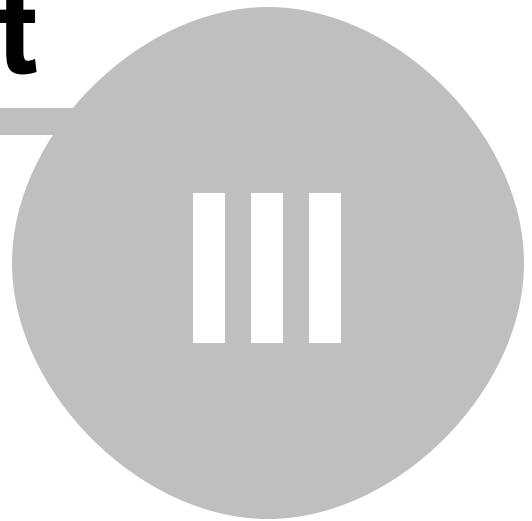
The **2D Solid** command is limited to rectilinear shapes, so although it can be quite useful in emulating 3D surfaces, it may prove easier to use the Solid Hatch as described in the previous example.

**Note:** Refer to the AutoCAD LT help for further information on the 2D Solid command. This command is native to AutoCAD LT but is useful when used with an altered **THICKNESS** setting.

**Exercise:** Change the Thickness setting and experiment with the 2D Solid command. **SO** or **SOLID** can be entered on the command line to run the command.

**Note:** Any entities created to this point are not necessary for further exercises, you can either delete them, pan them off the screen, or save them and start a new drawing.

**Part**



## 3 Tutorial 2

### 3.1 Exercise 1

Tutorial one looked at using some standard AutoCAD LT commands to simulate 3D objects. In this tutorial and the ones to follow, we will be using the 3D modelling commands introduced by LT3D.

In the Help file for full AutoCAD, a 3D face is described as "*a 3 or 4 sided surface anywhere in space*". It also states that the corners of the 3D face can have varying Z coordinate values, but this does prevent it from being extruded. We will look at extrusions in a later exercise.

### 3.1.1 3D Faces

In this exercise we'll look at a number of options using the 3D Face command.

#### Creating 3D faces

**Step 1:** Clear the screen of any objects created in previous exercises or start a new drawing.

**Step 2:** Use the **3D Face** command to draw a similar shape to **Figure 2.1.1**.

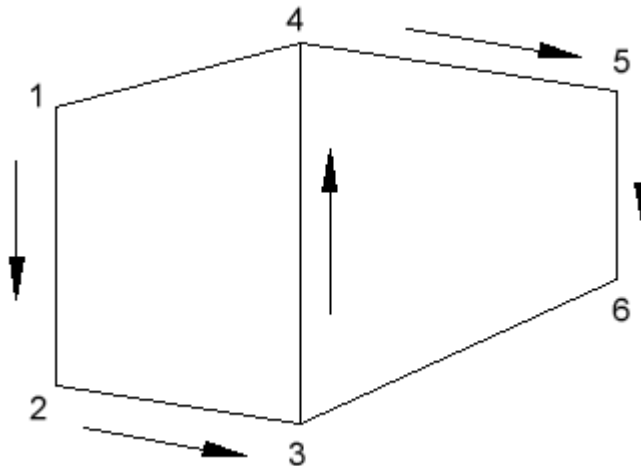


Figure 2.1.1

Command: **3dface** ↵

Invokes the 3D Face command.

**Note:** Pick points on the screen, following the sequence in **Figure 2.1.1** to create a similar shape.

Specify first point or [invisible]: **Pick Point 1**

Specify second point or [invisible]: **Pick Point 2**

Specify third point or [invisible] <exit>: **Pick Point 3**

Specify fourth point or [invisible] <create three-sided face>: **Pick Point 4**

**Note:** The last segment was automatically generated, closing the shape.

Specify third point or [invisible] <exit>: **Pick Point 5**

Specify fourth point or [invisible] <create three-sided face>: **Pick Point 6**

**Note:** The shape is once again automatically closed.

Specify third point or [invisible] <exit>: ↵

Exits the command.

**Note:** 3D Faces can only be drawn with a maximum of 4 sides, hence the automatic closure.

**Step 3:** Use **SHADE** to view the 3D Faces with colour applied, then used **SHADEMODE** and restore the object to a 2D wire frame.

## The Edge Command

Unwanted edges of a 3D Face can be made invisible by using the **Edge** command. This command is used once the faces have been generated.

**Step 1:** Invoke the **Edge** command

Command: **EDGE** ↵  
Command line entry for **Edge** command.

Select Object to change Visibility: **Pick the edge between points 3 and 4.**  
Makes the right-hand edge of the first 3D face invisible.

Select 3d edge to make Invisible...<Exit>: **Pick the edge between points 3 and 4.**  
Makes the left-hand edge of the second 3D face invisible.

Select 3d edge to make Invisible...<Exit>: ↵  
Exists the command.

**Note:** Remember there are two rectilinear faces (4 sides each), hence the need to pick twice to hide both overlapping edges.

**Step 2:** Use **SHADE** to view the 3D Faces with colour applied, then used **SHADEMODE** and restore the object to a 2D wire frame.

## Creating 3D Faces with an Invisible Edge

The **Edge** command described above, is ideal for making edges invisible once the 3D faces have been drawn, but it is also possible to make edges of faces invisible as you draw them. This is achieved as follows:

**Step 1:** Make sure the drawing is a 2D wireframe and that you have enough room on the screen to continue drawing.

**Step 2:** Invoke the **3D Face** command and draw a similar shape to last one, but this time take note of where the edges are set invisible. **Figure 2.1.2.**

Command: **3dface** ↵  
Invokes the 3D Face command.

**Note:** Pick points on the screen, following the sequence in **Figure 2.1.2** to create a similar shape.

Specify first point or [Invisible]: **Pick Point 1**

Specify second point or [Invisible]: **Pick Point 2**

Specify third point or [Invisible] <ext>: **i** ↵

Sets the Invisible option.

Specify third point or [Invisible] <exit>: **Pick Point 3**

Specify fourth point or [Invisible] <create three-sided face>: **Pick Point 4**

**Note:** The edge between points 3 and 4 is invisible.

Specify third point or [Invisible] <exit>: **Pick Point 5**

Specify fourth point or [Invisible] <create three-sided face>: **Pick Point 6**

Specify third point or [Invisible] <exit>: ↵

**Note:** The two faces are drawn with the common face invisible.

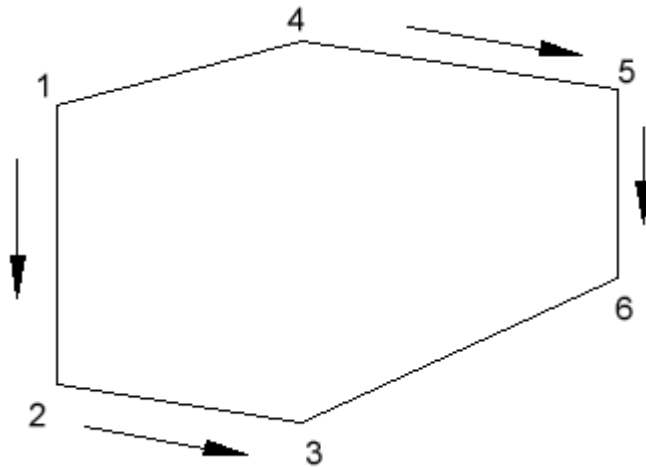


Figure 2.1.2

### 3.1.2 Creating Openings in Faces

We are going to generate a simple example of a shape with an opening.

**Step1:** Use the AutoCAD LT **Rectangle** command to draw a rectangle with the following sizes.

**X = 10**

**Y = 7**

**Step 2:** **Offset** the rectangle, 3 units to the inside. Refer to **Figure 2.1.3**

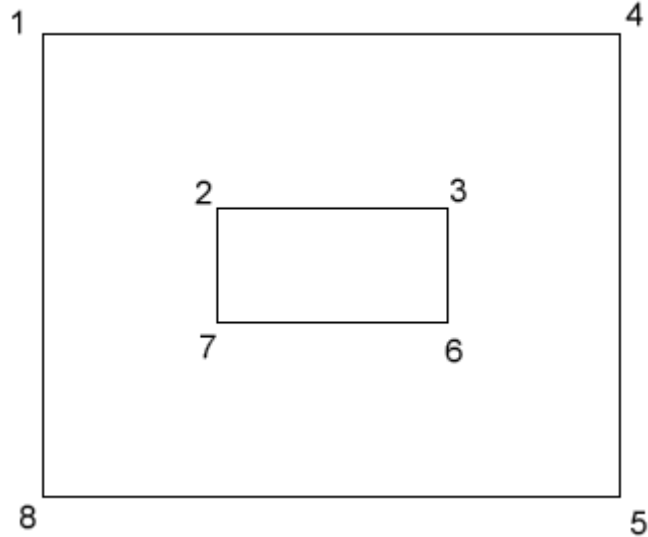


Figure 2.1.3

**Step 3:** Use the **3D Face** command to create a surface between the two rectangles as follows: Refer to **Figure 2.1.3**.

Command: **3DFACE** ↵

Invokes the **3D Face** command.

```
Specify first point or [Invisible]: Pick Point 1
Specify second point or [Invisible]: Pick Point 2
Specify third point or [Invisible] <exit>: Pick Point 3
Specify fourth point or [Invisible] <create three-sided face>: Pick Point 4
Specify third point or [Invisible] <exit>: Pick Point 5
Specify fourth point or [Invisible] <create three-sided face>: Pick Point 6
Specify third point or [Invisible] <exit>: Pick Point 7
Specify fourth point or [Invisible] <create three-sided face>: Pick Point 8
Specify third point or [Invisible] <exit>: Pick Point 1
Specify fourth point or [Invisible] <create three-sided face>: Pick Point 2
Specify third point or [Invisible] <exit>: ↵
```

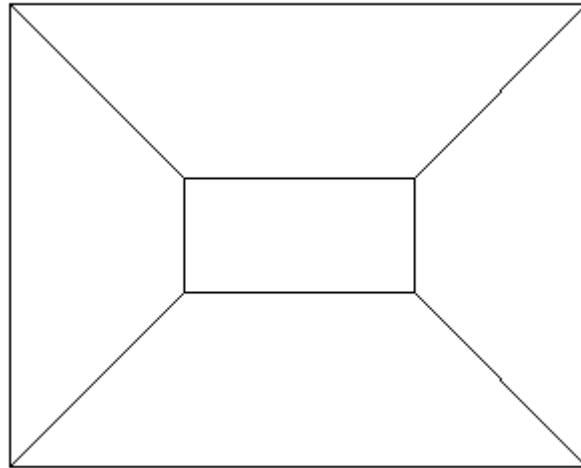


Figure 2.1.4

**Exercise:** Use the **EDGE** command to make the edges between the 3D Faces invisible.

**Note:** Each corner of the 3D Face can have a different Z coordinate value. This can be achieved by drawing the 3D faces to points that are already at set heights above the XY plane, or by stretching or dragging the corners or the 3D Faces with grips.

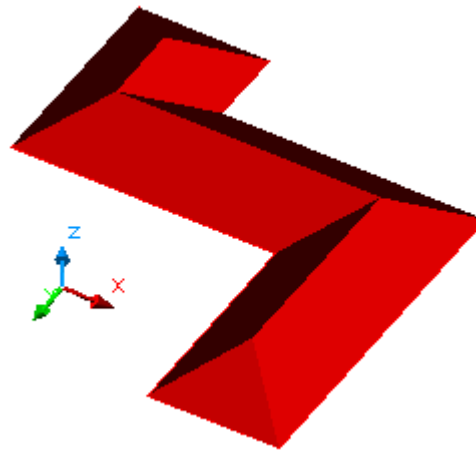


Figure 2.1.5

**Note:** Any entities created to this point are not necessary for further exercises, you can either delete them, pan them off the screen, or save them and start a new drawing.

## 3.2 Exercise 2

In this exercise you will learn how to create a varied number of surfaces, using the 3D Mesh, Revolved Surface, Tabulated Surface, Ruled Surface and Edge Surface commands.

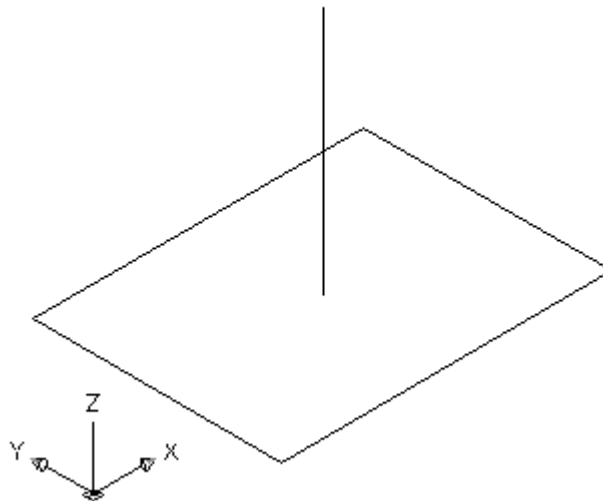
### 3.2.1 Poly Face

The AutoCAD help describes the PFACE command as producing a "polyface (polygon) mesh, with each face capable of having numerous vertices".

**Step 1:** Draw the entities in **Figure 2.2.1** using standard AutoCAD commands:

**Rectangle:** 4 units x 3 units (use Rectangle command)

**Line from the centre of the rectangle @0,0,3**



**Figure 2.2.1**

**Step 2:** Use the **PFACE** command to create pyramidal shape.

**Note:** The object we're about to create is made up of 5 surfaces - the bottom and four triangular sides. In the first part of the command we nominate all the endpoints of the vertices. Make use of snap modes. Refer to **Figure 2.2.2**.

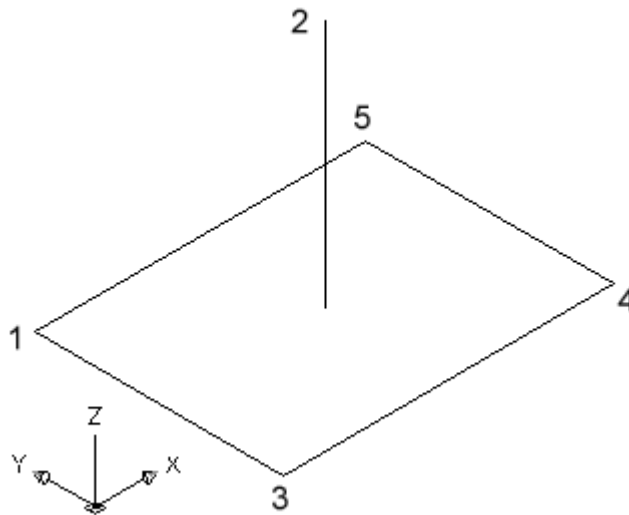


Figure 2.2.2

Command: **PFACE** ↵

Specify location for vertex 1: **Pick Point 1**

Specify location for vertex 2 or <define faces>: **Pick Point 2**

Specify location for vertex 3 or <define faces>: **Pick Point 3**

Specify location for vertex 4 or <define faces>: **Pick Point 4**

Specify location for vertex 5 or <define faces>: **Pick Point 5**

Specify location for vertex 6 or <define faces>: ↵

**Note:** The second part of the command is to specify the vertices that make up the 5 different faces.

Face 1, vertex 1:

Enter a vertex number or [Color/Layer]: **1** ↵

Begin entering the vertices that make up the first face.

Face 1, vertex 2:

Enter a vertex number or [Color/Layer] <next face>: **2** ↵

Face 1, vertex 3:

Enter a vertex number or [Color/Layer] <next face>: **3** ↵

Face 1, vertex 4:

Enter a vertex number or [Color/Layer] <next face>: ↵

The return button closes the face between points 1, 2 and 3 and begins the process for the next face.

Face 2, vertex 1:

Enter a vertex number or [Color/Layer]: **3** ↵

Begin entering the vertices that make up the second face.

Face 2, vertex 2:

Enter a vertex number or [C o b r / L a y e r] <next face>: 2 ↵

Face 2, vertex 3:

Enter a vertex number or [C o b r / L a y e r] <next face>: 4 ↵

Face 2, vertex 4:

Enter a vertex number or [C o b r / L a y e r] <next face>: ↵

Face 3, vertex 1:

Enter a vertex number or [C o b r / L a y e r]: 4 ↵

Begin entering the vertexes that make up the third face.

Face 3, vertex 2:

Enter a vertex number or [C o b r / L a y e r] <next face>: 2 ↵

Face 3, vertex 3:

Enter a vertex number or [C o b r / L a y e r] <next face>: 5 ↵

Face 3, vertex 4:

Enter a vertex number or [C o b r / L a y e r] <next face>: ↵

Face 4, vertex 1:

Enter a vertex number or [C o b r / L a y e r]: 5 ↵

Begin entering the vertexes that make up the fourth face.

Face 4, vertex 2:

Enter a vertex number or [C o b r / L a y e r] <next face>: 2 ↵

Face 4, vertex 3:

Enter a vertex number or [C o b r / L a y e r] <next face>: 1 ↵

Face 4, vertex 4:

Enter a vertex number or [C o b r / L a y e r] <next face>: ↵

Face 5, vertex 1:

Enter a vertex number or [C o b r / L a y e r]: 1 ↵

Begin entering the vertexes that make up the fifth face.

Face 5, vertex 2:

Enter a vertex number or [C o b r / L a y e r] <next face>: 3 ↵

Face 5, vertex 3:

Enter a vertex number or [C o b r / L a y e r] <next face>: 4 ↵

Face 5, vertex 4:

Enter a vertex number or [C o b r / L a y e r] <next face>: 5 ↵

Face 5, vertex 5:

Enter a vertex number or [C o b r / L a y e r] <next face>: ↵

Completes the 5<sup>th</sup> face.

Face 6, vertex 1:

Enter a vertex number or [C o b r / L a y e r]: ↵

Exits the **PFACE** command

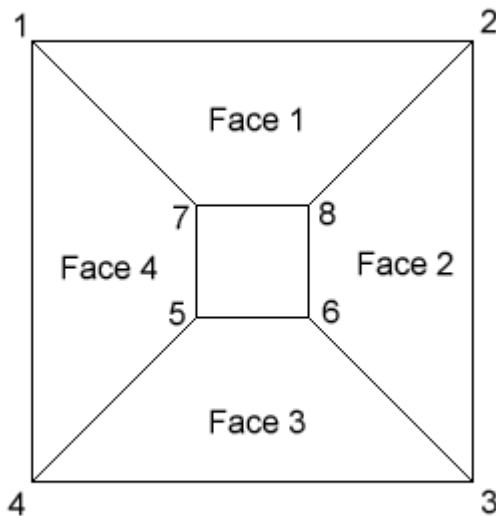
## Invisible Edges

- Edges can be set invisible by placing a minus sign in front of the starting vertex. For example, if I wanted the edges between 1 & 5, 2 & 6, 3 & 7 and 4 & 8 (Refer to **Figure 2.2.3**) to be invisible, I would enter the vertexes for **Face 1** as follows:

```
-1 ↵
Forces edge 1 to 5 to be invisible.
5 ↵
-6 ↵
Forces edge 6 to 2 to be invisible.
2 ↵
```

The vertexes for **Face 2** would be entered as follows:

```
-2 ↵
Forces edge 2 to 6 to be invisible.
6 ↵
-7 ↵
Forces edge 7 to 3 to be invisible.
3 ↵
```



**Figure 2.2.3**

**Exercise:** Draw two concentric squares as in **Figure 2.2.3** and apply the PFACE command with invisible edges using the principle laid out above.

**Note:** The System Variable **SPLFRAME** can be used to display edges which have been drawn invisible, as shown in **Figure 2.2.4**. However, **SPLFRAME** will only display or hide edges you have set invisible with the minus value, it does not turn off otherwise visible edges.

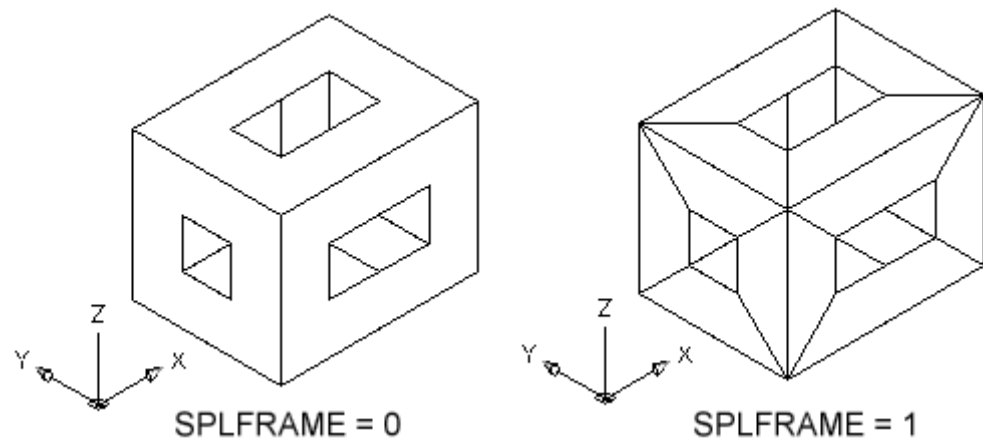


Figure 2.2.4

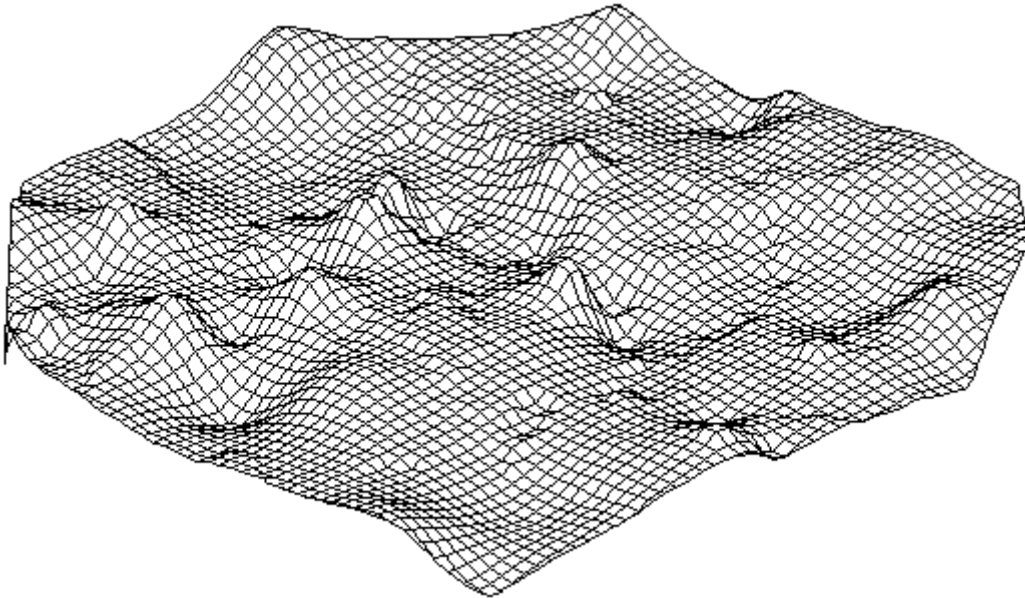
**Note:** Any entities created to this point are not necessary for further exercises, you can either delete them, pan them off the screen, or save them and start a new drawing.

### 3.2.2 3D Mesh

**The AutoCAD help states: "AutoCAD defines a polygon mesh by a matrix, the size of which is determined by  $M$  and  $N$  size values.  $M \times N$  equals the number of vertices that you must specify."**

An example of the use of this command, would be in the creation of a terrain model for architecture or surveying. However it is a not an easy command to use and creating complex shapes can be difficult. LT3D has a number of pre-defined surface shapes which we will look at in a future exercise.

**Figure 2.2.5** is an example of a topographical model of an undulating terrain.



**Figure 2.2.5**

**Step 1:** Invoke the **3DMESH** command to create a mesh.

Command: **3DMESH** ↵

Invokes the command.

**Step 2:** Enter the Number of vertices in the **M** direction. Refer to **Figure 2.2.6**.

Enter size of mesh in M direction: **4** ↵

**Note:** The **M** size can only be between 2 and 256.

**Step 3:** Enter the Number of vertices in the **N** direction. Refer to **Figure 2.2.5**.

Enter size of mesh in N direction: **3** ↵

**Note:** The **N** size can only be between 2 and 256.

**Step 4:** Specify the **X**, **Y** and **Z** coordinates for each vertex as prompted.

Specify location for vertex (0, 0): **10,1,3** ↵

Specify location for vertex (0, 1): **10,5,5** ↵

Specify location for vertex (0, 2): **10,10,3** ↵

Specify location for vertex (1, 0): **15,1,0** ↵

Specify location for vertex (1, 1): **15,5,0** ↵

Specify location for vertex (1, 2): **15,10,0** ↵

Specify location for vertex (2, 0): **20,1,0** ↵

Specify location for vertex (2, 1): **20,5, 1** ↵

Specify location for vertex (2, 2): **20,10,0** ↵

Specify location for vertex (3, 0): **25,1,0** ↵

Specify location for vertex (3, 1): **25,5,0** ↵

Specify location for vertex (3, 2): **25,10,0** ↵

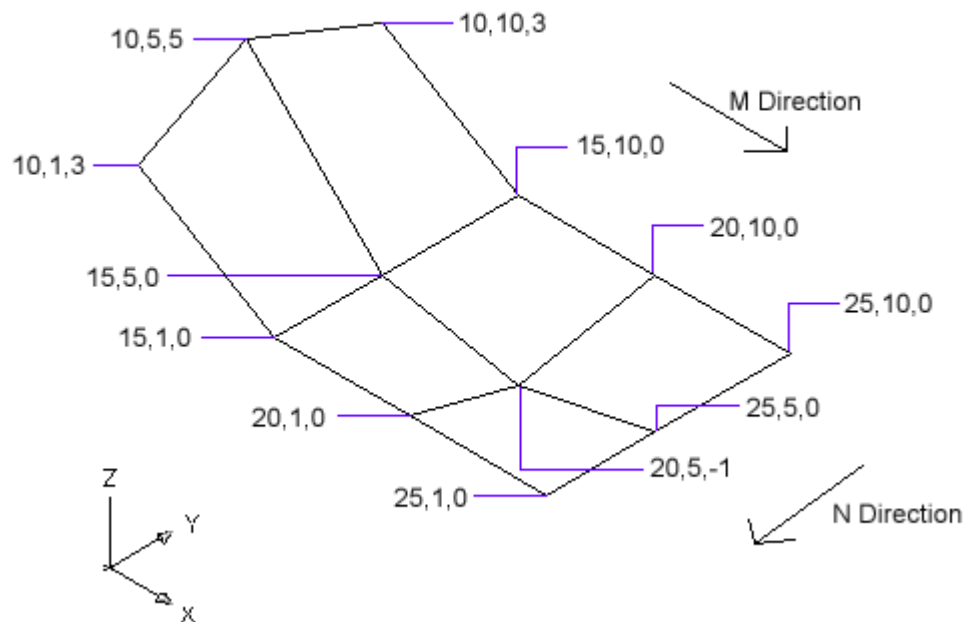


Figure 2.2.6

### 3.2.3 SURFTAB1 & SURFTAB2

In the following commands we will need to set the SURFTAB1 and SURFTAB2 System Variables. SURFTAB1 and SURFTAB2 control the *M* and *N* directions respectively, which will set the density of the mesh or number of facets in objects we are about to create.

The higher the values, the more dense the mesh, but it is not necessary to go overboard. However if the setting is too low, curved faces will appear faceted. Once again you must use an integer between 2 and 256. (Refer to the 3D Mesh).

The SURFTAB values must be set prior to creating surface objects. Existing surface objects will not update to new SURFTAB settings.

### 3.2.4 Revolved Surface

The AutoCAD help states that the REVSURF command "*creates a surface of revolution by rotating a profile of the object about an axis.*"

### 3.2.5 Tabulated Surface

Enter topic text here.

### 3.2.6 Ruled Surface

Enter topic text here.

### 3.2.7 Edge Surface

Enter topic text here.

### 3.2.8 Surface Shapes

Enter topic text here.

---

## Endnotes 2... (after index)

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