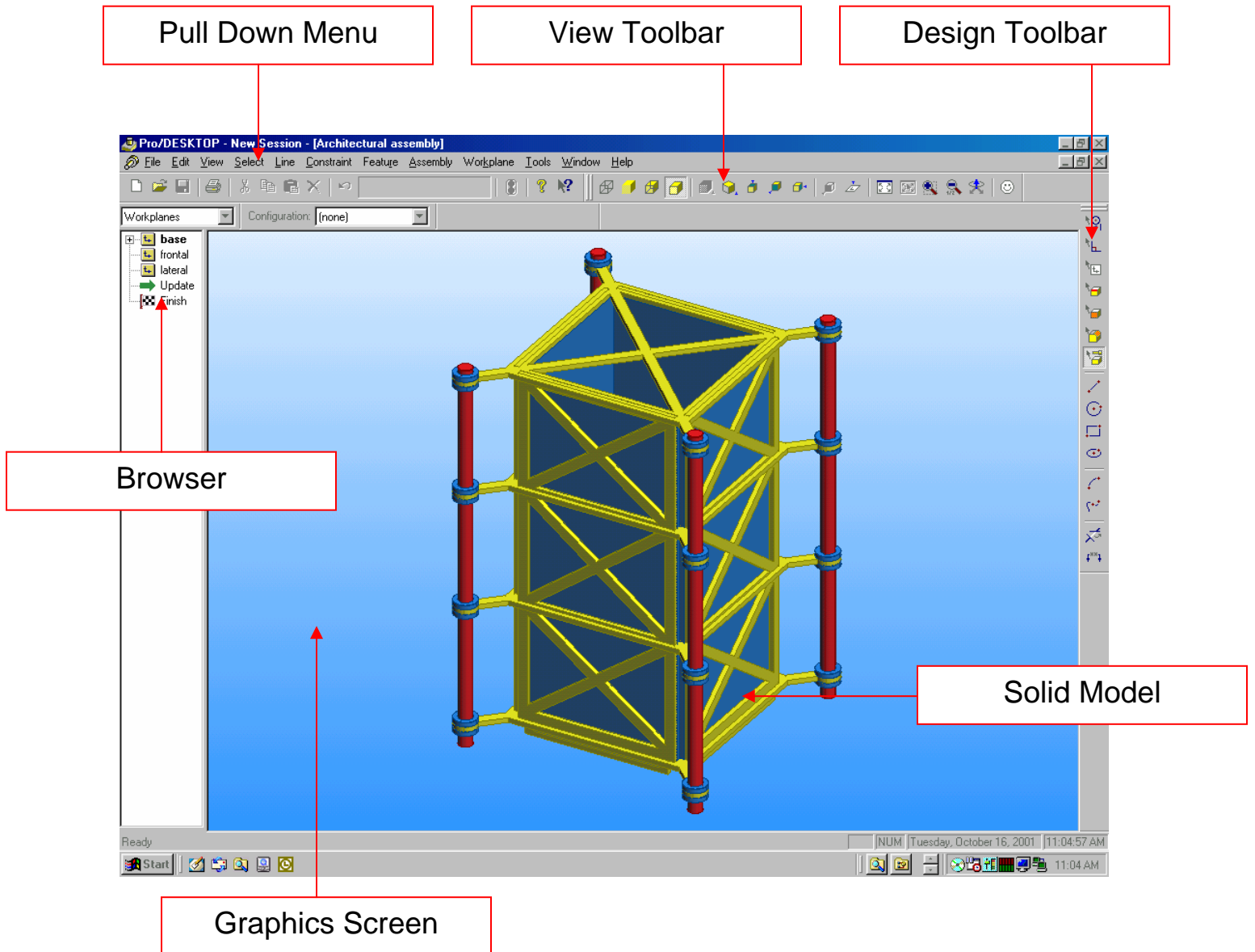Pro/DESKTOP Interface

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.

Pull Down Menu

View Toolbar

Design Toolbar

Browser

Solid Model

Graphics Screen

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 two:

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

Learning Outcomes:

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

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

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

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

1.10 Use the Extrude Profile feature creation tool to create a 3D-Object.
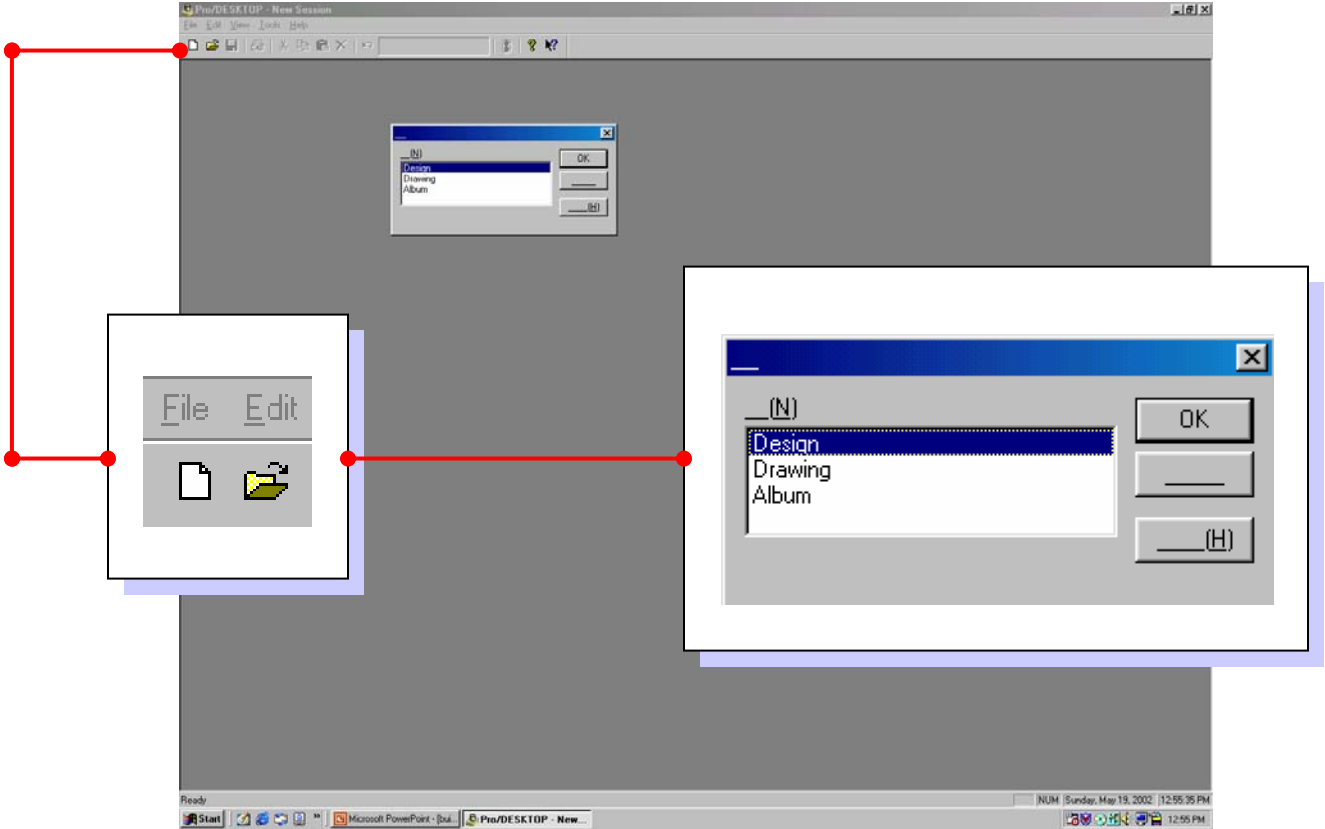
When you open the Pro/DESKTOP CAD software an interface appears on screen. There are three environments in Pro/DESKTOP that enable the user to concurrently create a solid model, an orthographic drawing and a photo-realistic album rendering. This exercise will guide you through creating a solid model in the Design environment.

From the Pull Down Menu select the white page icon.
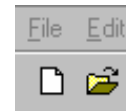
This icon allows you to open a new file in Pro/DESKTOP.

From the New File box select Design and click on OK.
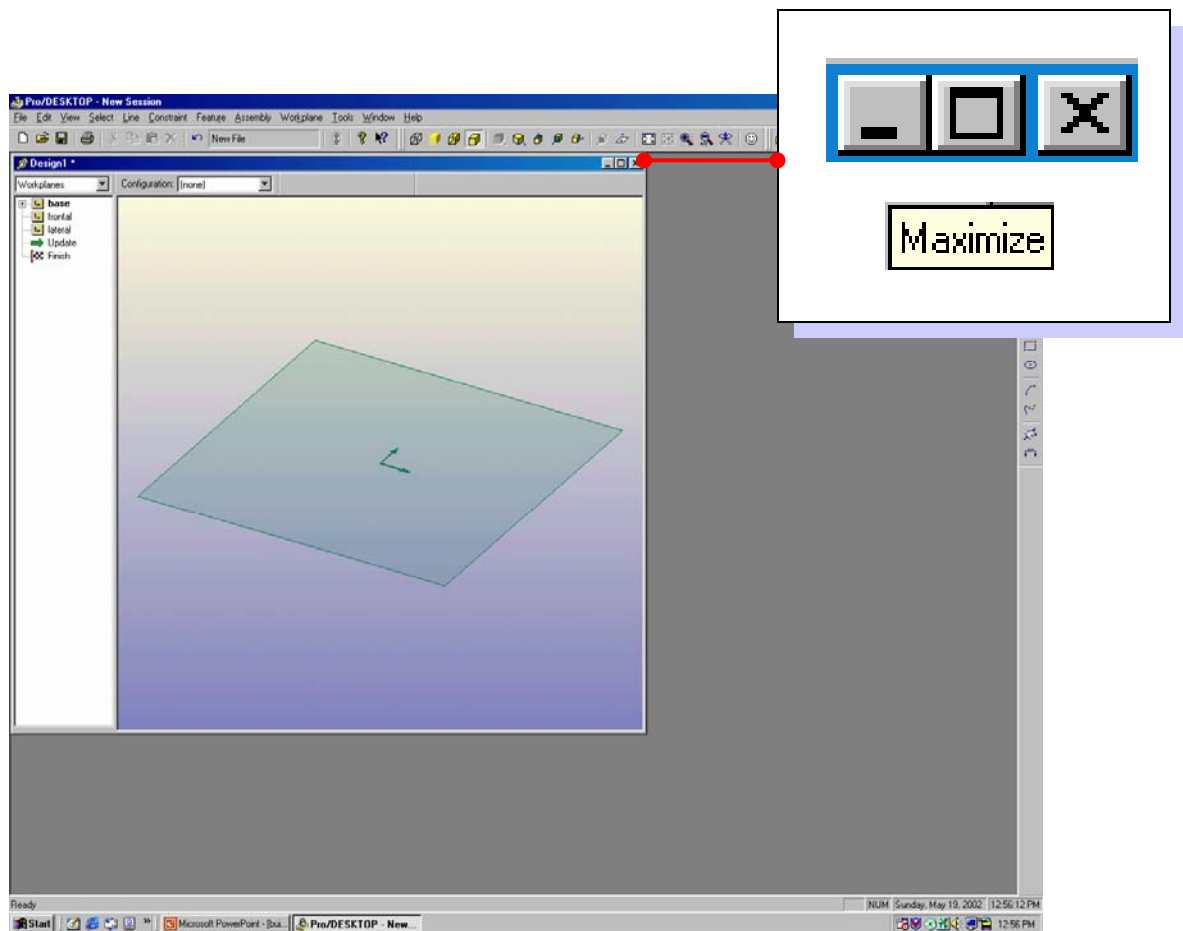
The Design interface appears on screen.

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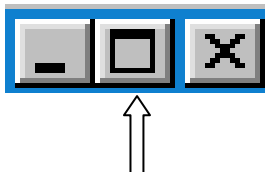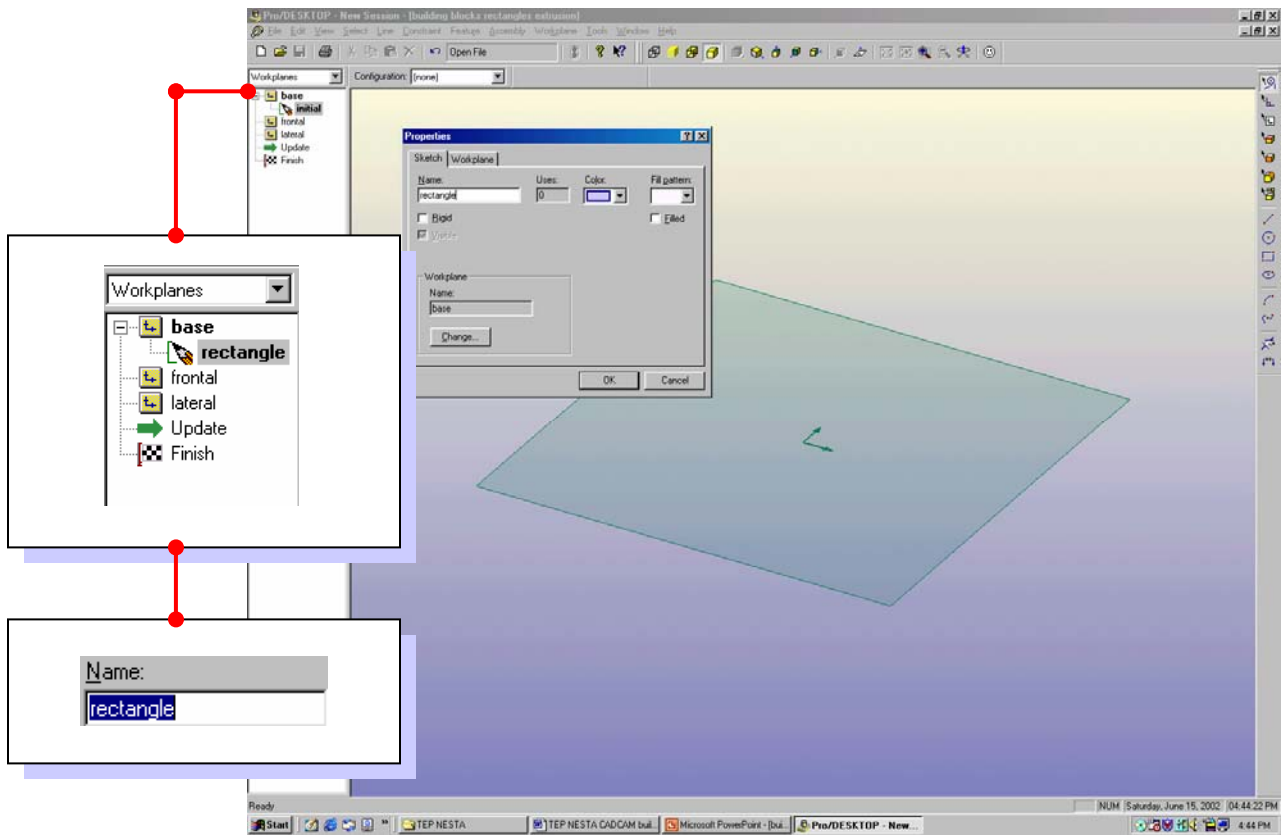
Double Left Click (DLC) on the default name for the Sketch titled initial in the Workplanes browser.

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 rectangle.

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.
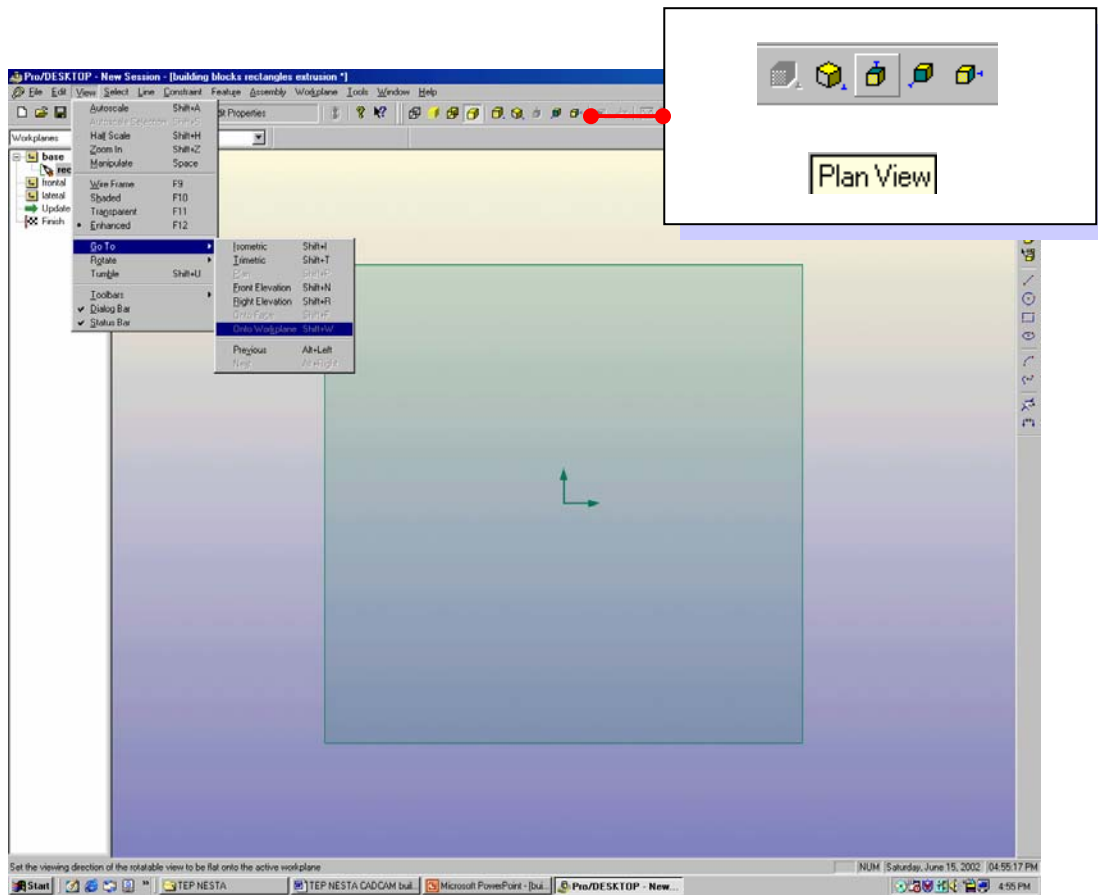
The Workplane can be repositioned to make drawing two-dimensional shapes easier.
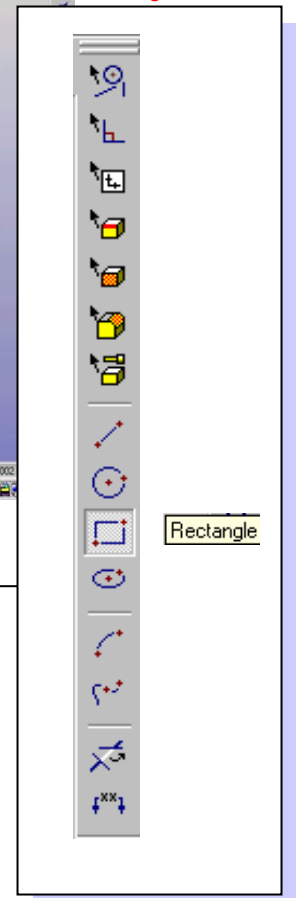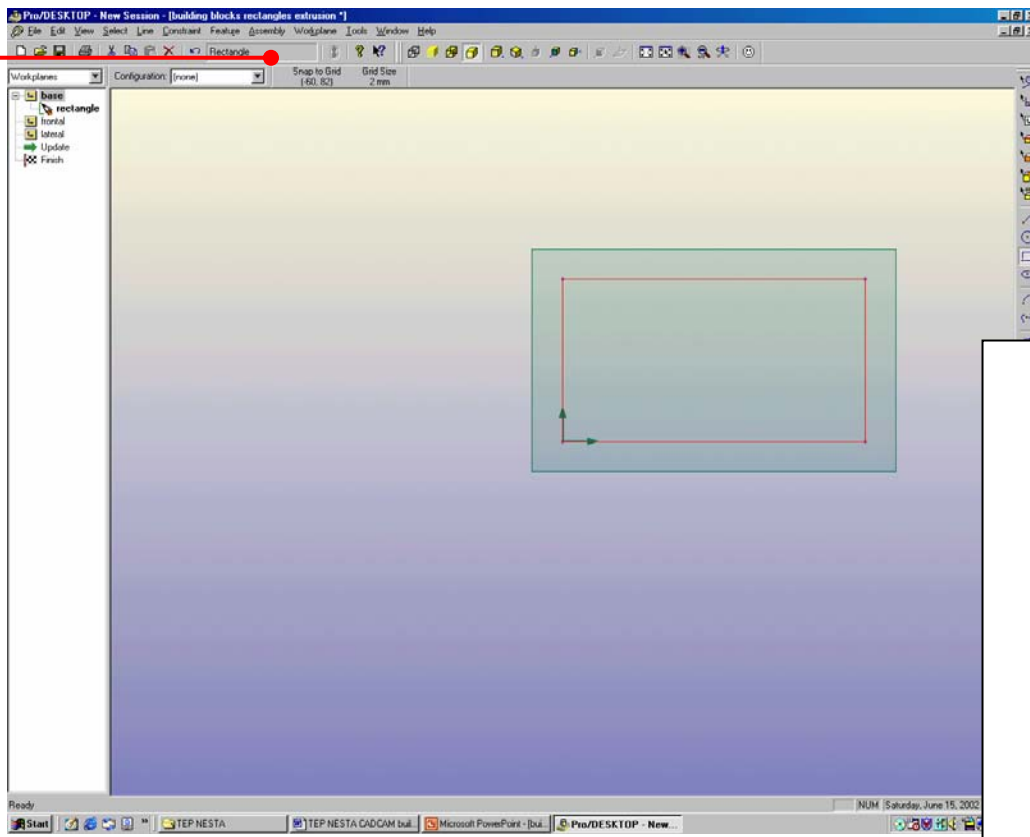
LC on the yellow cube icon with the top surface shaded in gray from the Views Toolbar

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 Rectangle 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 up to the right
to create a rectangle to the proportions shown in the snapshot.
Release the mouse to complete the rectangle command.
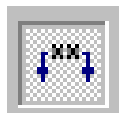
Rectangle

Autoscale selection
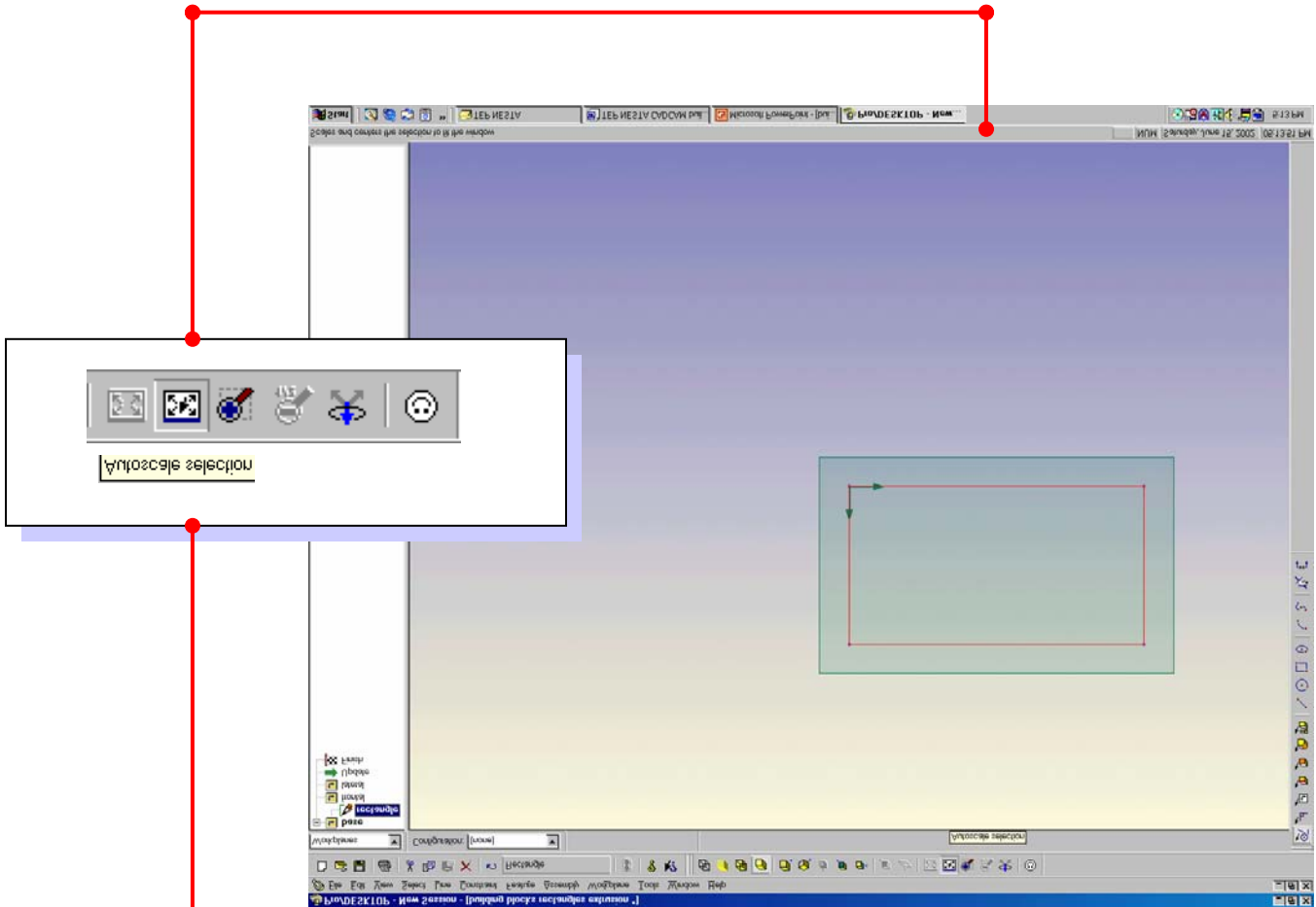
The next sequence will guide you through dimensioning the rectangle. Before we proceed it is important to reposition the rectangle into a convenient position in the graphic screen. Select the Autoscale selection icon n the Views toolbar from the top of the screen. The rectangle will be move to a central position. Select Halfscale selection to reduce the image by half. Tip: use the quick key selection of shift A then shift H to achieve the same result.

Autoscale selection

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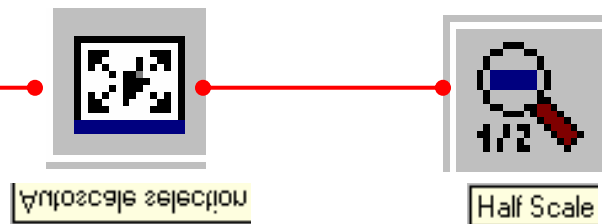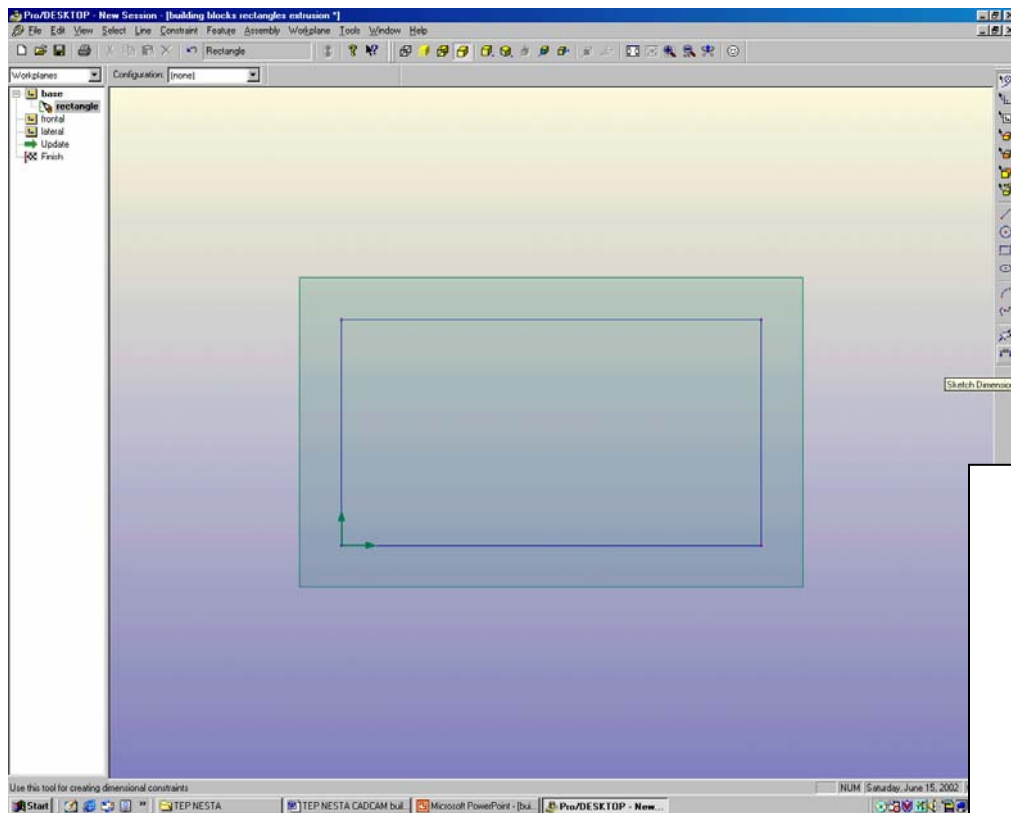
Pro/DESKTOP uses a Parametric system of dimensioning.
This means that the dimensions attached to the two-dimensional shapes not only inform the user of the size but they also control or drive the size. Dimension driven CAD technology offers the user a great deal of creative freedom.
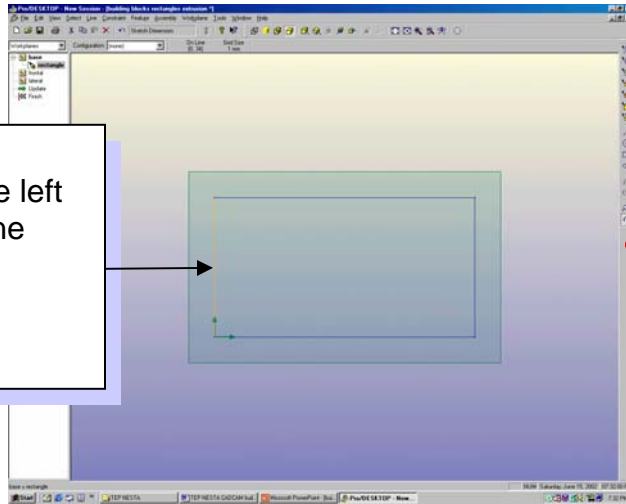
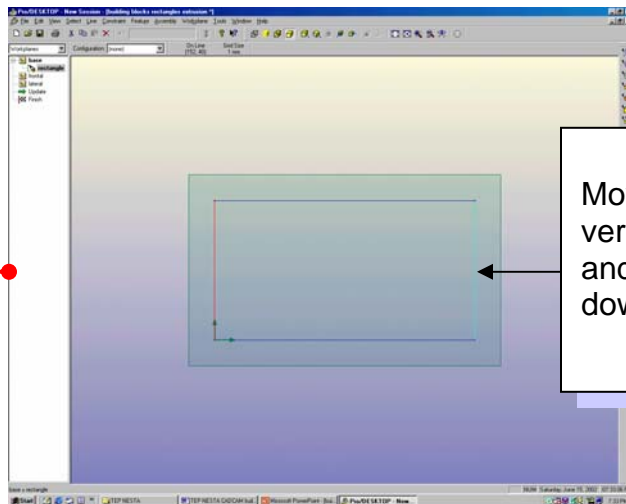Click on the Sketch Dimension icon in the Design Toolbar.

This tool will allow you to attach a linear dimension to the rectangle.
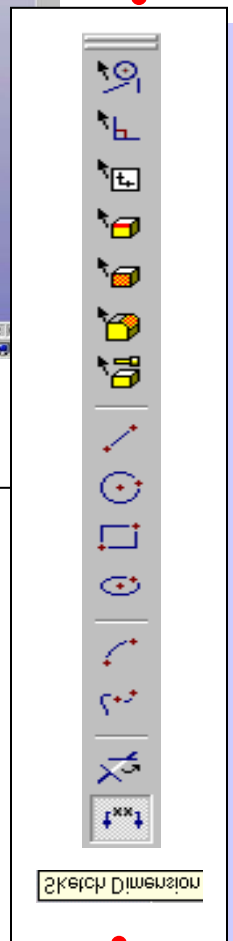
Move the cursor over to the left vertical line. Click on the line with the Left Mouse Button (LMB).
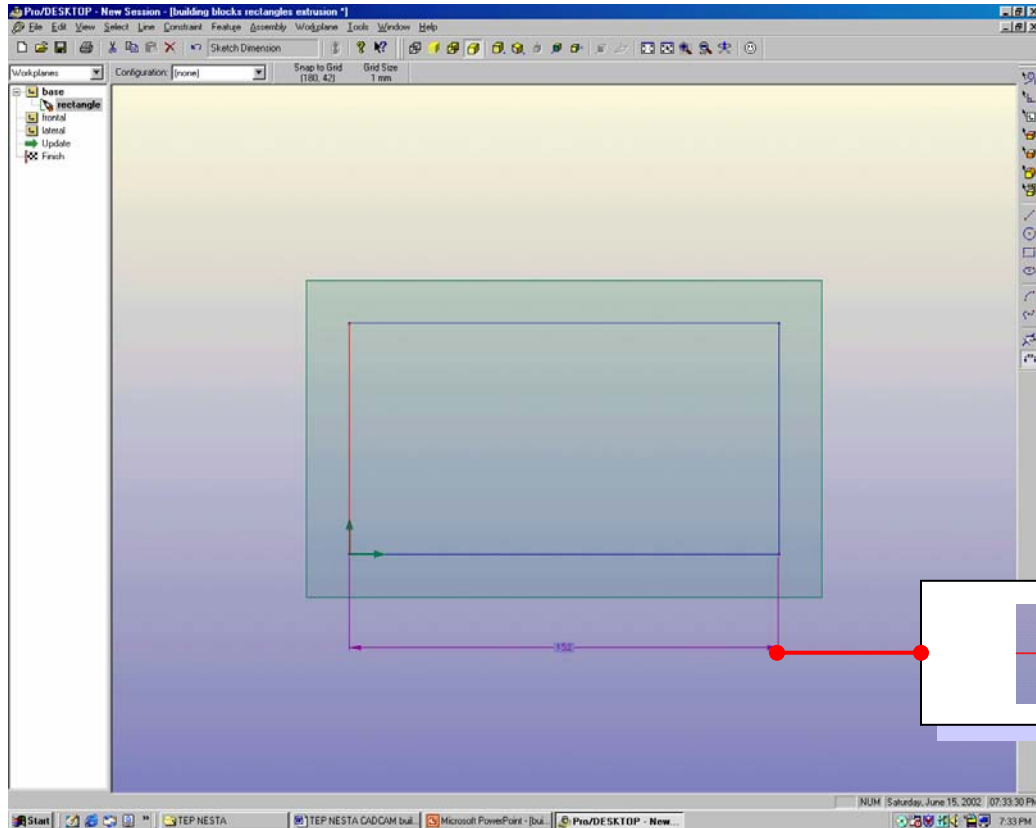
Move the cursor over the right vertical line, hold down the LMB and drag the mouse vertically downward.

A horizontal dimension constraint will appear that controls the distance between the two vertical lines.
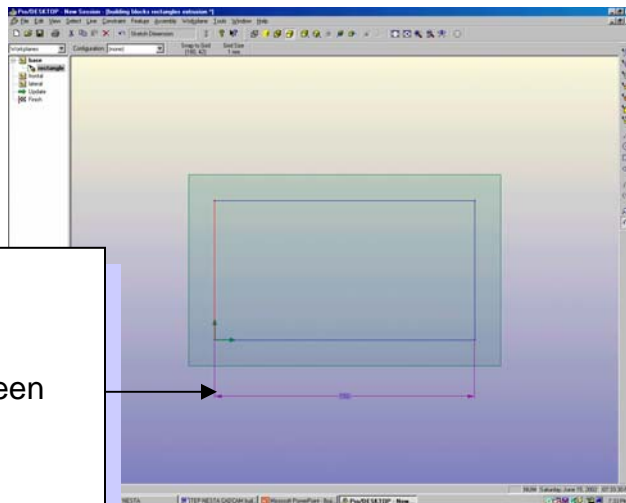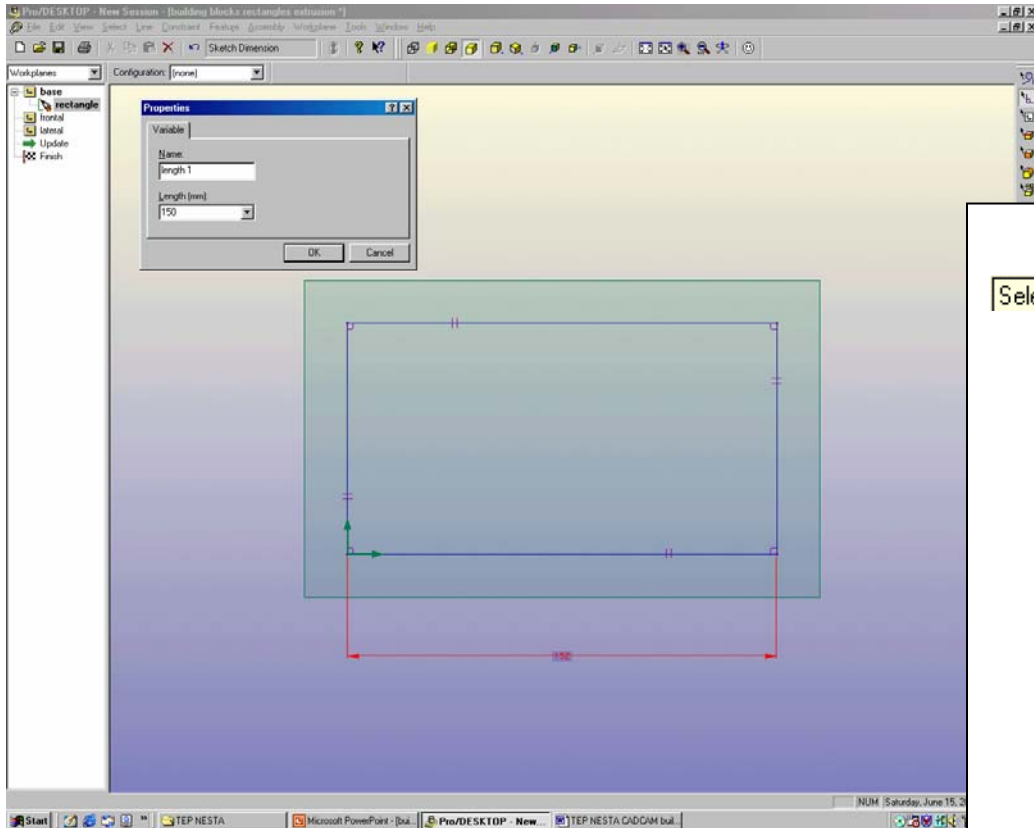
Earlier in this tutorial we described the software as *Dimension Driven.* The dimension that you have just attached to the rectangle will control its width and force the rectangle to an accurate value.

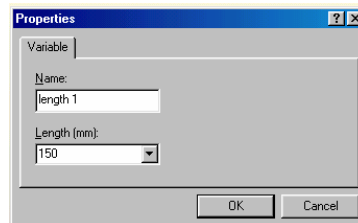In the next sequence you will use the Select Constraints tool to change the width to an exact value of 150 mm.
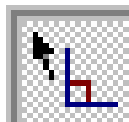
Click on the Select Constraints icon in the Design Toolbar
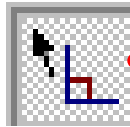
Move the cursor over the numbers of the dimension you have just attached and DLC to gain access to the Properties box.

Change the Length value to a diameter of 150 mm and click on OK to complete the command.
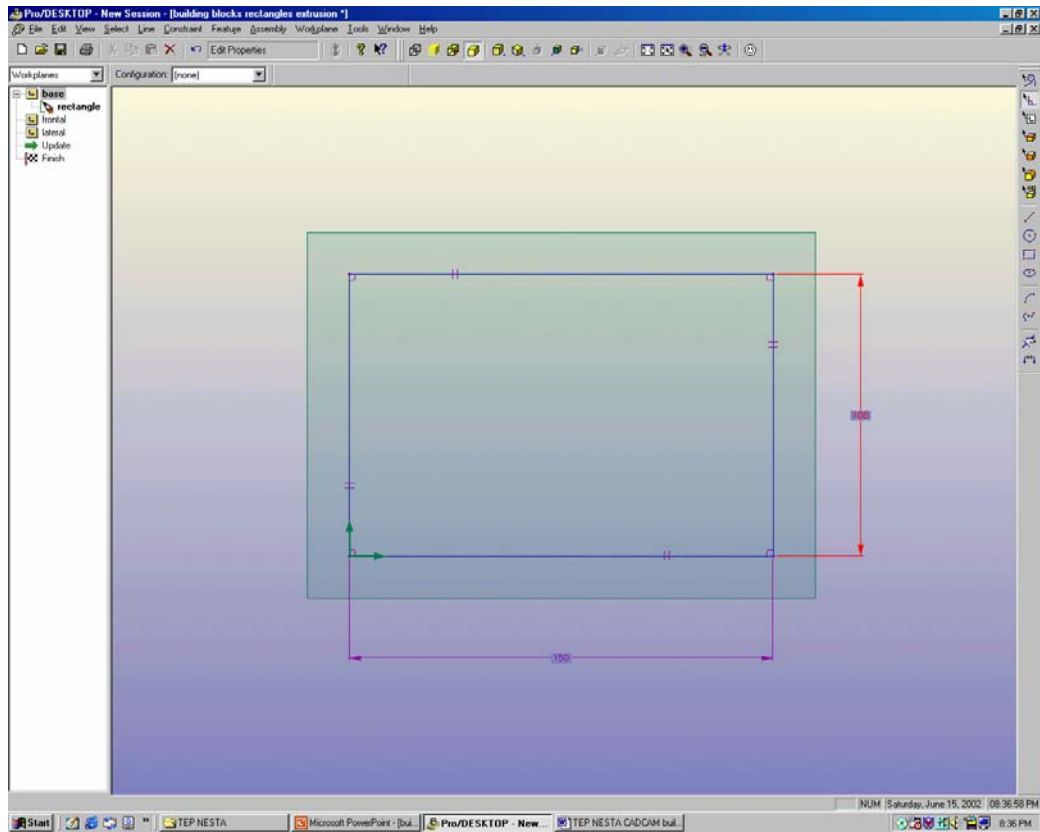
Repeat the previous exercise to create a vertical dimension line as shown in the snapshot.

Try and remember the following rules to help you create dimension constraints.

For a HORIZONTAL dimension constraint select opposing parallel VERTICAL lines.

For a VERTICAL dimension constraint select opposing parallel HORIZONTAL lines.

- Select Sketch Dimension icon from the Design Toolbar
- Left click on a line to select it
- Hold down the left mouse button on the opposite parallel line
- Drag the mouse away from the line to create a dimension constraint

View Isometric
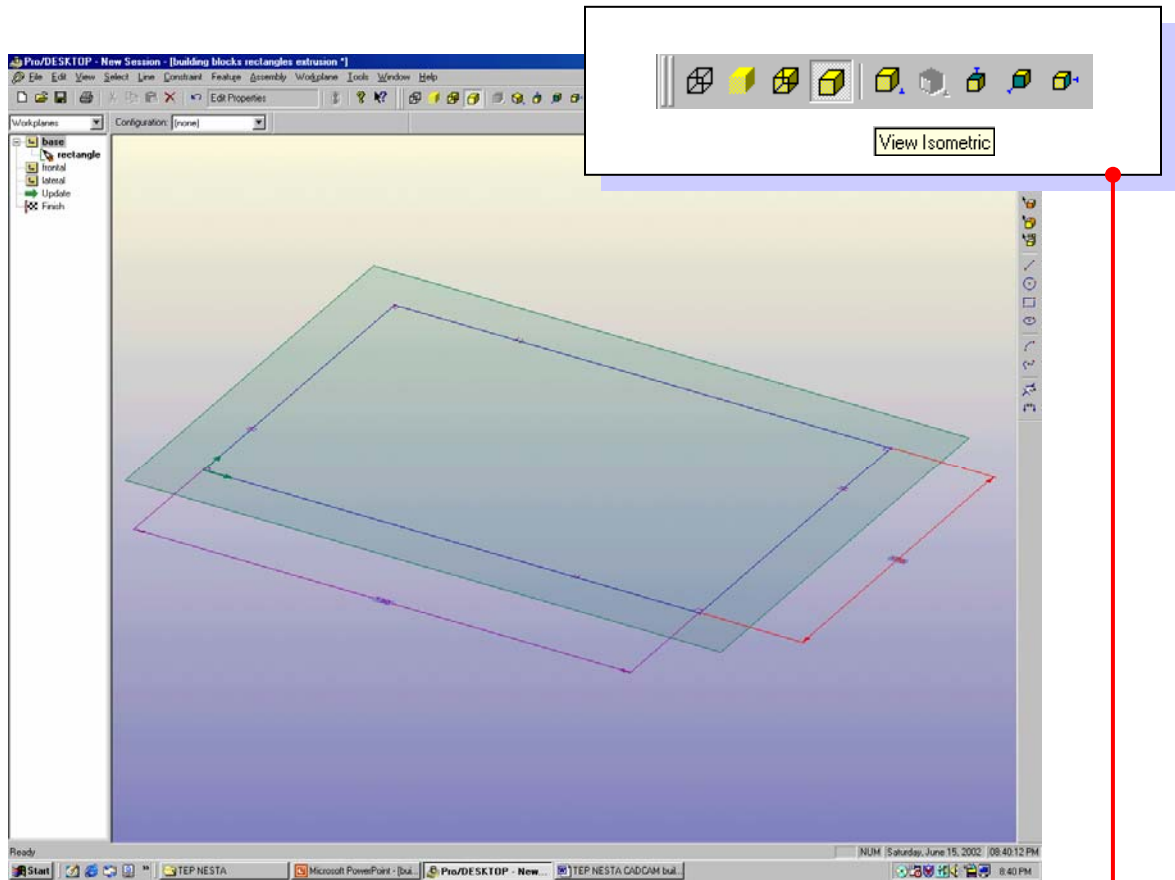
This is an example of creating two-dimensional shapes that are *Dimension Driven.*

In the next sequence you will use this rectangle to create a 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.
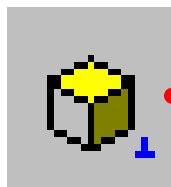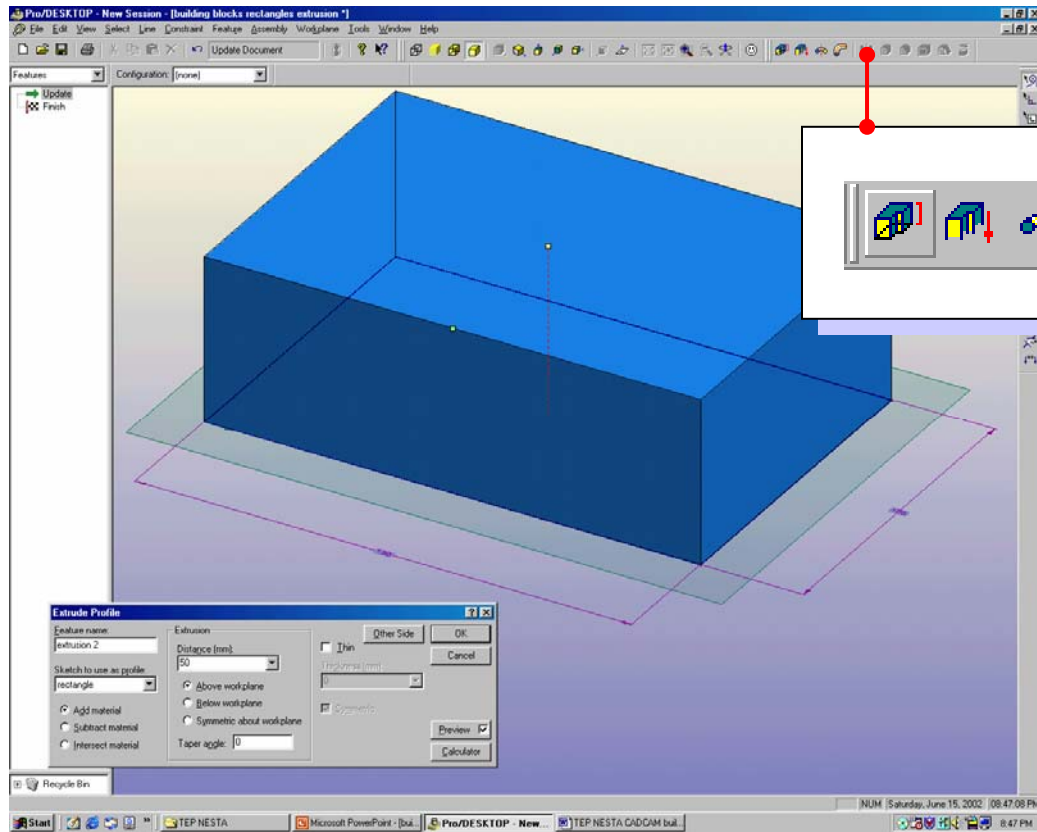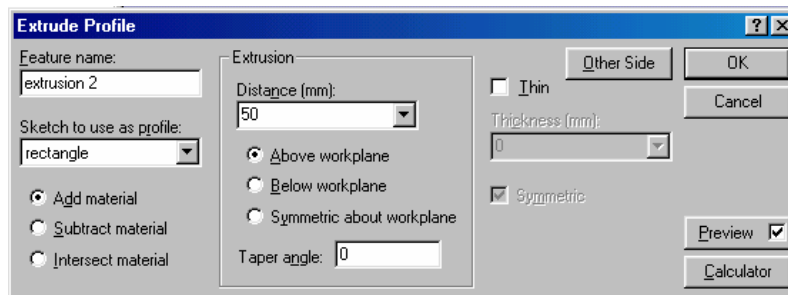
You are ready to create a three-dimensional feature from the two-dimensional rectangle. From the pull down menu select: Feature – Extrude Profile. An alternative is to select the extrusion icon. In the Extrude Profile box set the distance to 50 mm.

Extrude Profile

Feature name:

extrusion 2

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

Sketch to use as profile:

rectangle

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.

- ● Add material
- ○ Subtract material
- ○ Intersect material

Extrusion

Distance (mm):

50

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

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

- ● Above workplane
- ○ Below workplane
- ○ Symmetric about workplane

Taper angle: 0
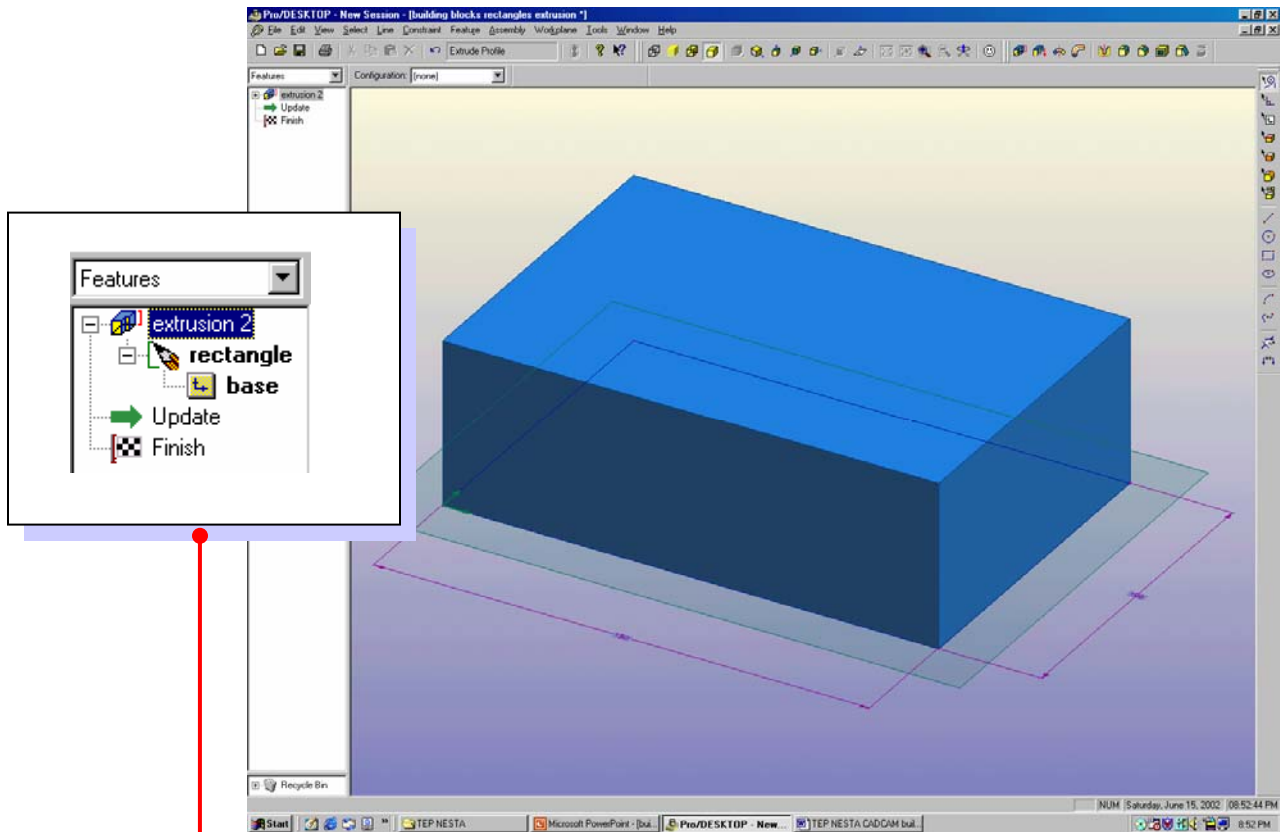
Move over to the browser at the top left hand side of the screen. Click on the black arrow to expose the browser options. Select Features as shown below.

- 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.