Pro/DESKTOP Computer Aided Design (CAD)

CAD Toolbox

First Principles of CAD using Feature Based Solid Modeling

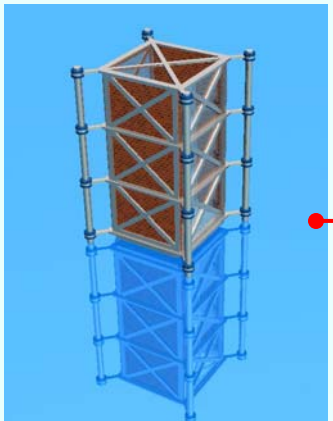
These pages describe how to use Pro/DESKTOP CAD software by guiding the learner through a series of solid modeling exercises. The exercises can be considered as elements of a CAD Toolbox that develops the learner's skill until they can use the tools at their command to build a broad range of CAD solid models.

The exercises are written with graphical images from Pro/DESKTOP to describe a CAD concept or command sequence. The exercises create an abstract outcome leaving the learner to transfer their skill and produce a product design suitable for manufacture.

Each exercise builds on the foundation of previous skills until the learner has gained expertise in the following solid modeling techniques:

- Extrude Profile
- Revolve Profile
- Sweep Profile
- Loft Through Profile

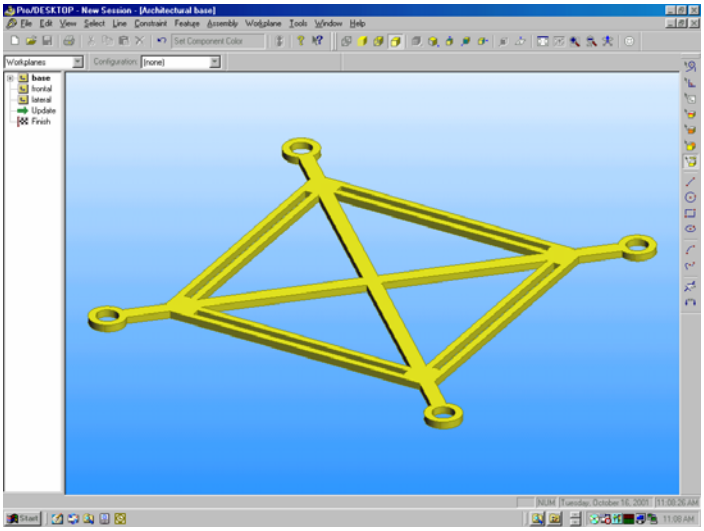
On completion of the full range of exercises the learner will have a CAD Toolbox that will enable them to use their skill to build solid models ranging from simple flat sheet designs to complex curvilinear forms.

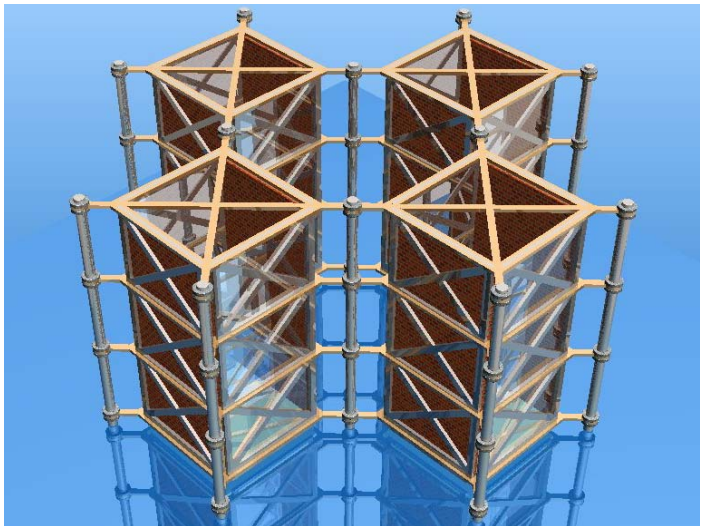


These Pro/DEKTOP exercises use simple geometric shapes such as circles, squares, rectangles and triangles to introduce the learner to Computer Aided Design (CAD).

The exercises describe how to use the solid modeling techniques in Pro/DESKTOP by producing a range of primitive objects.

The learner can use the skill sets gained from these tutorials to create their own unique designs.



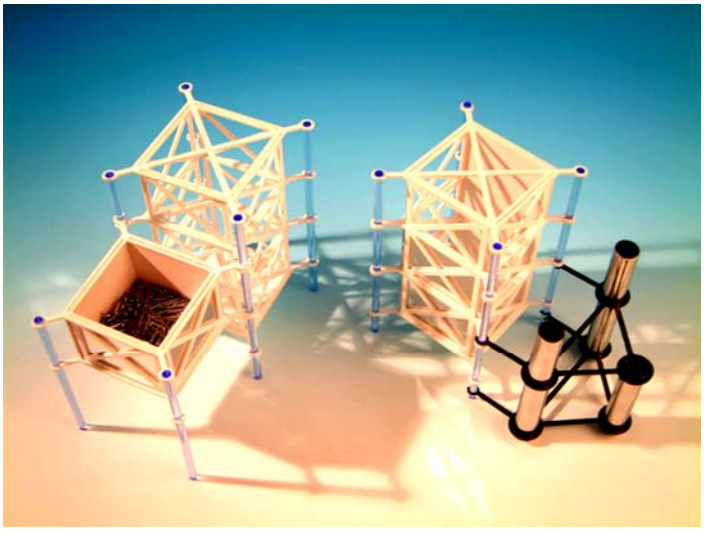


These Pro/DESKTOP images show a proposal for a desktop storage system. Architectural construction techniques have influenced the designs appearance.

The tutorials will enable the learner to transfer the knowledge gained from producing the primitive shapes to their own product designs.

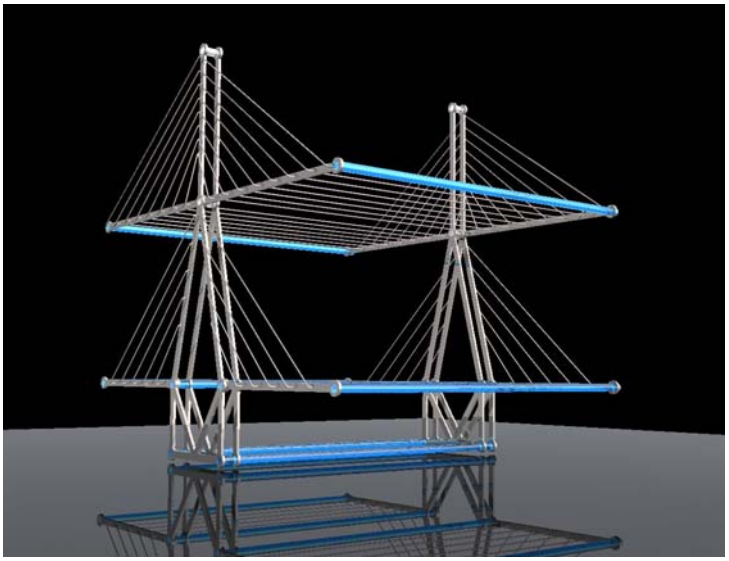
The tutorials are written for novice users of Pro/DESKTOP and explain in simple step by step illustrations how to use this exciting and creative software.

The final part of the tutorial explains how to take a design from Pro/DESKTOP and create a manufactured product.



Large scale civil engineering can be the inspiration for small scale product design. The cable stayed bridge shown below supports a deck for road traffic through a system of high tensile cables.

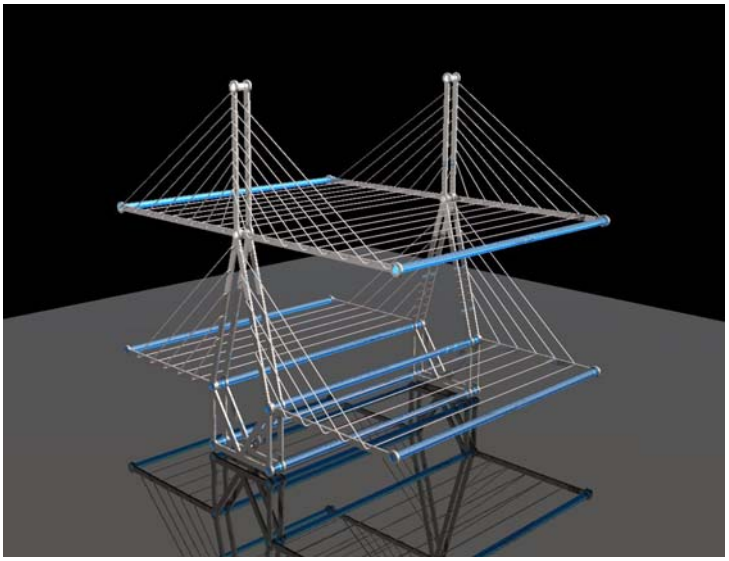
The same physical principles that make the bridge a successful design can be transferred in a design for a paper tray shown in these Pro/DESKTOP illustrations.



This type of project is an opportunity for the student to consider the aesthetic of a product design and the mathematical principles that ensure it performs under stresses and strains.

This approach emulates the way in which a professional engineer, architect or product designer works in industry.

This design has been produced on Pro/DESKTOP using only two simple shapes, a triangle and a circle. The solid modeling technique is EXTRUSION which is covered in Exercise 1 of this tutorial.



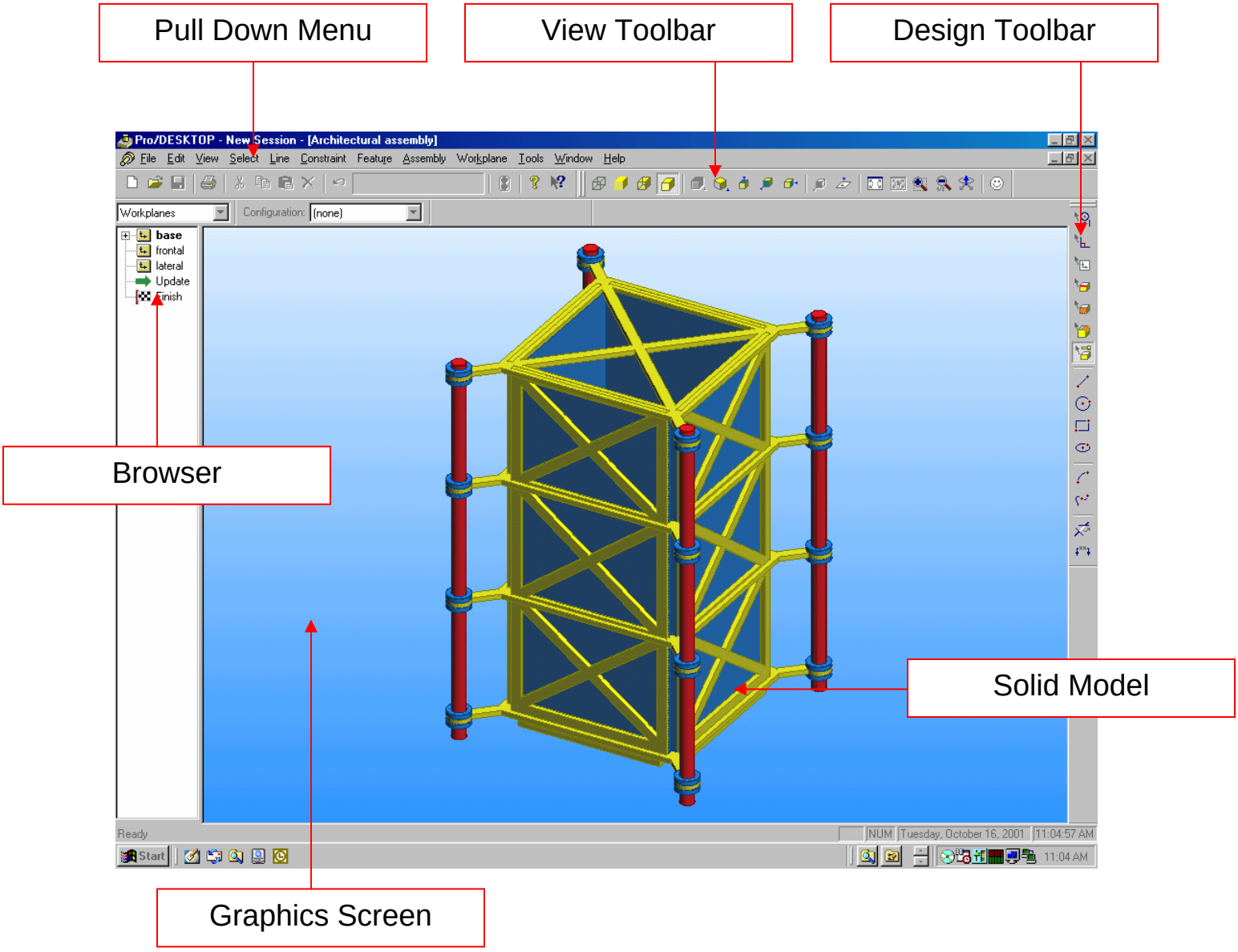
CAD/CAM Tutorial

Exercise 1

Create a cylindrical solid model using the Feature tool EXTRUSION in the Design environment of Pro/DESKTOP.

Pro/DESKTOP Interface

The instructions in this tutorial refer to the Pro/DESKTOP interface and toolbars. The illustration below describes the main elements of the graphical interface and toolbars.



To achieve the target designs of the Building Blocks of Pro/DESKTOP you will need to access the drawing and solid modeling tools. Please refer to this illustration to help you locate the tools and complete the instructions.

Exercise one:

Create a cylindrical solid model using the Feature tool EXTRUSION in the Design environment of Pro/DESKTOP.

Learning Outcomes:

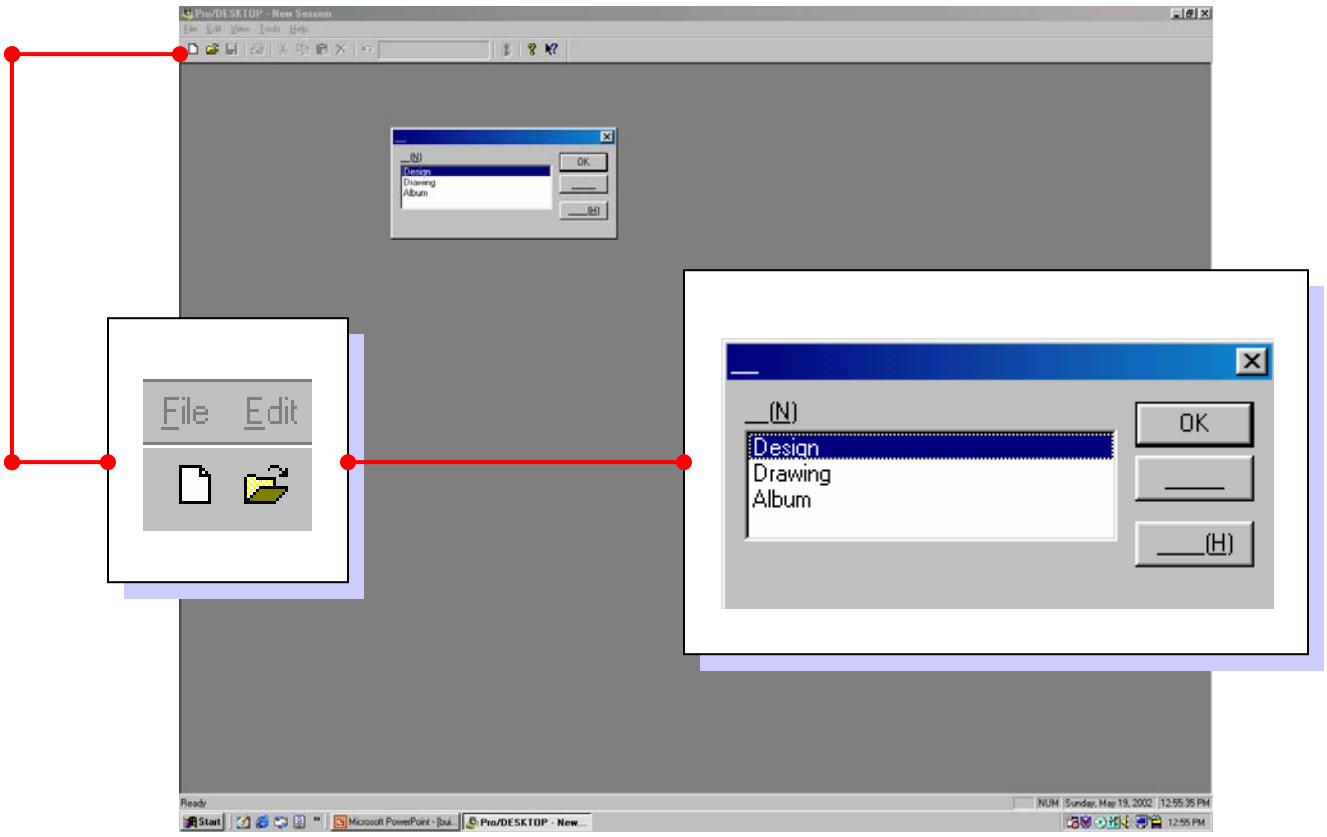
1.1 Open a Design File in the Design Environment of Pro/DESKTOP.

1.2 Create a sketch on a Workplane using the Design Object Browser.

1.3 Create a 2D-sketch object and add dimension constraints.

1.4 Edit dimension constraints to establish the size of the 2D-sketch object.

1.5 Use the Extrude Profile feature creation tool to create a 3D-Object.

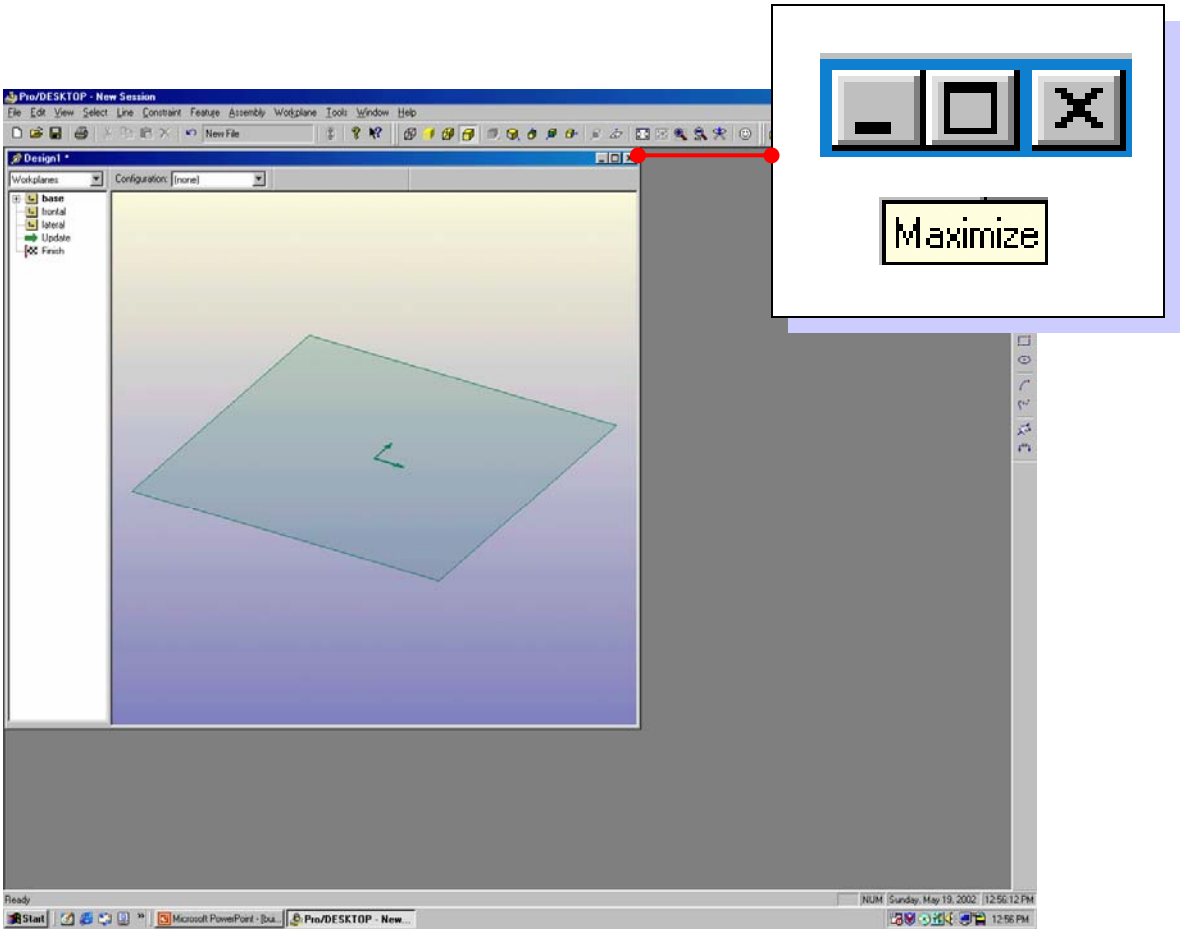


When you open the Pro/DESKTOP CAD software an interface appears on screen. There are three environments in Pro/DESKTOP that enable the user to concurrently create a solid model, an orthographic drawing and a photo-realistic album rendering. This exercise will guide you through creating a solid model in the Design environment.

From the Pull Down Menu select the white page icon.

This icon allows you to open a new file in Pro/DESKTOP.

From the New File box select Design and click on OK.

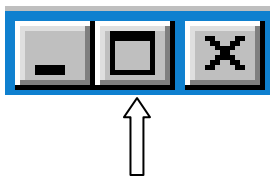


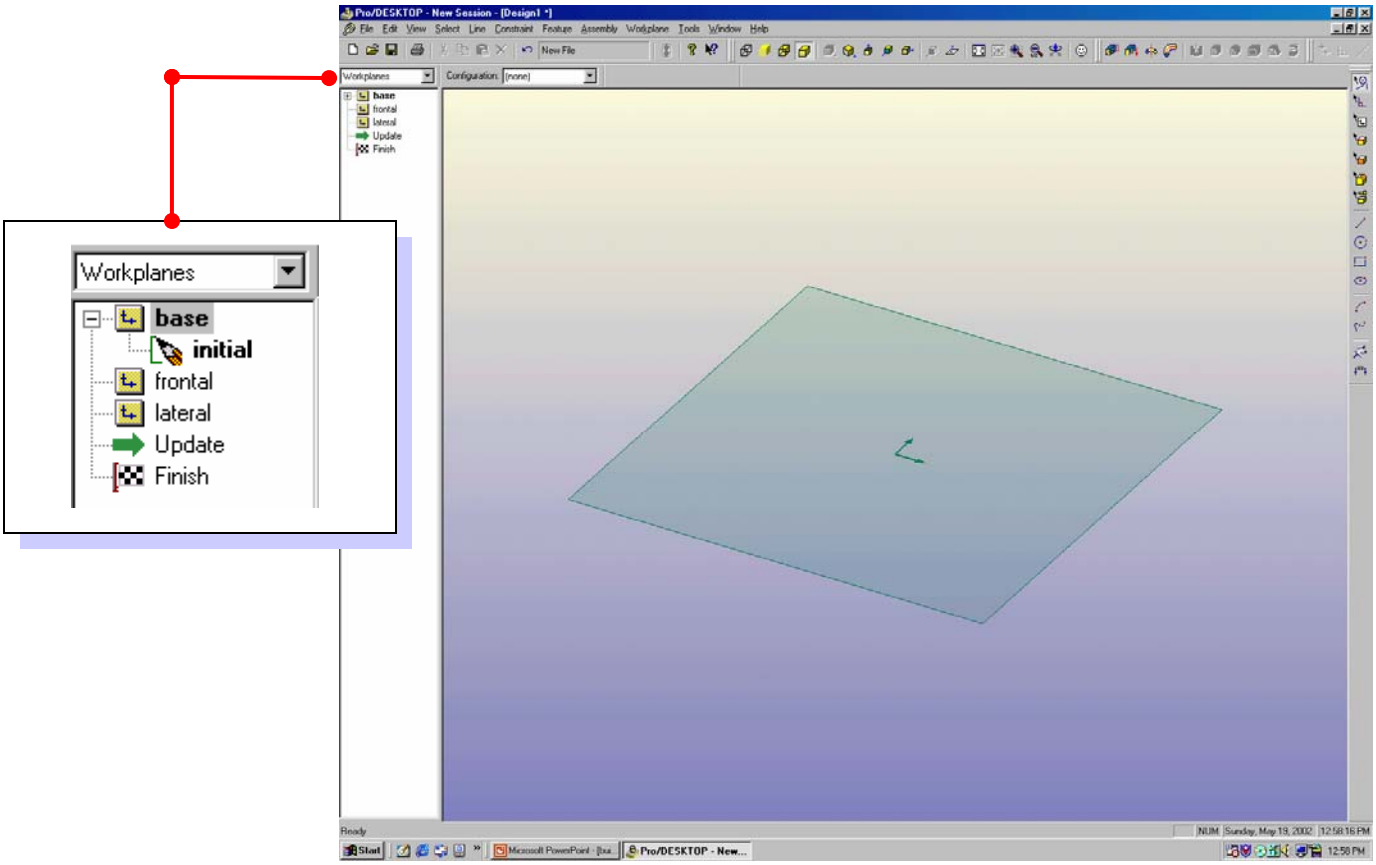
The Design interface appears on screen.

This interface enables you to draw two-dimensional shapes and through using the Feature Creation menu turn those shapes into three-dimensional solid models.

Your first task is to force the Design interface to fill the computer screen area.

From the extreme top right hand side of the Design interface left click (LC) on the square icon to maximize the interface on your computer screen.



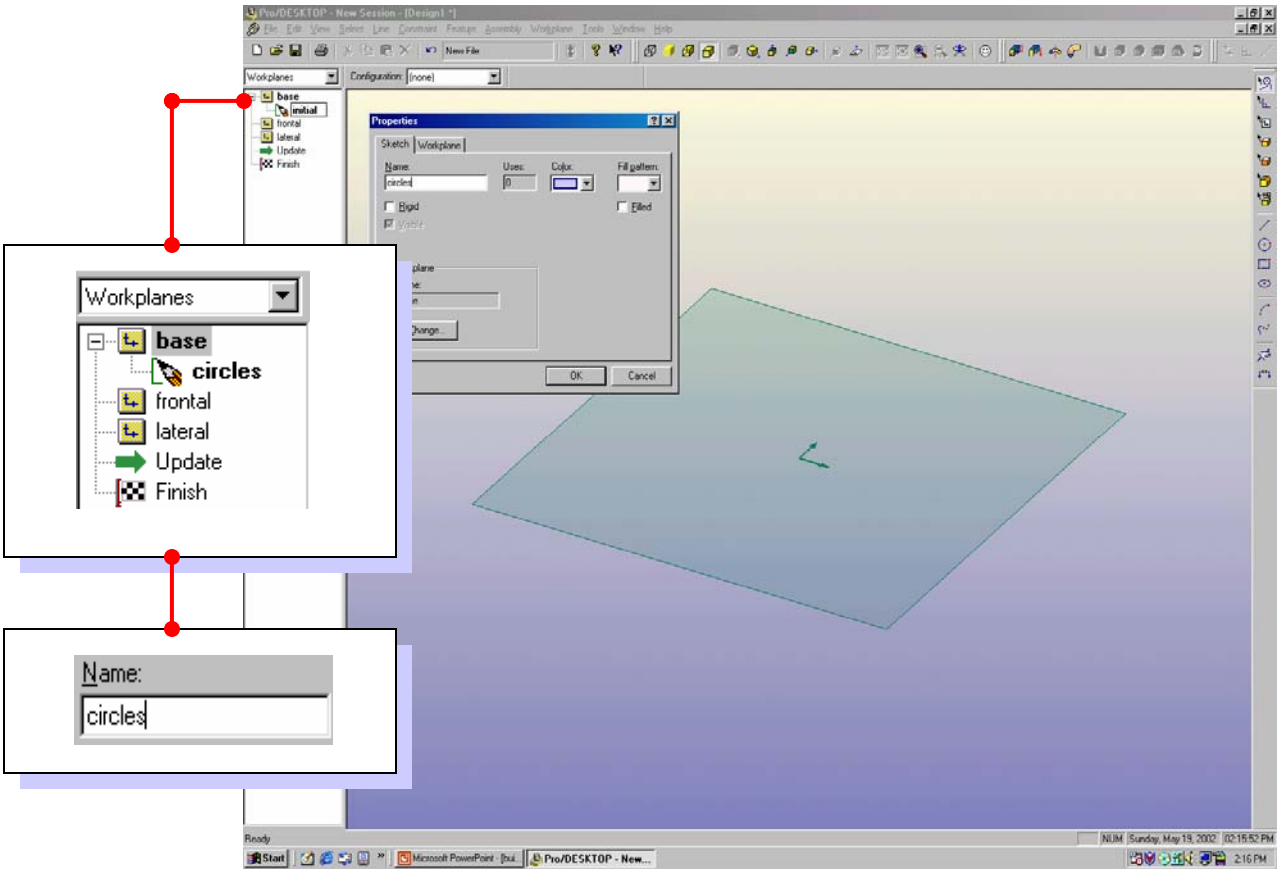


The image shows a square shaded area known as a Workplane. A Workplane is a platform that enables you to draw and dimension two-dimensional shapes. Automatically located on the Base Workplane is a Sketch with an axis to show the intersection of the X and Y plane at a coordinate position of 0,0.

An earlier page in this tutorial illustrated the Design interface and showed the position of the Browser. The Browser contains three menus relating to Workplanes, Features and Components.

The default situation for the Browser is to show information about Workplanes. LC on the plus sign next to the base Workplane to reveal the title of the first Sketch.

The first Sketch is titled initial; in the next sequence you will change the title to describe the type of shape to be drawn on it.

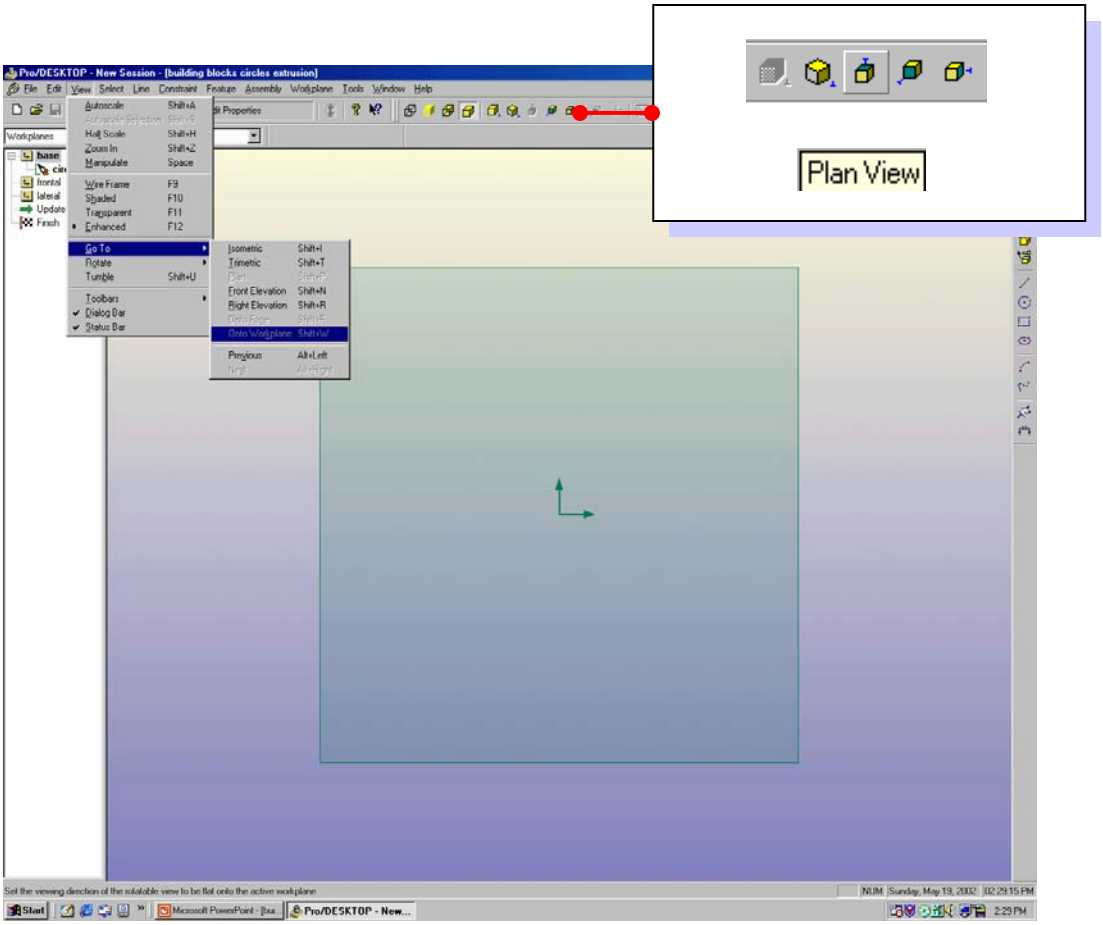


Double Left Click (DLC) on the default name for the Sketch titled initial in the Workplanes browser.

A Properties box appears with information relating to the Sketch and Workplane. Click in the Name window and change the title of the Sketch from initial to circles.

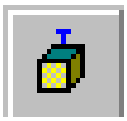
As you develop your skill with Pro/DESKTOP you will build solid models that contain many Workplanes and Sketches and it is important to use the best practice of the title reflecting the content of the Sketch.

Click on OK to complete the command and observe that the title has changed in the Browser.



The Workplane can be repositioned to make drawing two-dimensional shapes easier.

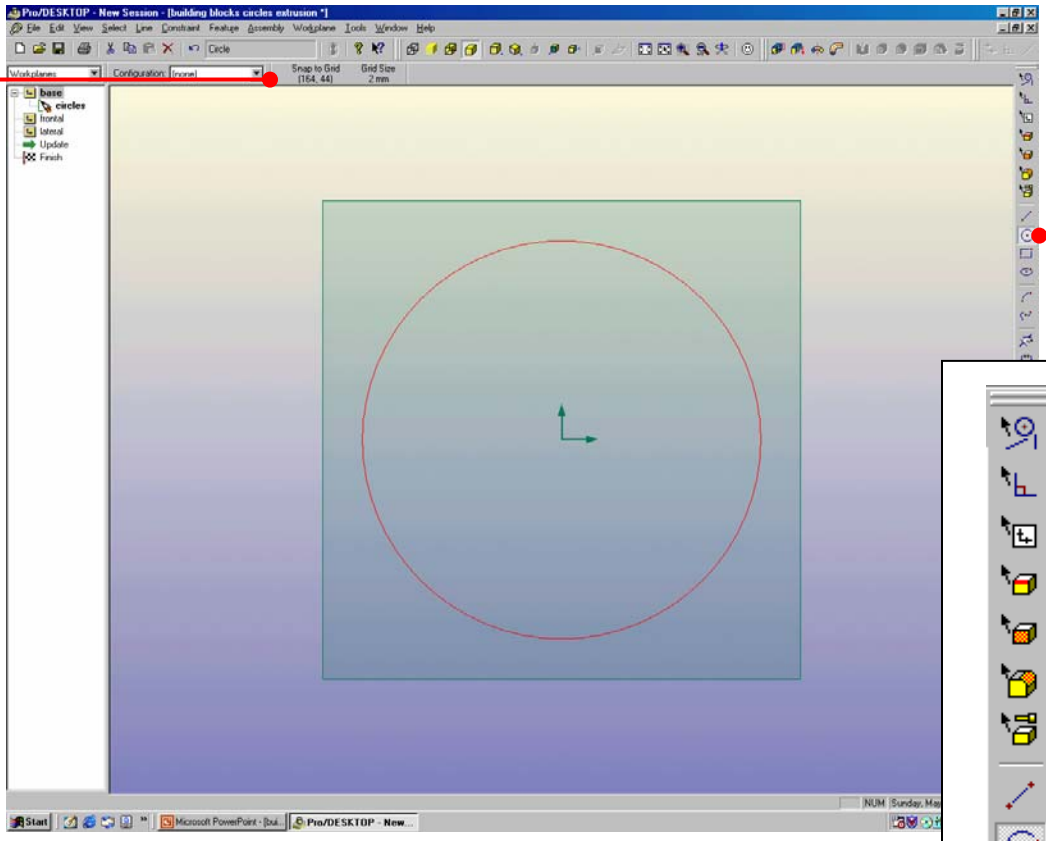
LC on the yellow cube icon with the top surface shaded in gray from the Views Toolbar

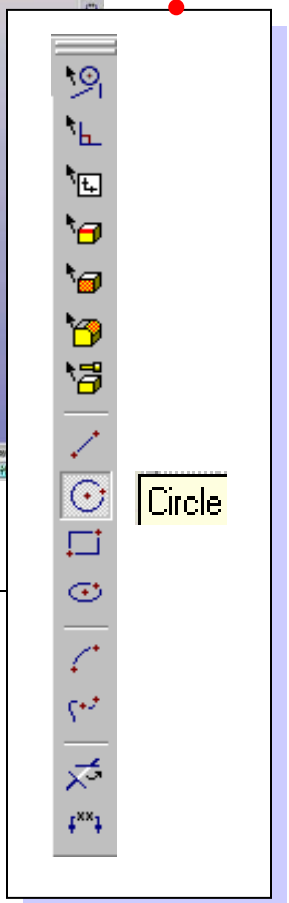


The Workplane will change to an orthographic plan view.

An alternative command sequence is from the Pull Down Menu:

View / Go To / Onto Workplane.



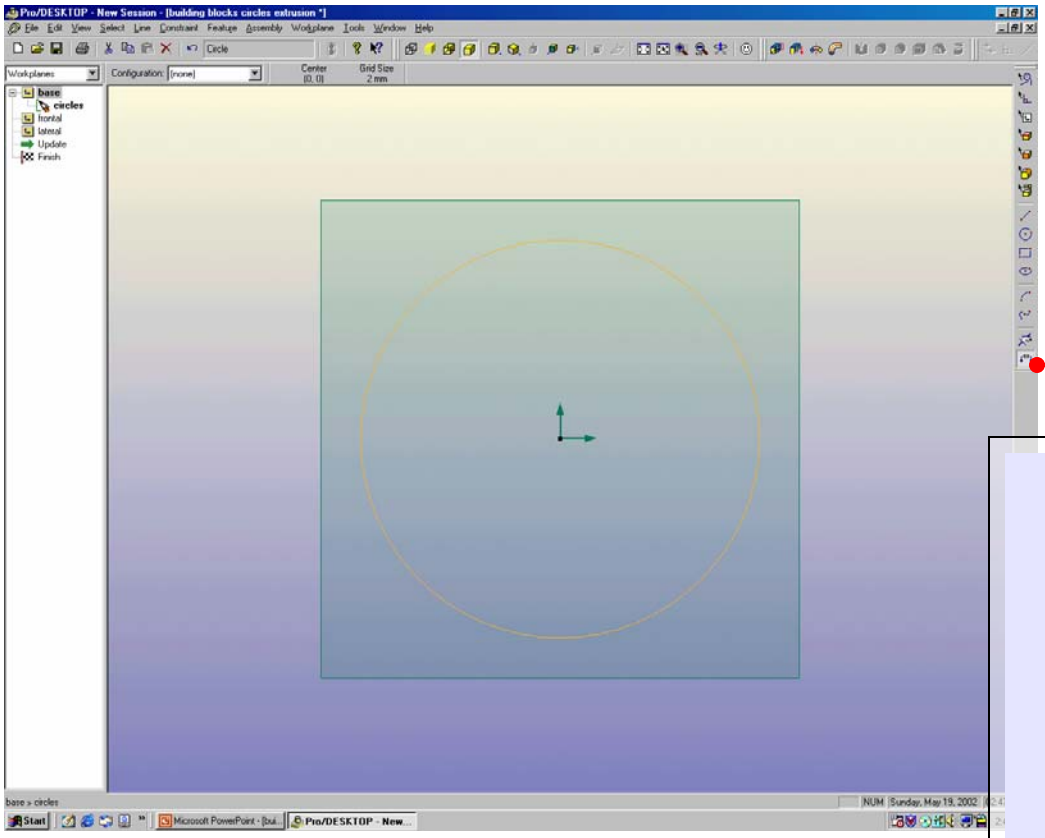


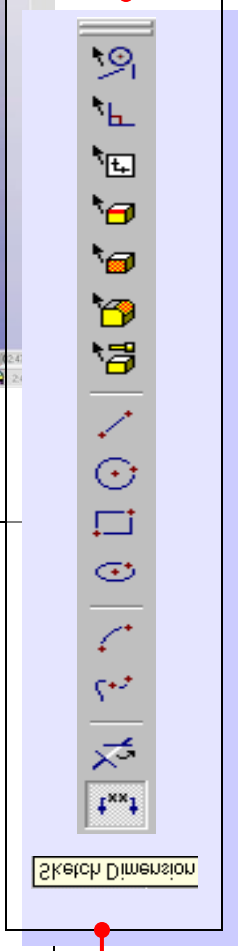
Click on the Circle icon from the Design Toolbar. Your cursor Will turn into a pencil, as you move the pencil across the screen A coordinate readout will trace the position in the X and Y plane.

Move the pencil to the intersection of the axis until the following Coordinate readout is achieved.

Snap to Grid	Grid Size
(0, 0)	2 mm

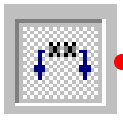
Hold down the Left Mouse Button and drag the pencil to create a circle to the proportions shown in the snapshot. Release the mouse to complete the circle command.



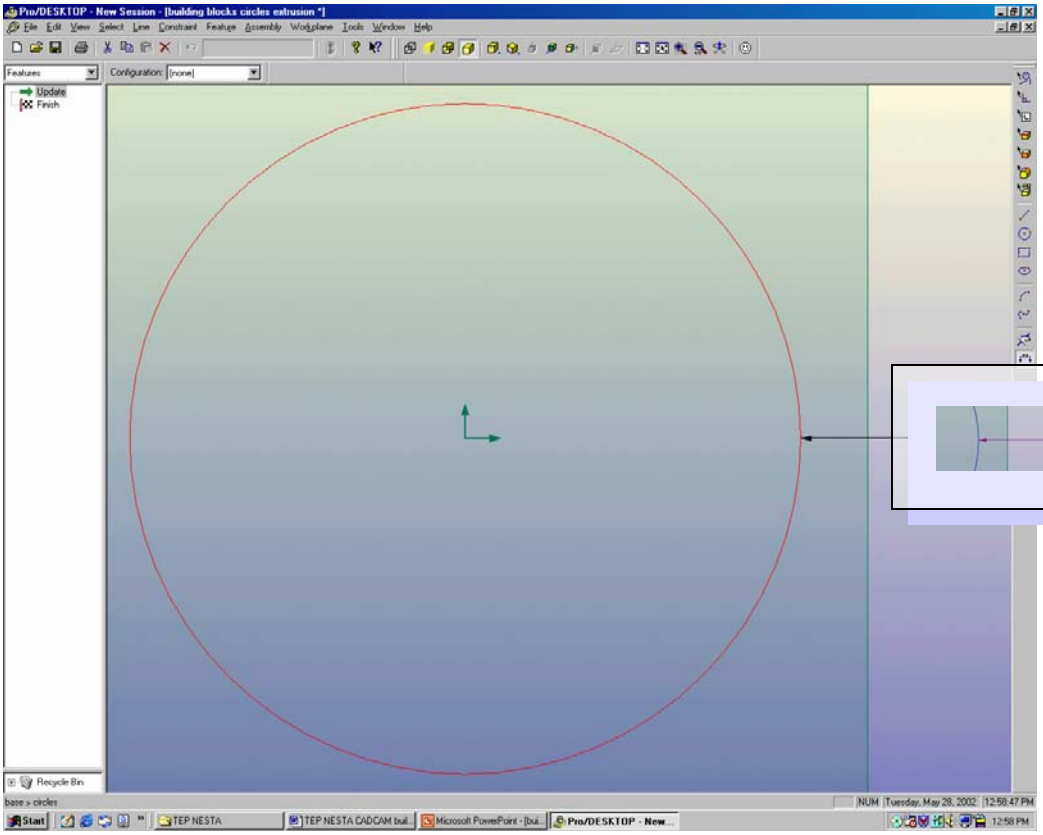


Pro/DESKTOP uses a Parametric system of dimensioning. This means that the dimensions attached to the two-dimensional shapes not only inform the user of the size but they also control or drive the size. Dimension driven CAD technology offers the user a great deal of creative freedom.

Click on the Sketch Dimension icon in the Design Toolbar.



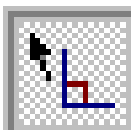
This tool will allow you to attach a diametric dimension to the circle.

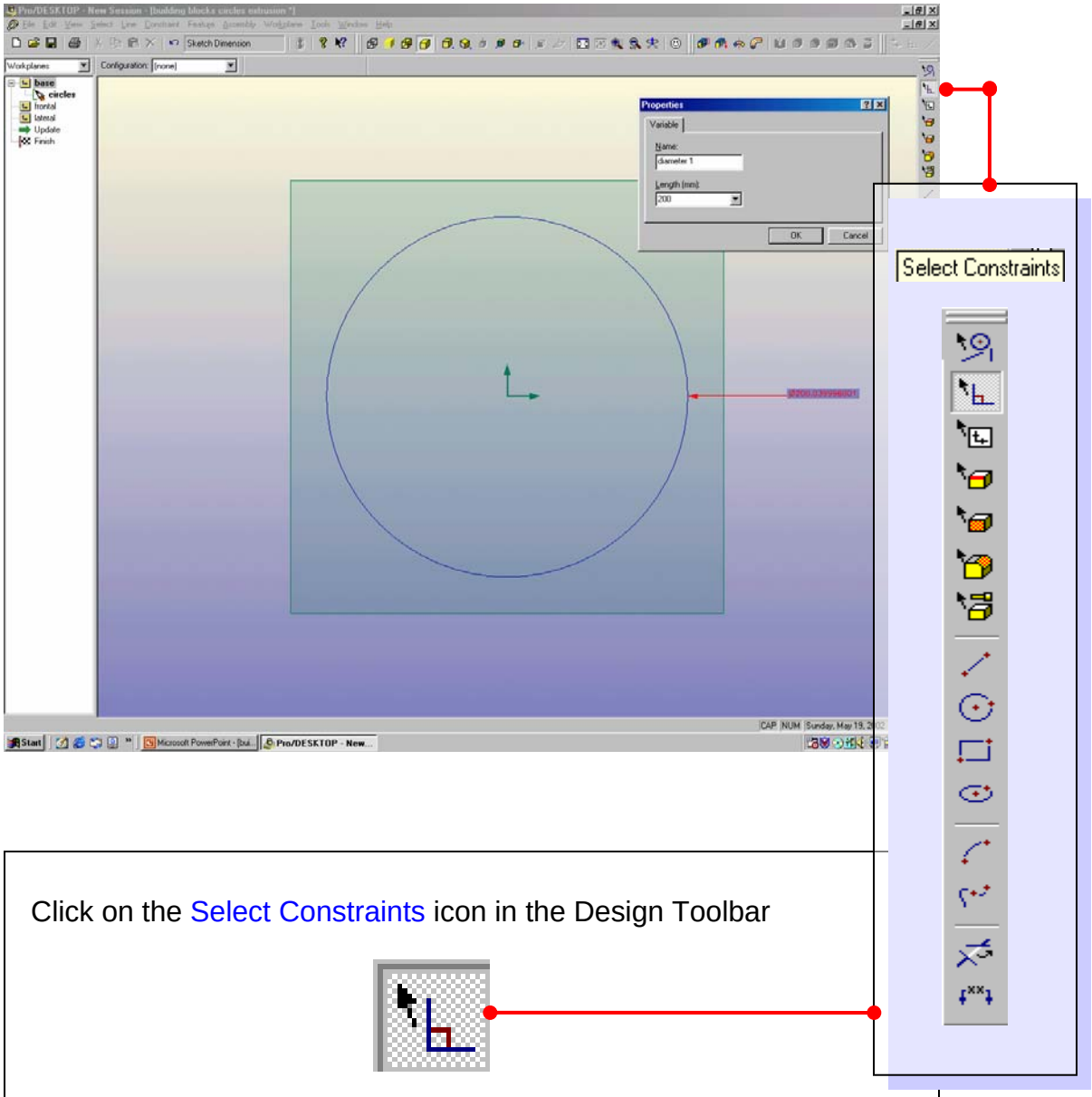


Move the cursor until it touches the circumference of the circle. The circle will highlight (change colour) to show it is selected. Hold down the Left Mouse Button (LMB) and drag the mouse out to the right to create a dimension constraint.

Earlier in this tutorial we described the software as *Dimension Driven.* The dimension that you have just attached to the circle will control its diameter and force the circle to an accurate value.

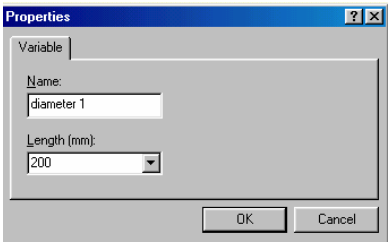
In the next sequence you will use the Select Constraints tool to change the diameter to an exact value of 200mm.



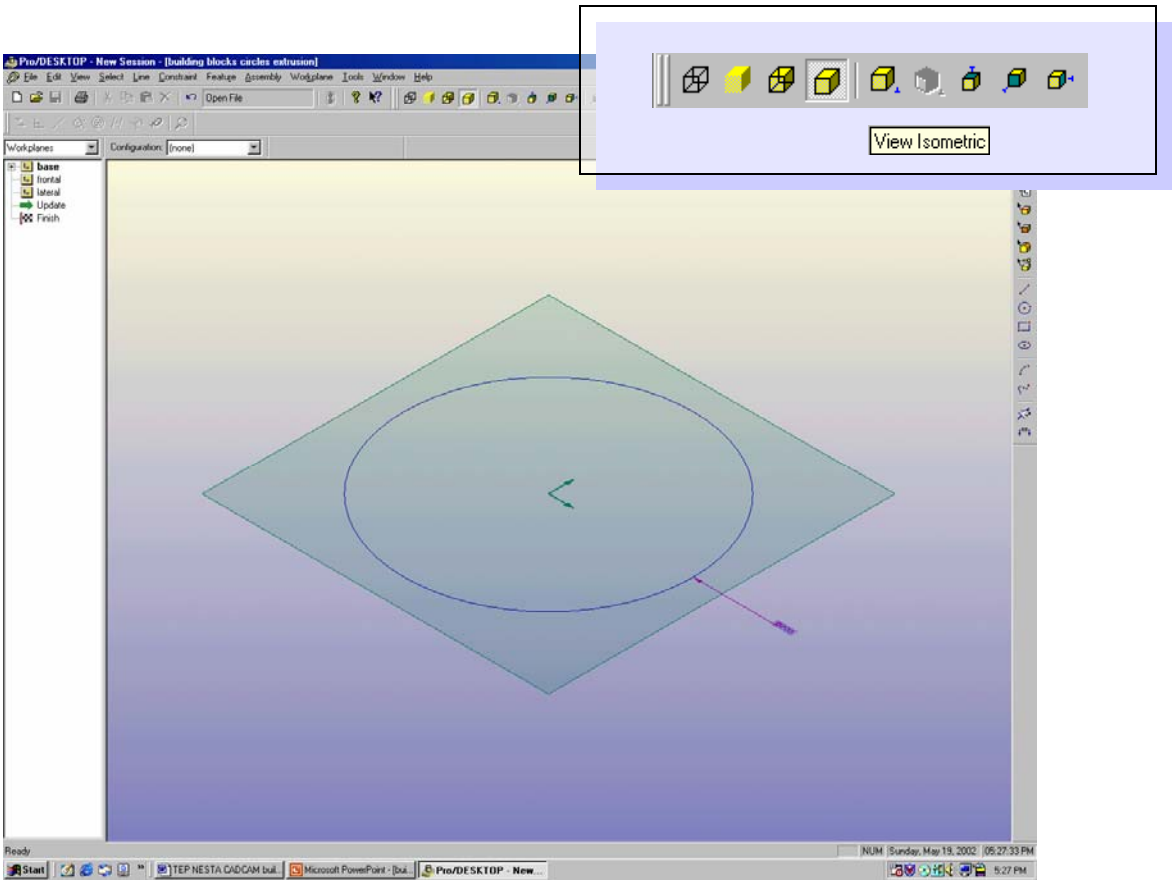


Click on the Select Constraints icon in the Design Toolbar

Move the cursor over the numbers of the dimension you have just attached to the diameter and DLC to gain access to the Properties box.



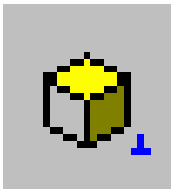
Change the Length value to a diameter of 200 mm and click on OK to complete the command.

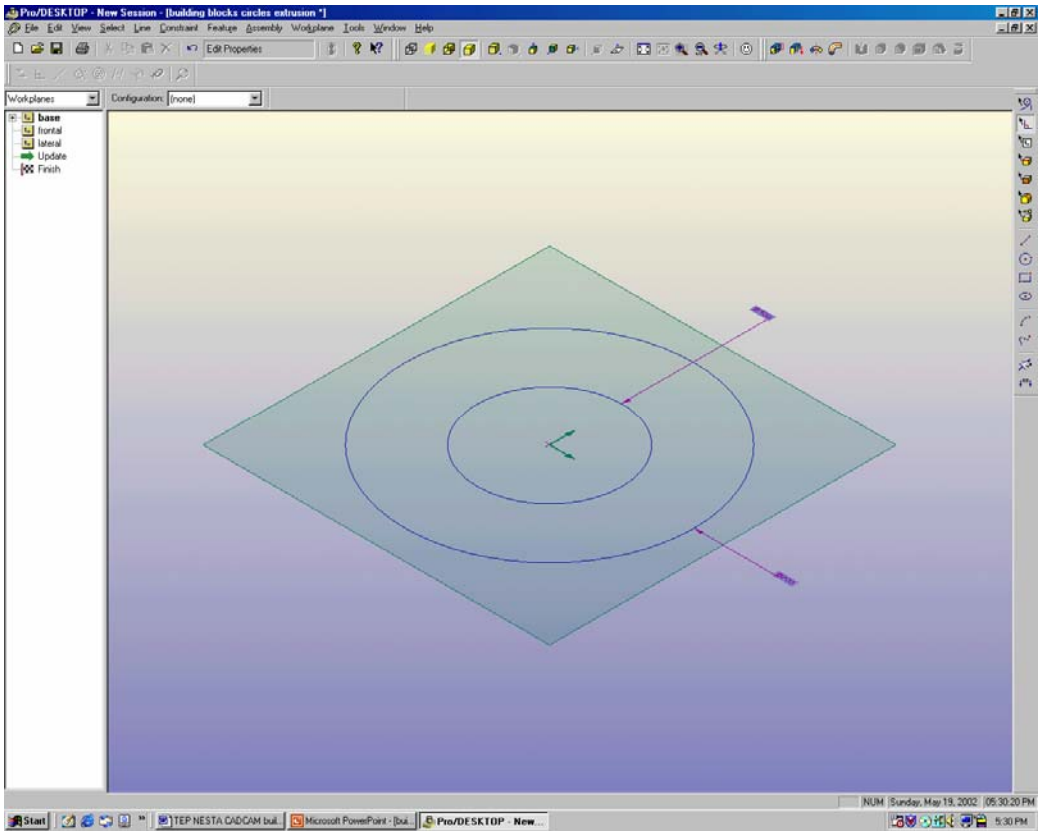


The diameter has changed to an exact value of 200 mm. This is the first example of creating two-dimensional shapes that are *Dimension Driven.*

In the next sequence you will use this circle to create the first feature in Pro/DESKTOP. The software has several methods of turning two-dimensional shapes into three-dimensional features. In this example you will use Extrusion.

The solid modeling procedure is best observed from an isometric viewpoint. Click on the Isometric icon in the Views toolbar or use the quick key shift I.

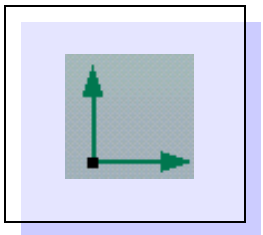


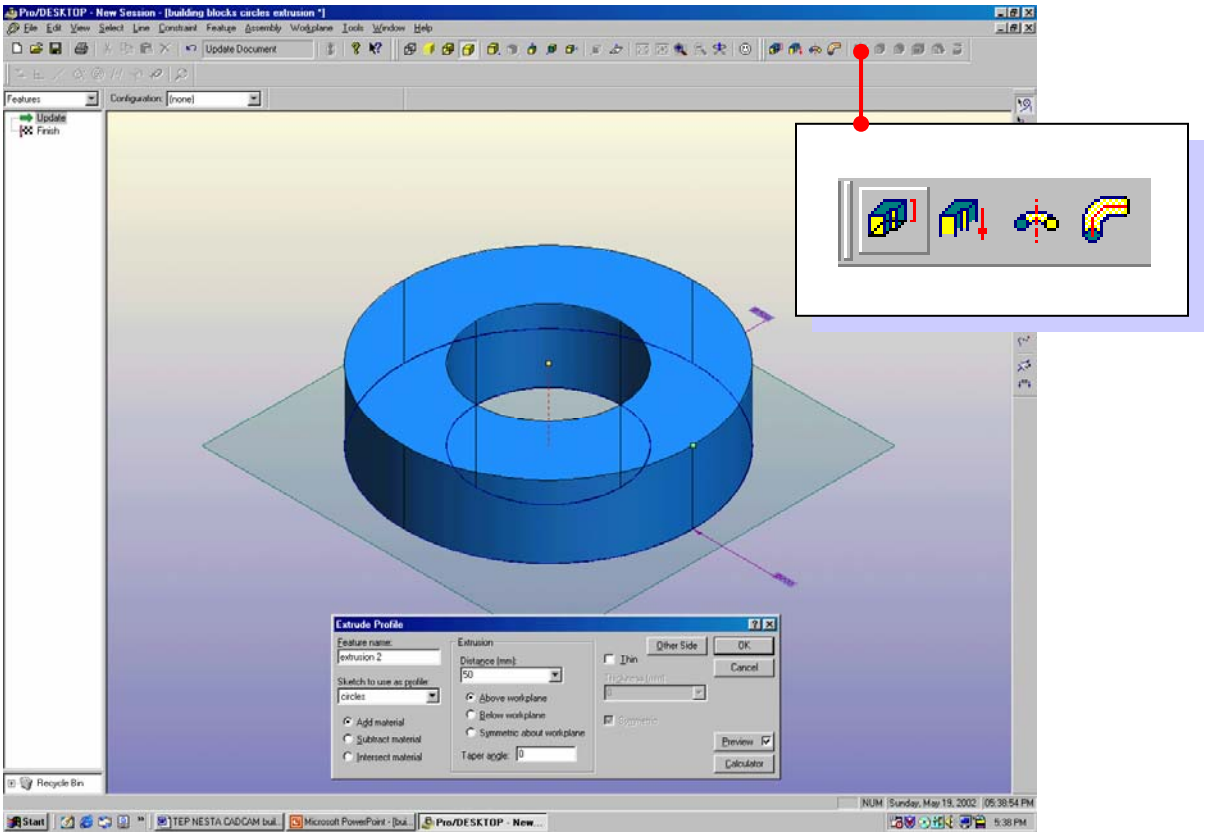


Practice your drawing skills by drawing a second concentric circle inside the original and set the diameter to 100 mm.

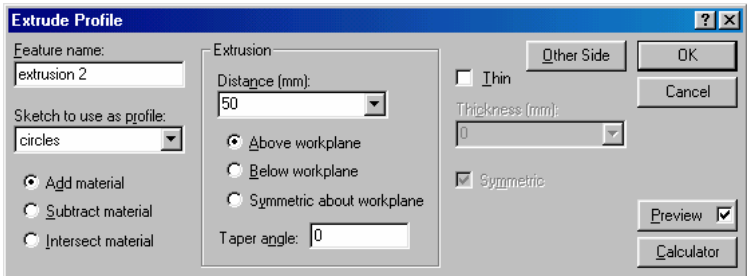
TIP:

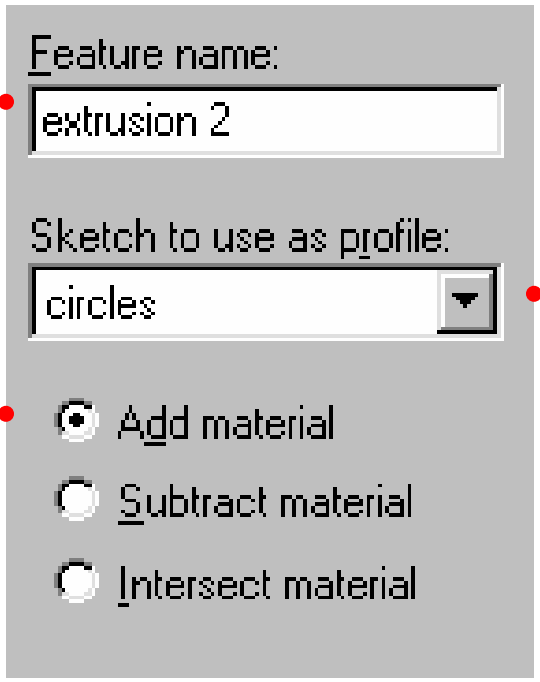
In order to ensure that the inner circle is concentric you can use a *snap* tool. The snap tool enables you to automatically draw from either the end, mid point, end point, intersection or center of a line or circle. Select the circle icon from the design toolbar and move the cursor to the circumference of the circle until it highlights. Carefully move the cursor very slightly inside the circumference until a black rectangle appears in the center of the circle. Whilst maintaining the black rectangle icon hold down the LMB and drag the mouse to draw a circle.





You are ready to create a three-dimensional feature from the two-dimensional circles. From the pull down menu select: Feature – Extrude Profile. An alternative is to select the extrusion icon as shown above. In the Extrude Profile box set the distance to 50 mm.

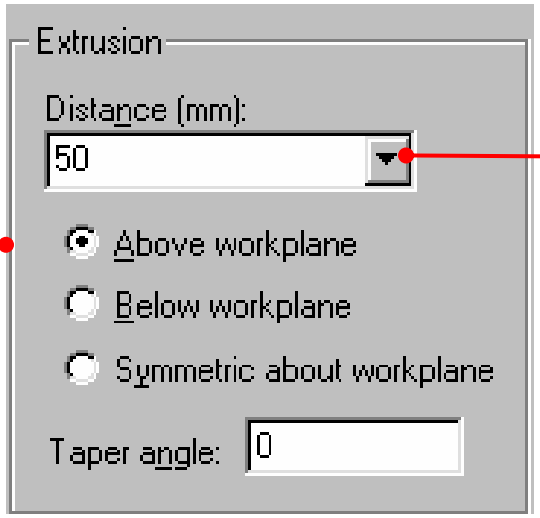




This window contains the name of the Feature. You can change this to reflect the description of your design.

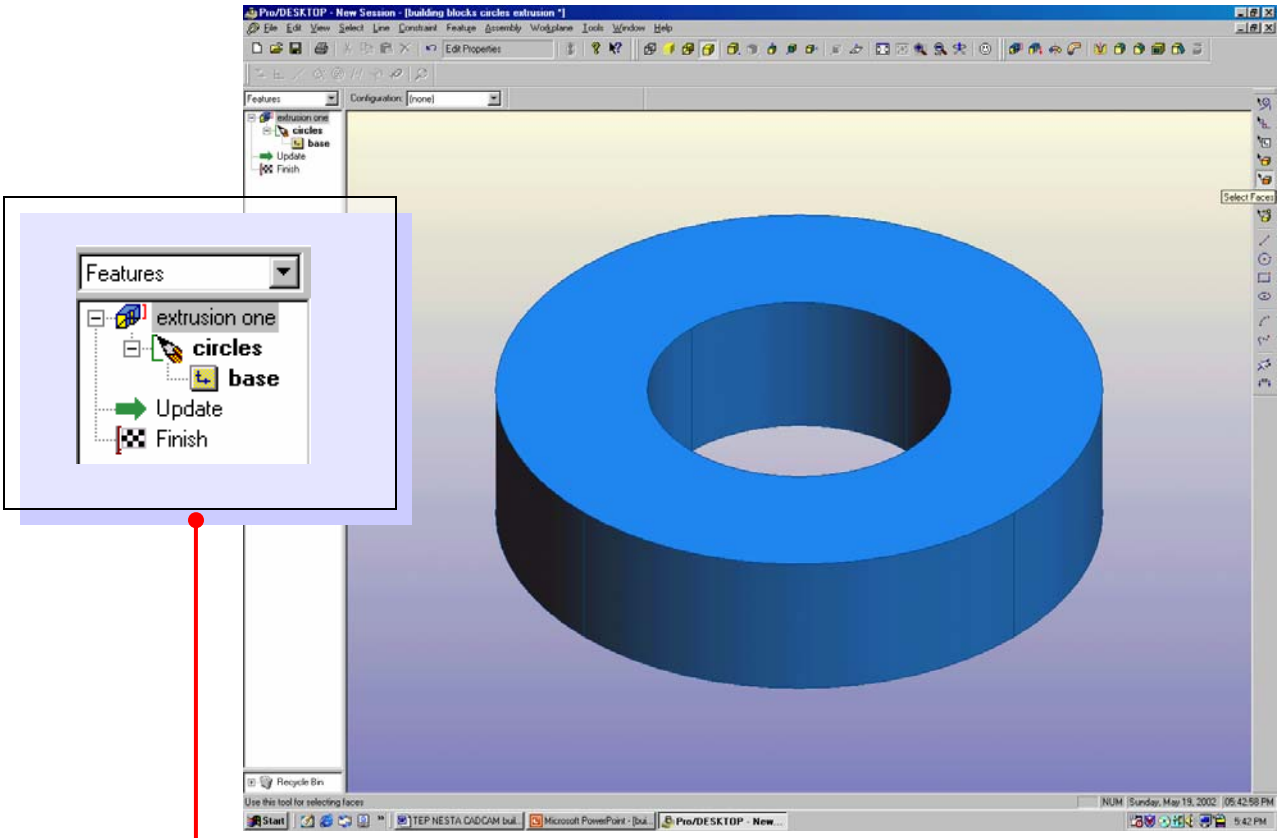
This button ensures the two-dimensional shape creates a solid form by adding material.

This window describes the name of the sketch you created earlier in the tutorial.



This button ensures the two-dimensional shape creates a solid form by adding material above the Workplane.

This window enables you to set the exact value the two-dimensional shape is extruded into a three-dimensional feature.



Move over to the browser at the top left hand side of the screen. Click on the black arrow to expose the browser options. Select Features as shown below.

- Click on the plus sign to reveal the name of the feature.
- Click on the next plus sign to reveal the name of the sketch.
- Click on the next plus sign to reveal the name of the Workplane.

You have created a simple solid model using feature based software. You used the technique of Extrusion to create the feature from two-dimensional shapes.