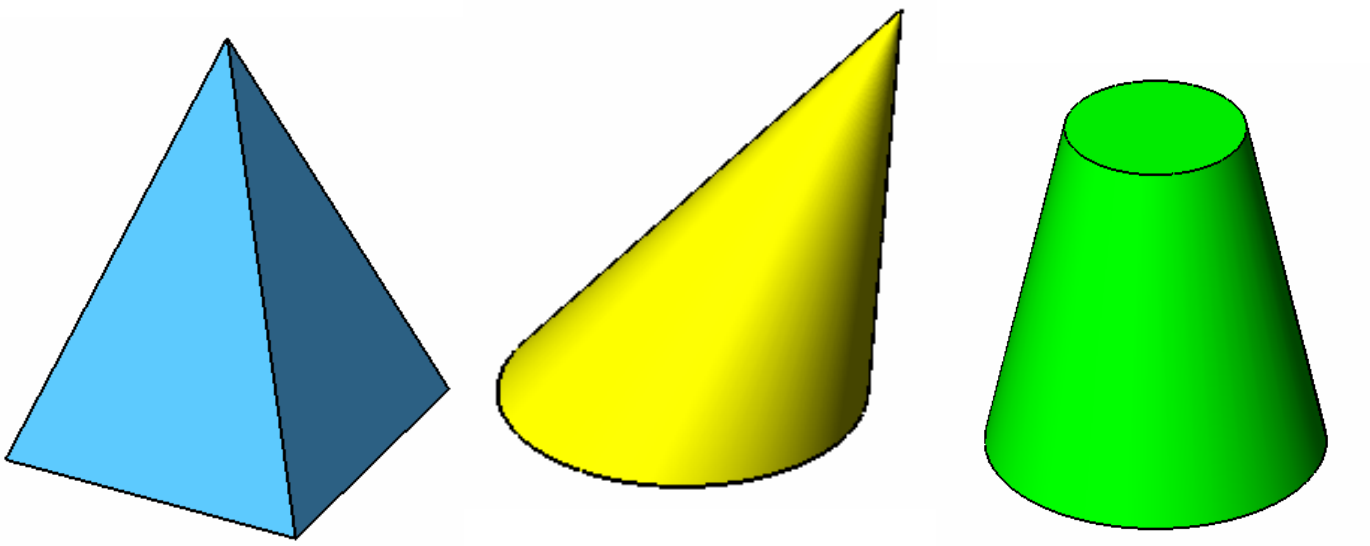
Lofted Solids Exercise



Prerequisite knowledge To complete this model you should have experience of the following: Sketching, Creating Planes, Creating drawings.

Focus of lesson This lesson will focus on using the following feature commands- **Lofting, Reference Planes**

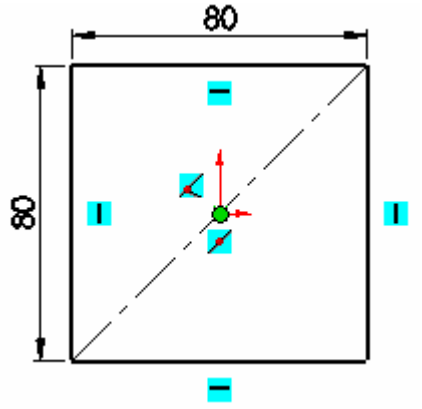
Commands Used This lesson includes Sketching**,** *Reference Geometry, Loft, Drawings*

Create new file

Create **new part file, Save As**, filename **Pyramid**.

Getting Started
Sketching the base

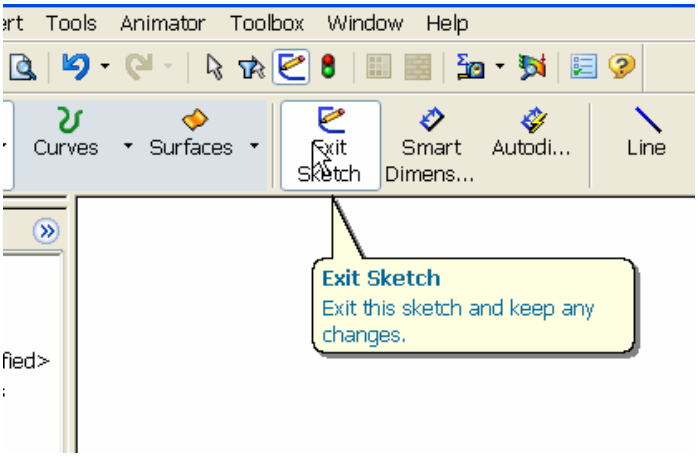
Create a sketch on the top plane using the dimensions shown.



Use the rectangle button 

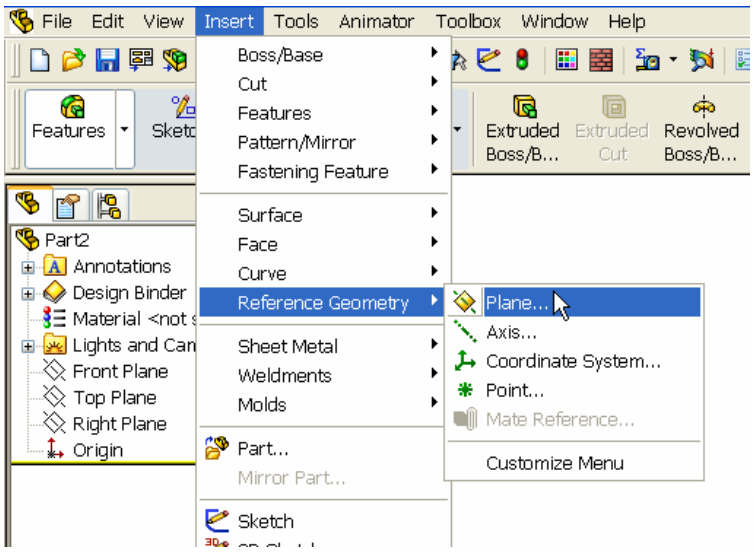
to draw a square of side 80mm.
Join the diagonals using a centre line.

Add a relation between the centre of the centre line and the origin to fuly define the sketch.
Approve the sketch.
Now **exit** the sketch.

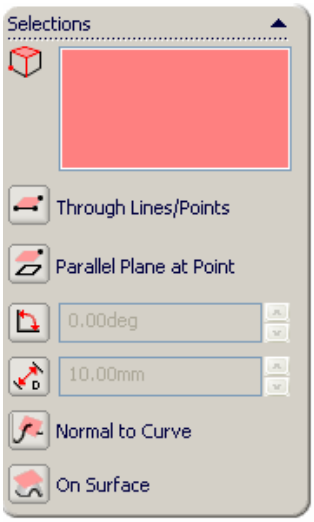


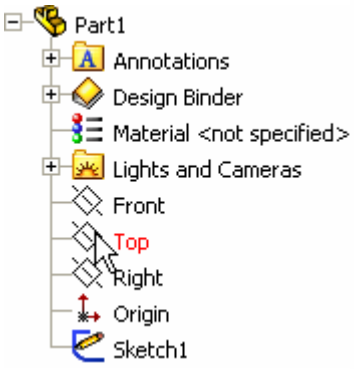
Creating a plane

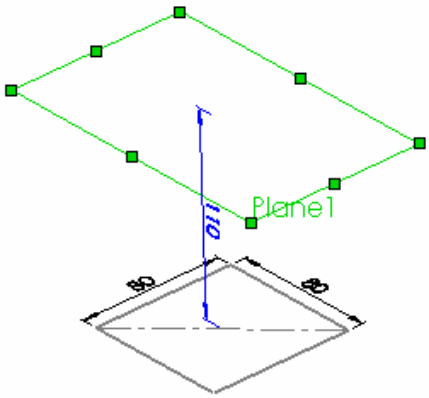
Insert a plane to draw the apex on. Go to insert/referencegeometry/plane.



Expand the design tree and select **the top plane.**
Set the distance of the plane to **110mm.**





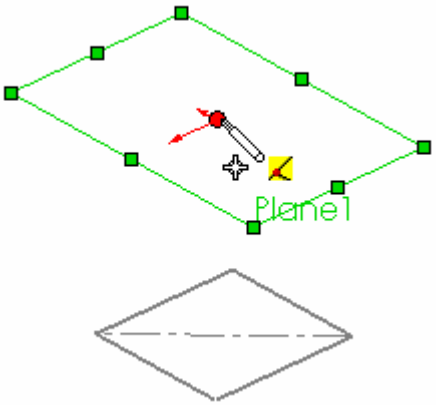


Creating the Apex

Open a sketch on the newly created plane.

Select the **Point tool** Point and draw a point concident with the origin.

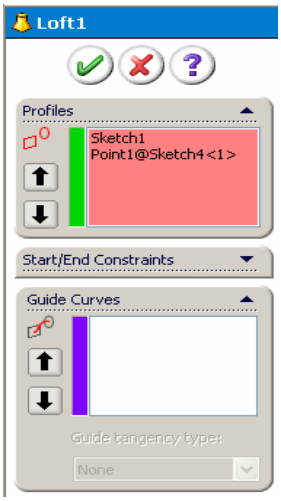
Approve this sketch to create the apex of our Pyramid.
Exit the sketch.

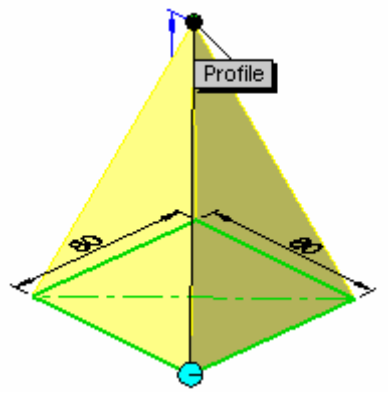


Lofting the Pyramid

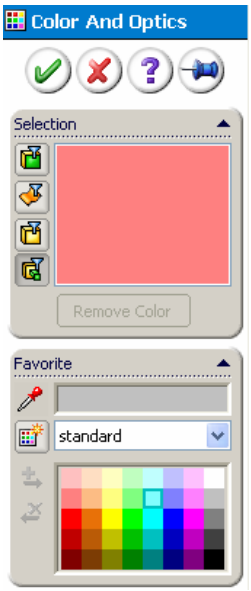
Select the features ribbon and select the

Lofted Boss/Base tool. Lofted Boss/Base
A window will appear on screen.
In the area marked profile select the two sketches to create the preview shown below.





Approve the feature to create the final part.



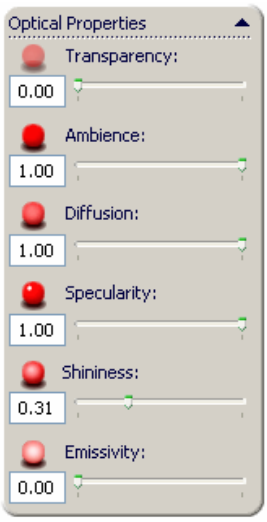
To add colour to the part select the **edit colour** button from the standard toolbar. 

A new window will appear.

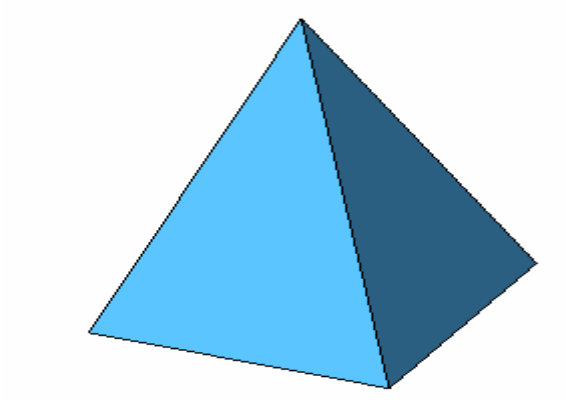
Select the object from the model and chose a colour from the set of standard colours available.

If you scroll down the edit colour window you will find the optical properties control window.

By dragging the respective slider bars you can control many of the objects various optical properties, such as transparency, shininess and ambience.



With this done,**Save** the file to finish the solid.

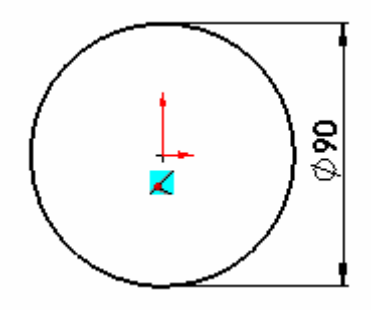


Save File

Create **new part file, Save As**, filename **Oblique Cone.**

Getting Started
The Oblique Cone

Create a sketch on the **top plane** using the dimensions shown. Use the **Circle** command placing the centre of the circle on the origin.



Approve the sketch to create the base of the oblique cone.
Exit the sketch.

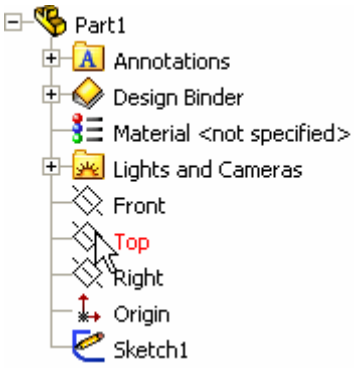
Creating a plane

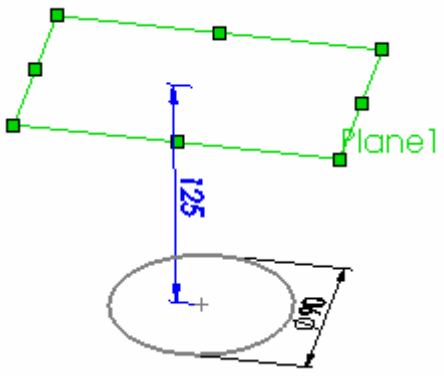
Open the features ribbon and select the feature **reference geometry**.

Select the plane option 

to enter the **create plane** window.

Expand the design tree and select **the top plane.**
Set the distance of the plane to **125mm.**





The preview shown on previous page should appear.

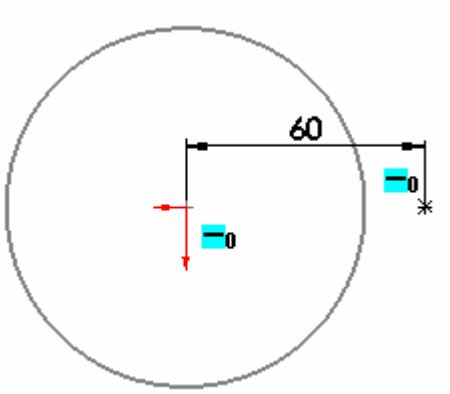
Approve the feature  to create reference plane.

Sketching the apex

Open a sketch on the newly created plane.

Select the **Point tool** 

and draw a point. Create a horizontal relation between the poin and the origin. Set the distance between the two to **60mm.**



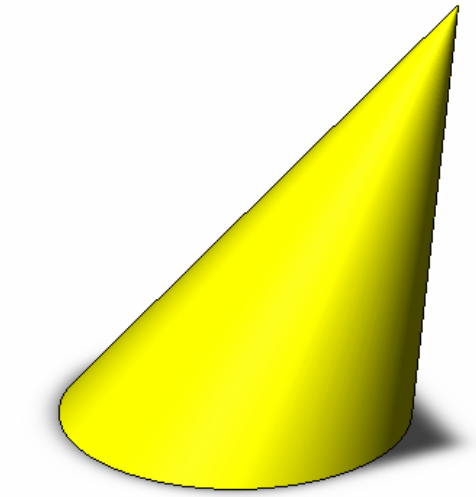
Approve this sketch to create the apex of our Oblique Cone. **Exit** the sketch.

Lofting the Oblique Cone

Select the features ribbon and select the 

Lofted Boss/Base command.

A window will appear on screen.
In the area marked **profile,**select the two sketches.
To create the solid shown below,
add a colour of your choice using the same steps shown previously.



Save the file

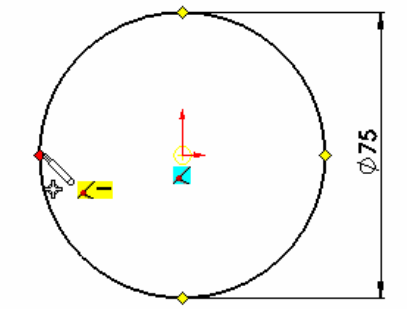
Save the file to complete the Oblique Cone.

Create new part

Create **new part file, Save As**, filename **Truncated Cone.**

Getting Started: The Truncated Cone

Create a sketch on the **top plane** using the dimensions shown. Use the **Circle** command placing the centre of the circle on the origin.



Create a **point** on the left quadrant of the circle N.B as you browse the cursor over the quadrant a diamond symbol will appear to indicate. the quadrant . Place the **point** on this symbol (it will be used later during the loft phase).

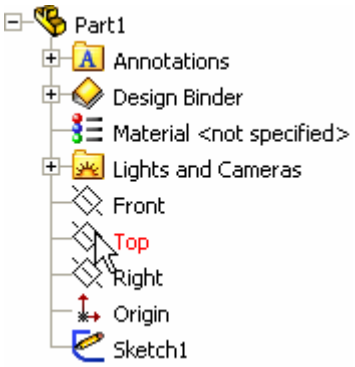
Approve the sketch to create the base of the truncated cone. Exit the sketch.

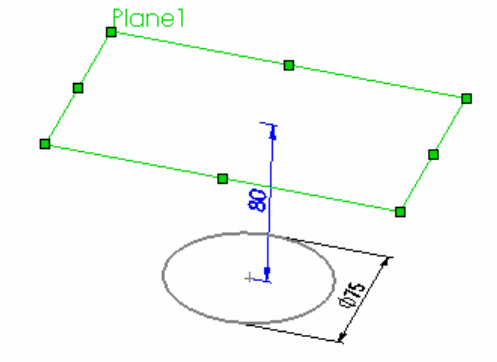
Creating a plane

Open the features ribbon and select the feature **Reference Geometry**.

Select the plane option 

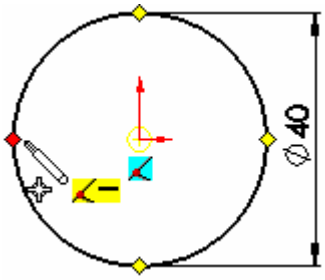
to enter the **create plane** window.
Expand the design tree and select the **top plane.**
Set the distance of the plane to **80mm.**





Creating the second sketch

Open a sketch on the newly created plane. Draw a circle of diameter 40mm with its centre on the origin.



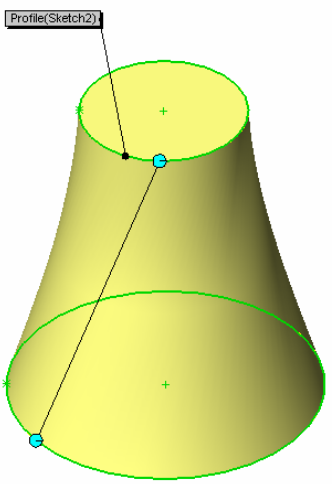
As with the first sketch, create a point on the left quadrant of the circle.

Approve the sketch and exit.

Lofting the Truncated Cone

Select the features ribbon and select the **Lofted Boss/Base** command.

A window will appear on screen.
In the area marked **profile** select both sketches to create the preview shown opposite.

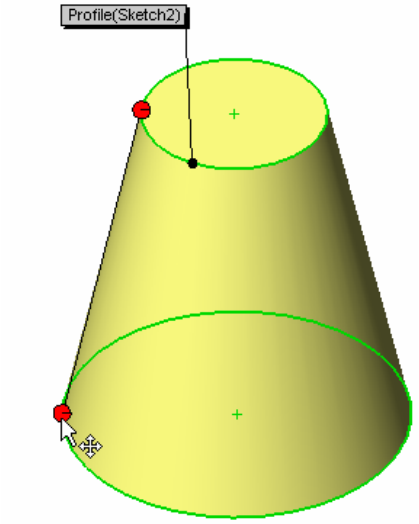


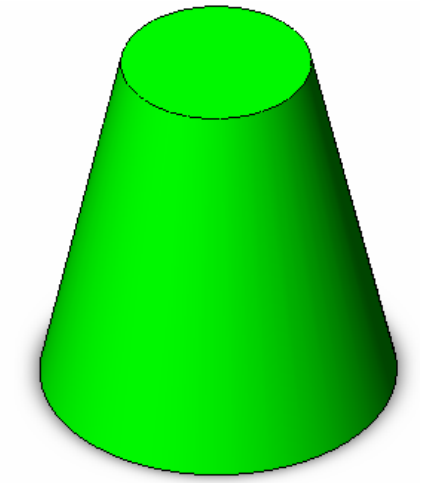
As can be seen, the loft does not automatically assume the shape to be conical.

This feature can be used to create hyperboloids of revolution.

The loft can be made conical by dragging each end of the guide generator line onto the pair of points created within each sketch.
This ensures the loft is conical in shape.

Approve the shape and save the truncated cone.





Creating the drawing

Open a **New drawing** file and save the part under the filename **Truncated Cone**.

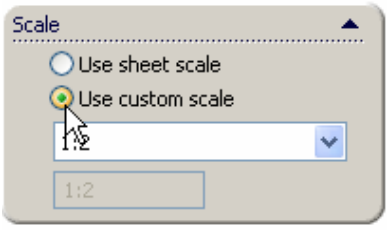
Creating the drawing

Select the **drawing ribbon** 

Select the **standard 3 views** option. 

Double click on the **Pyramid** option in the window that appears.
This will bring in the plan, elevation and end view for the pyramid.

Left click over the front elevation.
This brings up the drawing view window.
Tick the use custom scale option to activate the scale toolbox.



Select the dropdown arrow to view the scale options.
Select the **scale 1:1** and move the views into position on the drawing sheet.

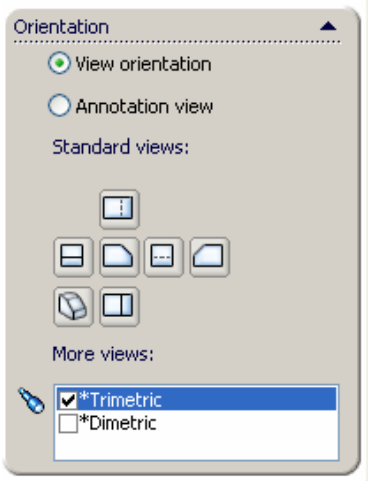
Select the M**odel View** option 
.

Again, double click the **Pyramid** option.

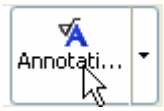
In the **view orientation** window that appears select the option **Trimetric.**

As with the previous views tick the custom scale option and select the scale 1:1

Drag the cursor into the drawing sheet and place the drawing into position by left clicking.



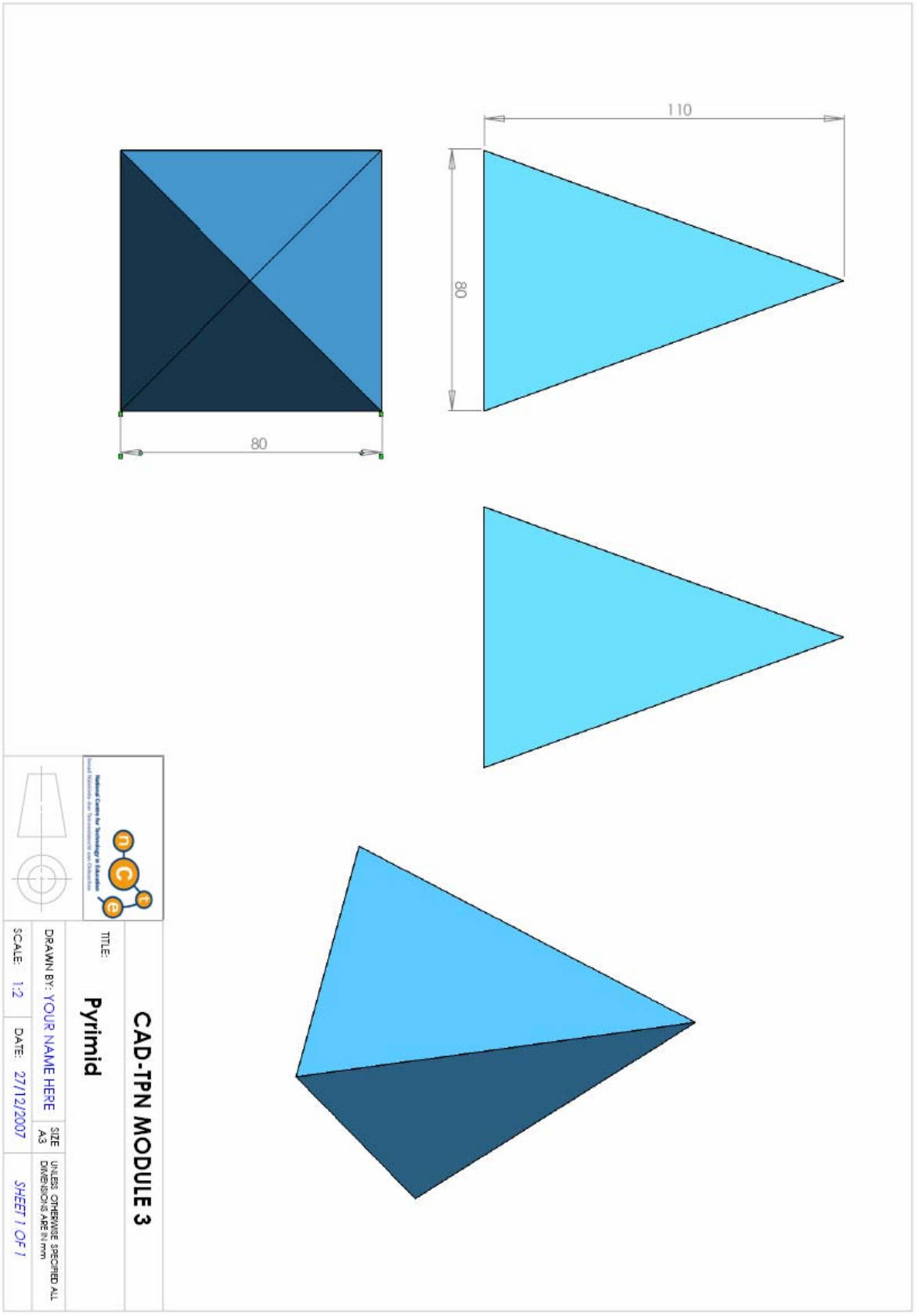
Dimensioning the drawing

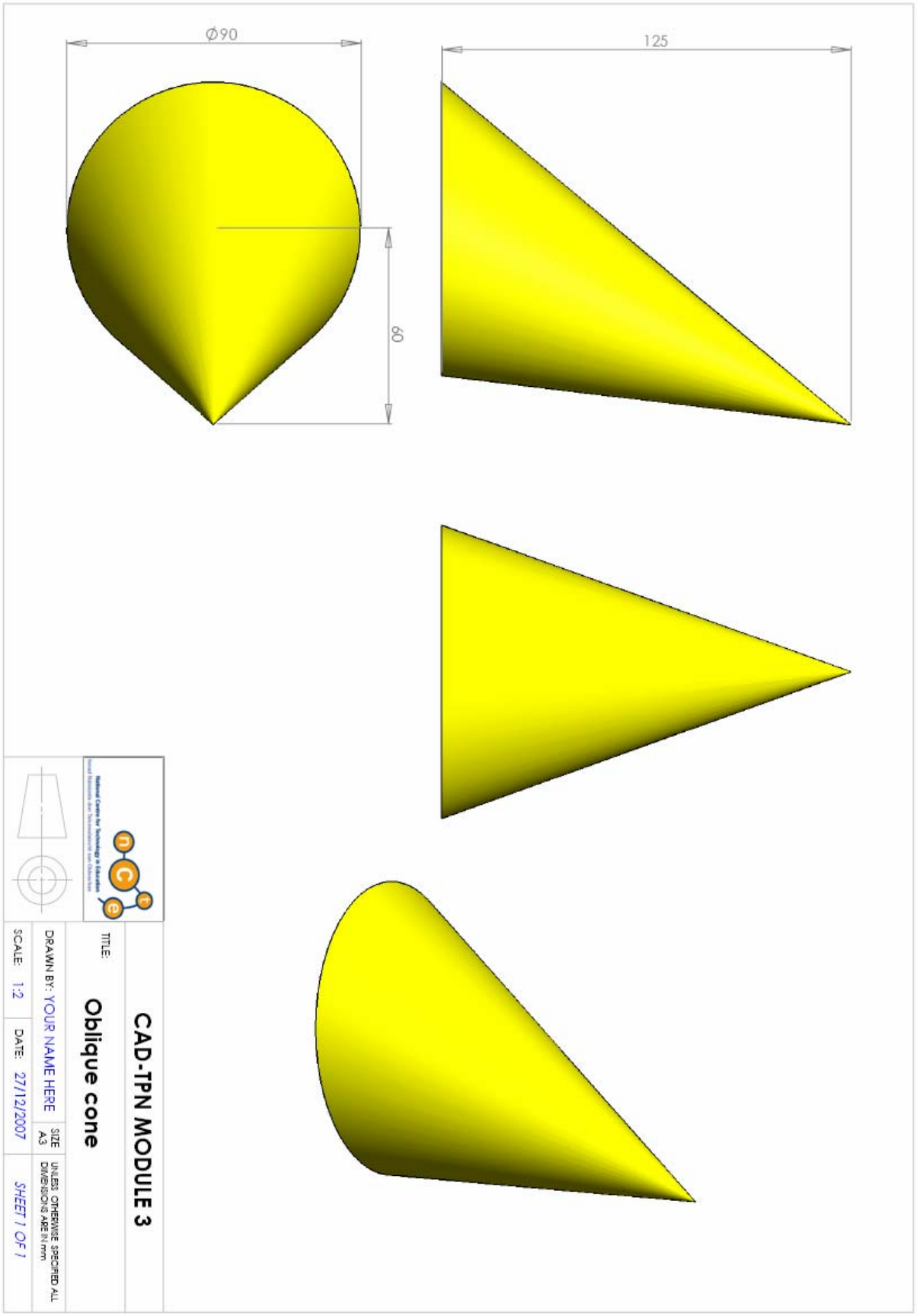
To dimension the drawing, first select the annotations ribbon to bring up the dimensioning button. 

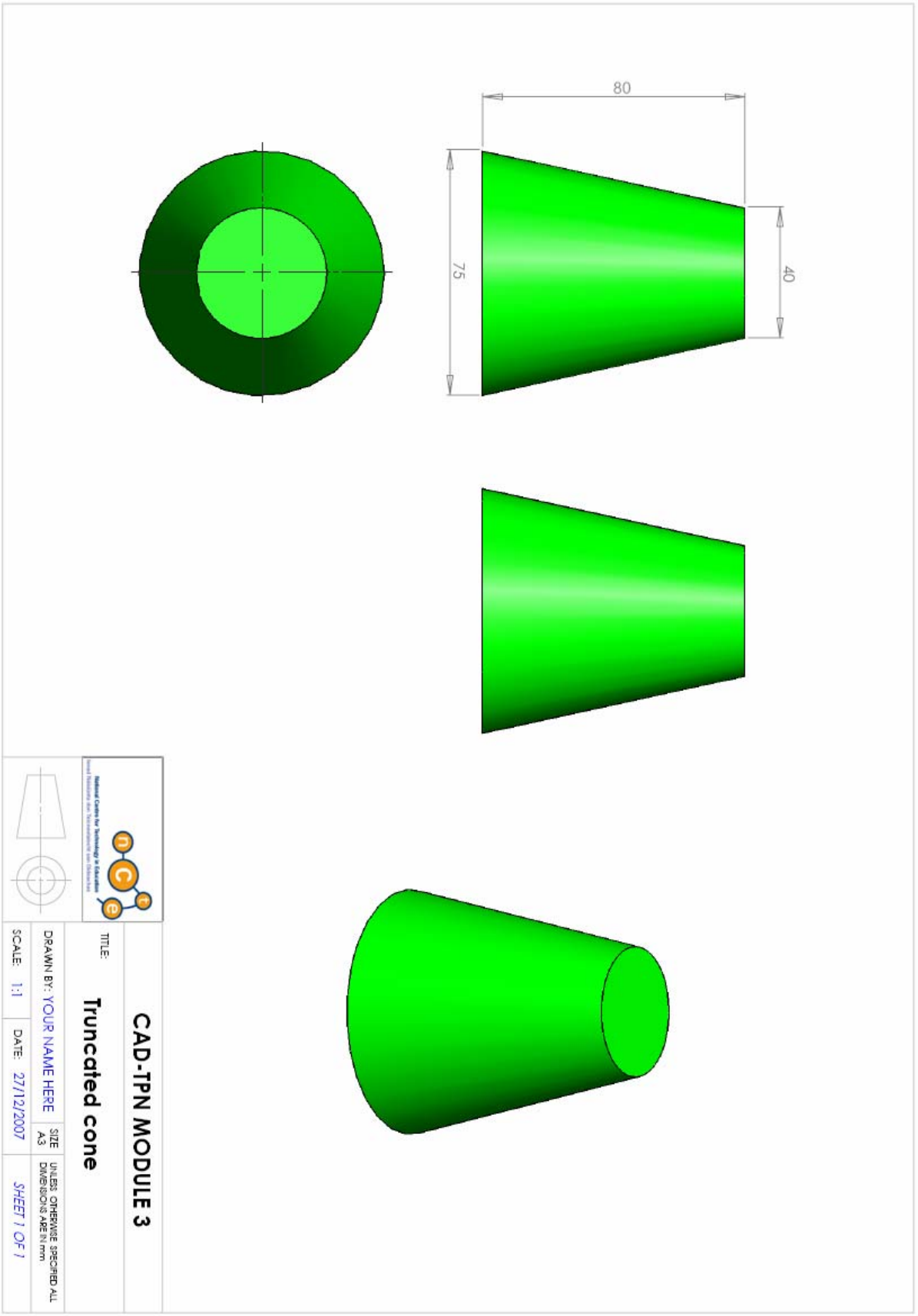
Select the **Smart Dimension** button 

and select the points which are to be dimensioned.
Pull out the dimensions into an acceptable position and left click to enter.
Repeat this process for all relevant dimensions.

Save the file and repeat for each of the solids modelled in this exercise.


110
80
80
CAD-TPN MODULE 3
TITLE:
Pyrimid
DRAWN BY: YOUR NAME HERE
SIZE
A3
DATE: 27/12/2007
SCALE: 1:2
UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm
SHEET 1 OF 1


Ø90
125
60
CAD-TPN MODULE 3
TITLE:
Oblique cone
DRAWN BY: YOUR NAME HERE
SIZE A3
SCALE: 1:2
DATE: 27/12/2007
UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm
SHEET 1 OF 1


80
75
40
CAD-TPN MODULE 3
TITLE:
Truncated cone
DRAWN BY: YOUR NAME HERE
SIZE A3
UNLESS OTHERWISE SPECIFIED ALL DIMENSIONS ARE IN mm
SCALE: 1:1
DATE: 27/12/2007
SHEET 1 OF 1