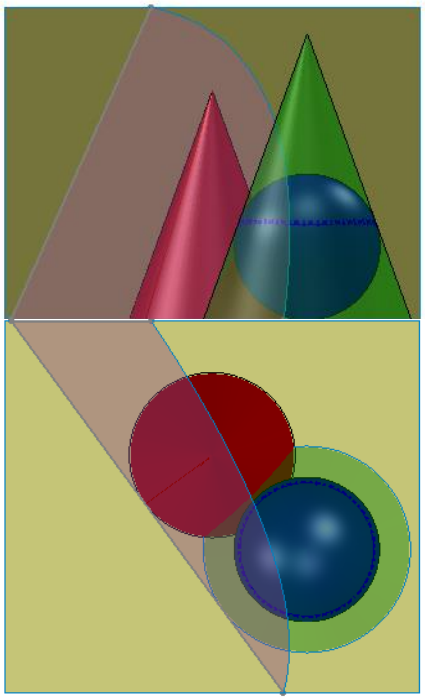
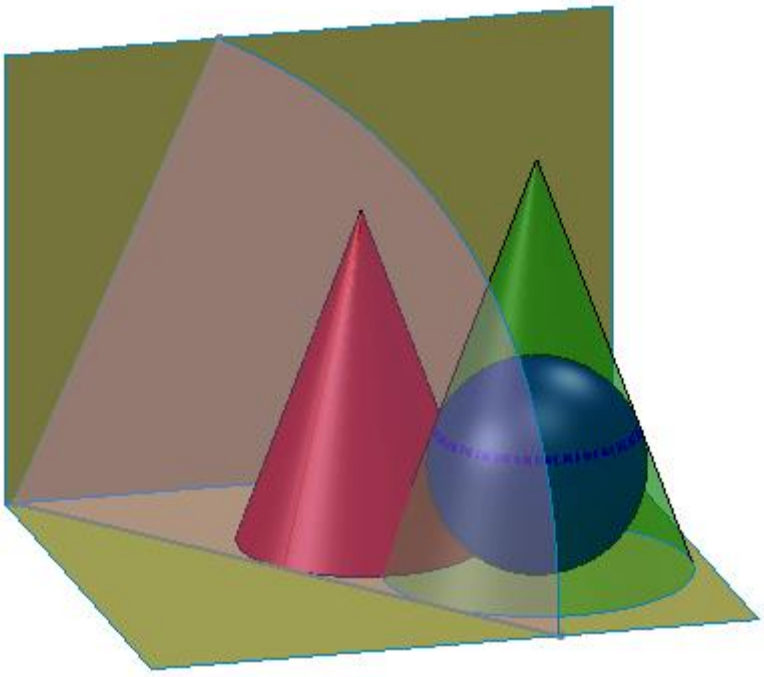
Tangent Planes





Prerequisite knowledge

To complete this model you should have a working knowledge of Solidworks 2006/2009.

Focus of lesson

This lesson focuses on using SolidWorks to solve a geometrical problem. The following **Surfaces** tools are used: **Planar Surface, Ruled Surface, Surface Revolve and Filled Surface.**

Getting started.

New File

Create a new part file and save it as **Tangent Planes** in the desired location.

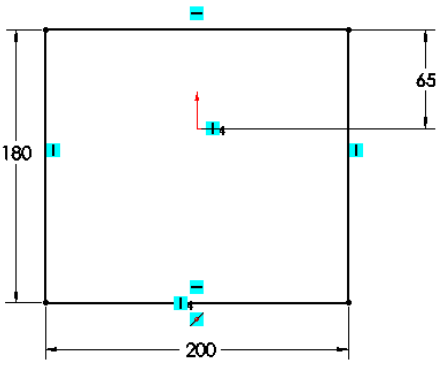
New Sketch

We are going to begin by creating a sketch to represent the outline of portion of the **Horizontal Plane**.

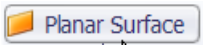
Create the sketch shown on the Top plane.

Smart Dimension and apply the relations as shown.

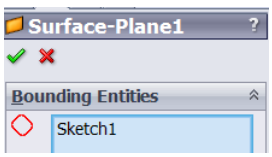
We want to transform this rectangle into a **Planar Surface.**



Planar Surface

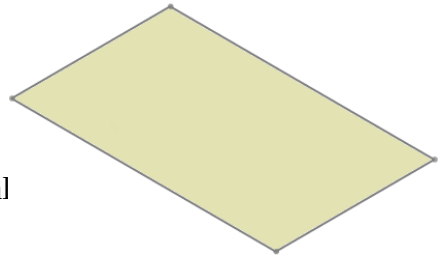
Select **Planar Surface** 

from the **Surfaces** toolbar.



Select **Sketch 1** as the **Bounding Entities.**

Select **OK**



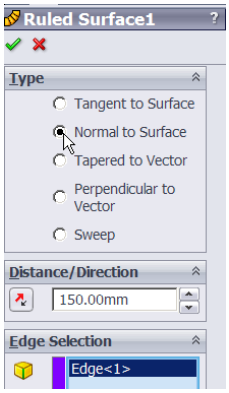
Rename Feature

Rename the feature as **Horizontal Plane.**

We have now created a portion of the horizontal plane. This planar surface has no thickness but can be used as a datum for measurements, a surface to project views onto, or a surface to sketch on.

The Vertical Plane

To create the vertical plane we use **Ruled Surface.**
Ruled Surface command Ruled Surface creates surfaces that extend out in a specified direction and distance from selected edges.



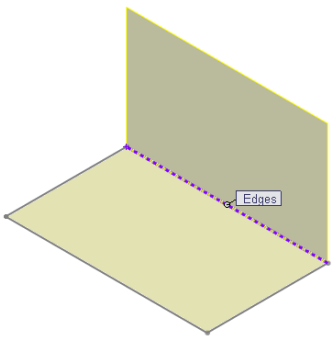
Ruled Surface

Select **Ruled Surface** from the **Surfaces** toolbar.

Select **Normal to Surface** as the **Type**.
This will create a ruled surface at 90 degrees to another surface at a specified edge.

Set the **distance** to 150mm. This will extend the surface out 150mm from the selected edge.
The width of the surface will be determined by the length of the edge selected.

Select the edge of the **Horizontal Plane** shown as the edge to set up the ruled surface from.



Select **OK**.

Rename Feature

Rename the feature as **Vertical Plane**.

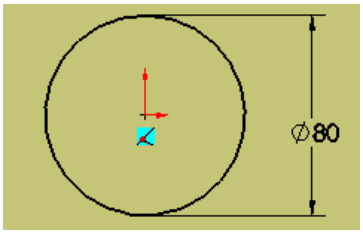
The Cone

We will use ***Lofted surface*** to create the cone.
We also require a plane to contain the apex of the cone.
(Alternatively we could use ***3D Sketch***)

Sketch

Create the sketch shown on the ***Horizontal Plane.***

Smart Dimension as indicated.



This sketch forms the base of the cone.

Rename Sketch

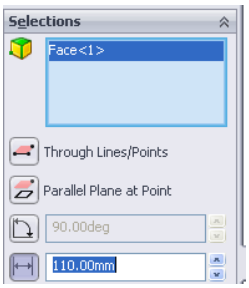
Rename the sketch as ***Trace of cone.***

Insert Plane

We must now insert a horizontal plane equal in altitude to the height of the cone.

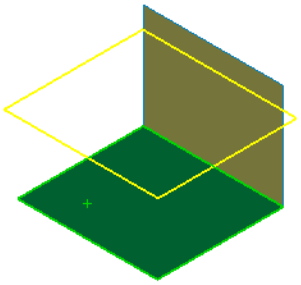
From the ***Surfaces*** menu select ***Reference Geometry,Plane***.

Make the following selections:



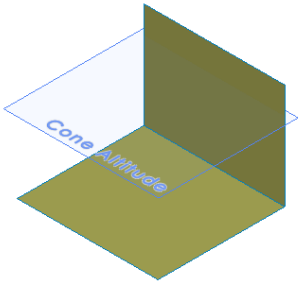
Choose the ***horizontal plane*** as the ***reference entity.***

Set the distance to 110 which is the altitude of the cone.



Select **OK**.

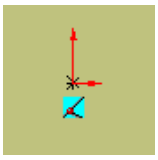
Rename Feature Rename the feature as ***Cone Altitude.***



This plane will contain the apex of the cone.

Sketch Create a new sketch on ***Cone Altitude.***

From the ***Sketch*** menu select ***Point.***
Make the point ***coincident*** with the ***origin.***



Confirm the Sketch.

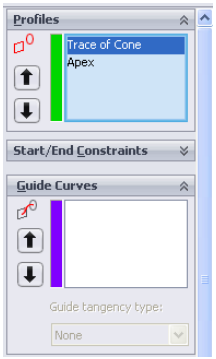
Rename Sketch Rename the sketch as ***Apex.***

Lofted Surface We will now create a ***Lofted Surface*** between the ***Trace of the cone*** and the ***Apex.***

About Lofted Surface The difference between lofted surfaces and lofted solids is that surfaces can use edges and curved features between which to loft rather than simply sketches and faces as is the case with solids. Guide Curves may be added, if necessary, to influence the resultant surface.

Creating the loft Choose **Lofted Surface** 

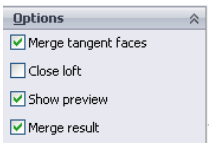
from the Surfaces toolbar



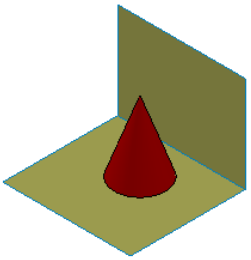
Select the two sketches as **Profiles**

We will not be using any guide curves.

Check - **Merge tangent faces.**
This will create one surface between the two profiles as opposed to a series of individual surfaces.



Show preview – will provide a preview of the loft.



Choose **OK**

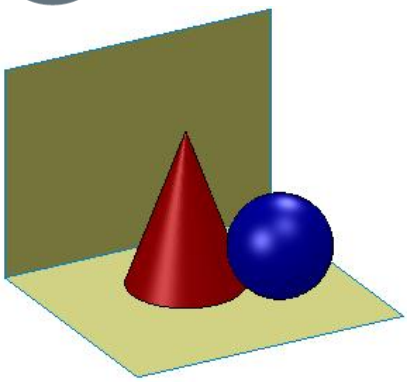
Hide Hide ***Cone Altitude.***

Appearance Apply a red colour as shown to the cone.

Rename Feature Rename the feature as ***Cone.***

Setting Up
The Sphere

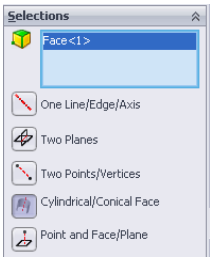
The ***sphere*** is in contact with the ***cone***.
The point of contact between the ***sphere*** and the ***cone*** is contained on a vertical plane which also contains the vertical axes of both the ***sphere*** and the ***cone***. The plane is inclined at ***45 degrees*** to the ***vertical plane***.



Axis of Cone

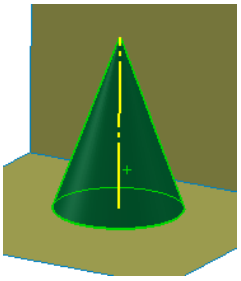
To define the plane we will use the axis of the ***cone***
To view the axis of the ***cone*** select ***Reference Geometry, Axis***. 

Make the following selections:



Choose ***Cylindrical/Conical Face***

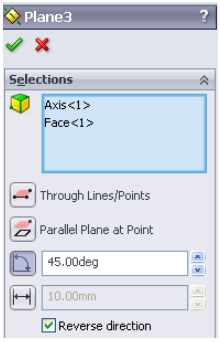
Then select the conical face of the ***Cone*** for the ***Reference Entities.***



Insert Plane

We must now insert a vertical plane which contains the axis of the ***cone*** and is inclined at 45 degrees to the ***vertical plane***.

From the ***Surfaces*** menu select ***Reference Geometry,Plane***.
Make the following selections:

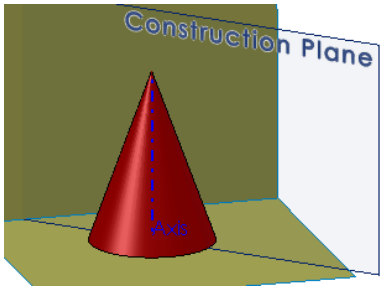


Choose the ***vertical plane*** and the ***Axis*** of the ***Cone*** as the ***reference entities.***

Set the angle to 45 degrees.

Reverse direction if required.

Select **OK**.



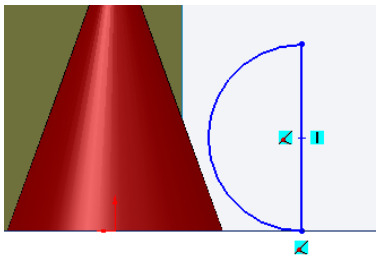
Hide

Hide the axis of the cone.

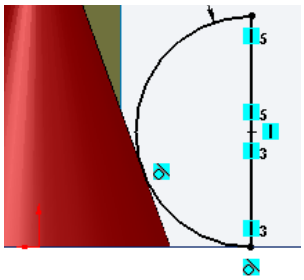
Sketch

Create a sketch on the ***Construction plane.***

Using the ***Line*** command set up the sketch opposite. Note the automatic relations.



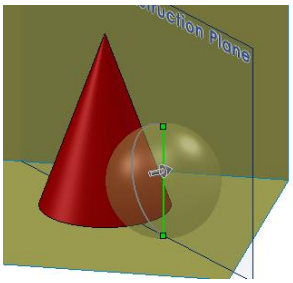
Smart Dimension and apply the relations as indicated. These relations ensure that the sphere and cone are tangent to each other.



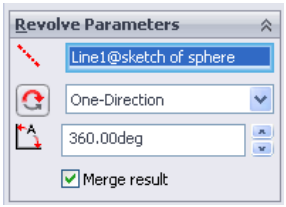
Confirm the Sketch.

Rename Sketch
Rename the sketch as ***Sketch of Sphere.***

Revolve
From the ***Features*** menu select ***Revolve.***



Select the diameter line of the Semi-circle as the ***axis of revolution.***

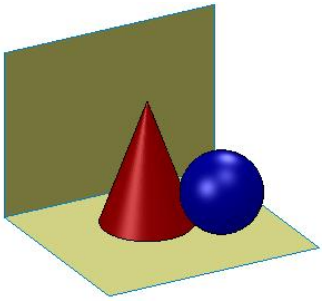


Select **OK**.

Appearance
Apply a blue colour to the ***Sphere***.

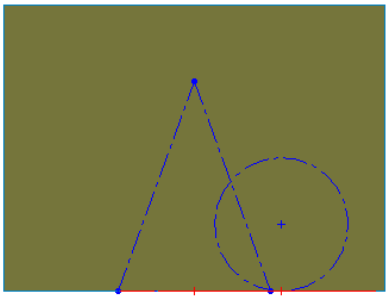
Rename Feature
Rename the feature as ***Sphere.***

We now have the ***"Solids in Contact"*** portion set up.



Creating the Orthographic Views
The orthographic projection of the Solids in Contact can be created using the ***Convert Entities*** command. 

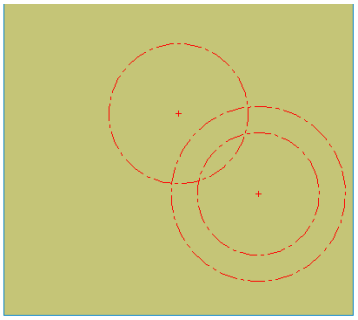
We must first of all create a new sketch on the ***Vertical plane***. Select a ***front*** view.



Hold down the ***Ctrl*** key and select all of the lines on the sketch. Now select ***Convert Entities.*** The elevation is created on the vertical plane.(Solids hidden for illustration)

Confirm the sketch. 

Rename Sketch
Rename the sketch as ***Elevation.***

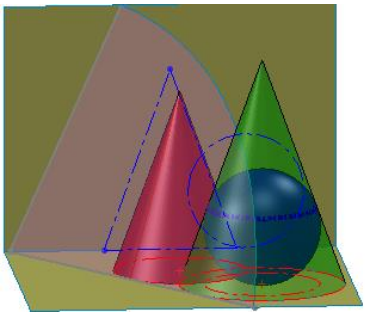


Repeat the process for the ***plan*** view, creating the sketch on the ***horizontal plane*** this time.

Confirm the sketch. 

Rename Sketch
Rename the sketch as ***Plan.***

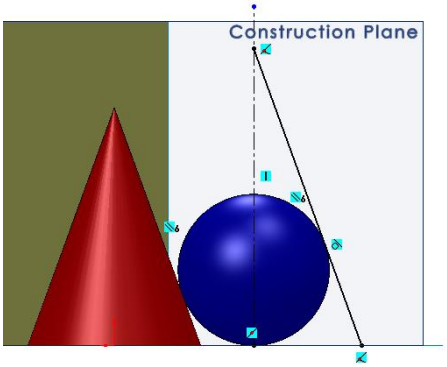
Setting up the Tangent Plane
In order to set up the tangent plane, we will have to create a construction cone which fits down over the ***Sphere***; is tangent to the ***Sphere*** and has the same base angle as the ***Cone***. The tangent plane will rest against both of these cones.



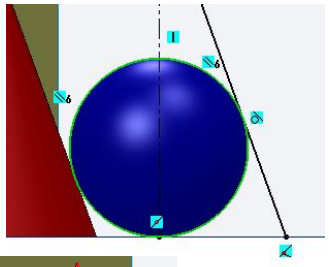
Sketch
Create a sketch on the ***Construction Plane.***

Set up the centreline first. This will be the axis of the ***construction cone***.

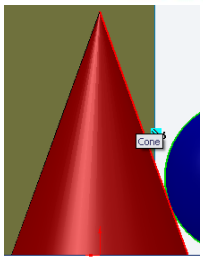
Sketch the line then which will be the ***generator*** of the ***construction cone*** and will be the line of intersection between the ***construction plane*** and the ***construction cone.*** (True Length)
Make one of the endpoints of the ***generator line coincident*** with the ***centreline*** as shown, and the other ***coincident*** with the groundline.



A ***tangent relation*** is applied between the ***sphere*** and the ***generator line***. The intersection of the ***construction plane*** and the ***sphere*** produces a circle so it is this circle that is selected for the relation. (True Length)



A ***parallel relation*** is applied between the ***generator line*** and the generator of the ***cone*** which contains the point of contact, which is also the line of intersection between the ***construction plane*** and the ***cone***.



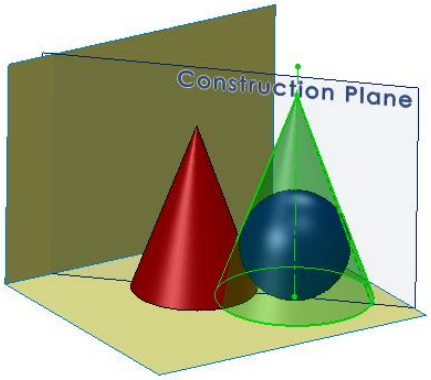
Confirm the sketch.

Rename Sketch

Rename the sketch as ***Generator Line.***

Surface Revolve

From the ***Surfaces*** menu select ***surface revolve.*** ***Surface revolve*** acts in the same manner as the ***revolve*** command in the ***features*** menu, but instead of producing a solid body it produces a hollow body.

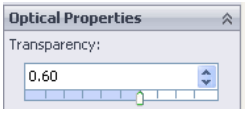


Select the centreline as the ***axis of revolution.***

Select OK.

Appearance

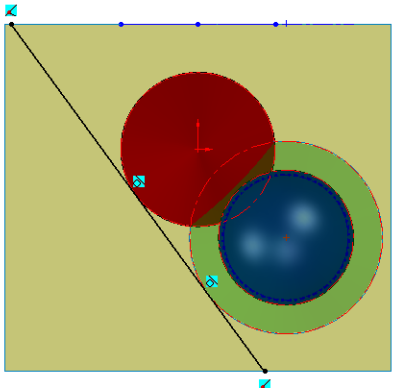
Apply a green colour to the ***Construction cone.***



Change the ***Transparency setting*** to ***0.6.*** This enables us to view the ***sphere*** "inside" the ***Construction cone***.

Hide

Hide the ***Construction Plane.***



Horizontal Trace

Create a sketch on the horizontal plane and using the line command draw the line shown Tangential to the bases of both cones i.e. ***HT.***

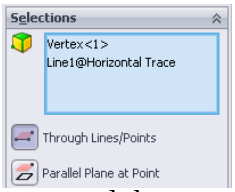
Rename Sketch

Rename the sketch as ***Horizontal Trace.***

Insert Plane

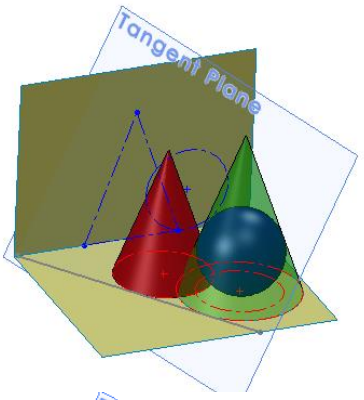
From the ***reference geometry*** tool select***Plane.***

Select ***Through Lines/Points*** as the end condition.



Choose the apex of either cone and the ***Horizontal trace*** as the ***reference entities***.

Select OK.



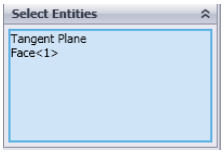
Rename Feature

Rename the feature as ***Tangent Plane.***

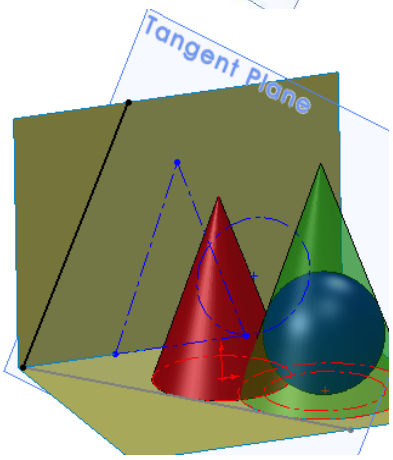
Vertical Trace

From the ***tools*** menu select ***sketch tools***, ***intersection curve***.

Select the ***tangent plane*** and the ***vertical plane*** as the ***entities to generate the intersection curve***



Select OK.



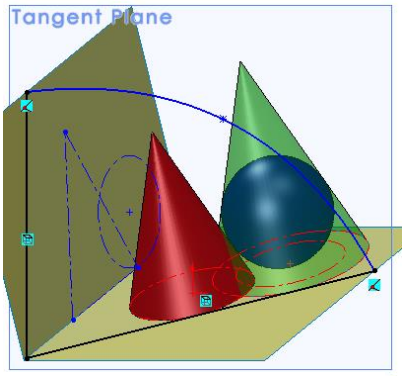
Rename Sketch

Rename the sketch as ***Vertical Trace.***

Portion Of Tangent Plane

Using the ***spline*** tool and ***convert entities*** create the sketch shown on the ***tangent plane***.

Confirm the sketch.



Rename Sketch

Rename the sketch as ***sketch of portion of tangent plane***

Hide

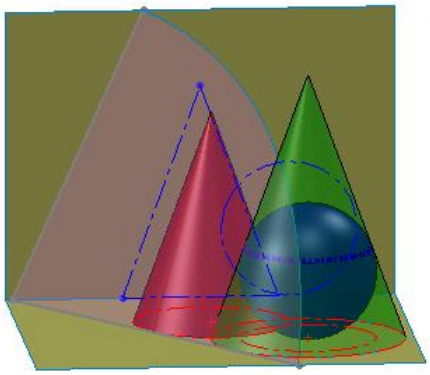
Hide the ***tangent plane***.

Planar Surface

From the ***surfaces*** menu select ***planar surface***

Select the ***sketch of portion of the tangent plane*** as the ***bounding entities.***

Select OK.



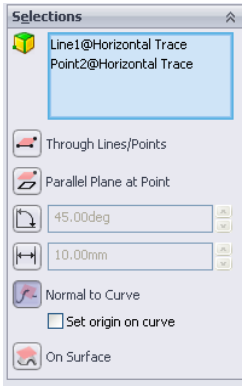
Rename Feature

Rename the feature as ***portion of Tangent Plane.***

Plane Perpendicular To Horizontal Trace

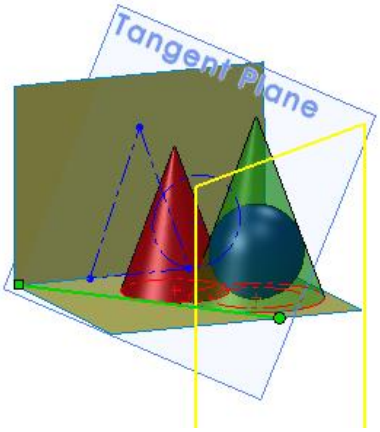
We now want to set up a plane onto which an auxiliary view could be projected showing a line view of the ***tangent plane,*** and the true inclination of the ***tangent plane*** to the ***horizontal plane***. Therefore we need to set up a plane perpendicular to the ***horizontal trace.***

Once again select ***reference geometry, plane.***



Choose ***normal to curve*** as the end condition.

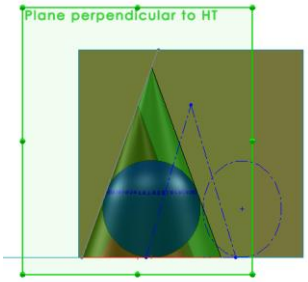
Select the ***horizontal trace*** and the end point of the ***horizontal trace*** as the ***reference entities.***



Select **OK.**

Rename Feature

Rename the feature as ***Plane Perpendicular To HT.***

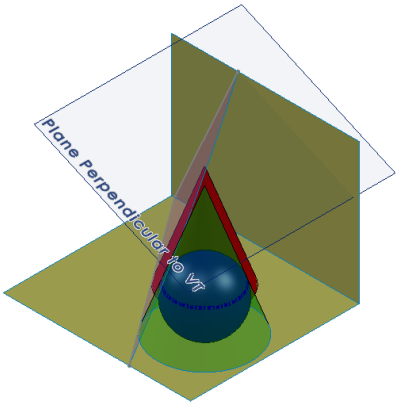


If we want to view the true inclination of the Tangent Plane to the horizontal plane, simply Select the ***Plane Perpendicular To HT*** and choose a ***normal to*** view.

Plane Perpendicular To Vertical Trace

Repeat the steps above to create a plane perpendicular to the ***Vertical Trace.***

This time select the ***vertical trace*** and its endpoint as ***the reference entities.***



Exercise complete!