



# ANSYS Pressure Equipment Module

ASME PVP 2007

Janet Wolf  
Sr. Application Engineer



- Overview of the module
- Modelling the components
- Analyzing the equipment (overview)
- Building the geometry
- Solving/Postprocessing



# ANSYS Pressure Equipment Module

## Overview



- **Why?**
  - reduce the time spent on designing pressure equipment
  - design your systems following best practice rules and design codes
  - lighten and/or rigidify a structure, or improve the quality and the life expectancy of your products
- **Application Range**
  - design and optimize any part of pressure equipment along with other accessories, structures and mechanical parts
  - Design will be made according to Codap, ASME and EN 13445 codes
  - Load cases not directly covered by the design codes: local loads due to piping and supports, wind loads, ends with offset or tilted nozzles near the knuckle radii and beyond the limits provided by the codes, adjacent nozzles ...

- The ANSYS Pressure Equipment Module is integrated within ANSYS Workbench and offers all necessary modelling functions for Pressure Equipment within ANSYS DesignModeler.
- Its numerous capabilities have been designed to help you gain a valuable amount of time and effort :
  - Parametric modelling of pressurised systems components and associated nozzles.
  - Local and global zones definition for the review of stresses according to the stress categories stated in Codap, ASME and EN 13445.
  - Review of the stress results according to chapter C10.1 of Codap, ASME VIII Div 2, and EN 13445

# Application fields



- Thermal and nuclear plants
- Oil & Chemicals Industry
- Marine
- Automotive industry





# ANSYS Pressure Equipment Module

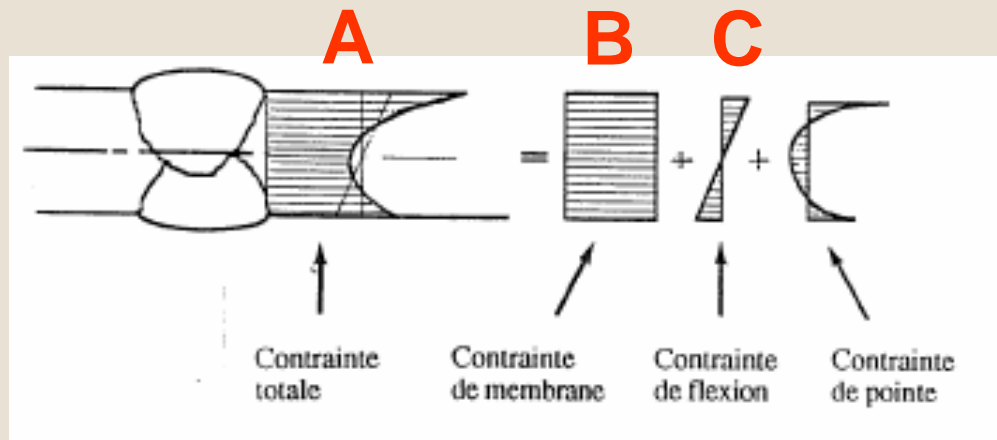
## Definitions



# Stress decomposition



- **Total stress (A):**
  - Elastic stresses as a result from external loads
- **Membrane stress (B):**
  - Average value through the thickness of the part
- **Bending stress (C):**
  - Linearly varying part of the stress through the thickness



# Which results are we looking for?



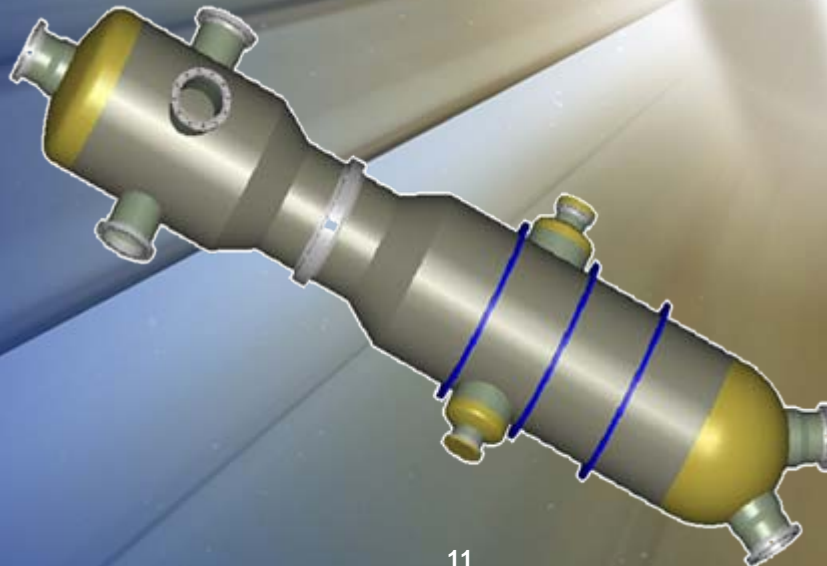
- **Primary stresses**
  - Due to mechanical loads
  - satisfies force and moment equilibrium
  - Primary stress that exceeds the yield stress by some margin will result in failure
  - Exclude stress concentrations
- **Secondary stresses**
  - arise from geometric discontinuities or stress concentrations
- **Primary membrane stresses**
  - Membrane component of stresses in the global area
- **Primary local membrane stresses**
  - Membrane component of stresses in zones with major discontinuities (local areas)

- Major discontinuity: geometric or material change affecting stresses through the entire thickness (cap junction, thickness variations, different materials...)
- Minor discontinuity: geometric or material change affecting stresses through part of the thickness (small fillets...)
- Local area: area around major discontinuities
- Global area: areas « well away » from local areas, where discontinuity effect is small enough



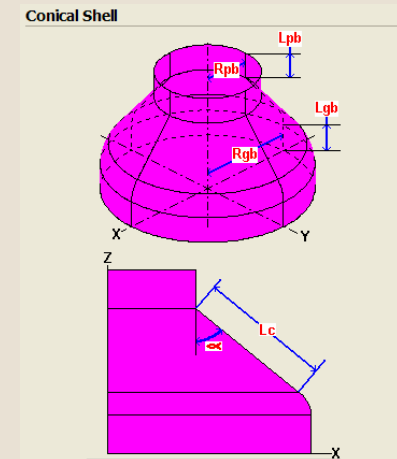
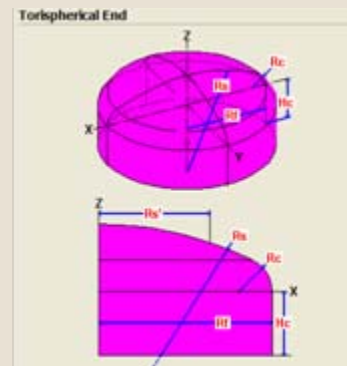
# ANSYS Pressure Equipment Module

Modeling the components

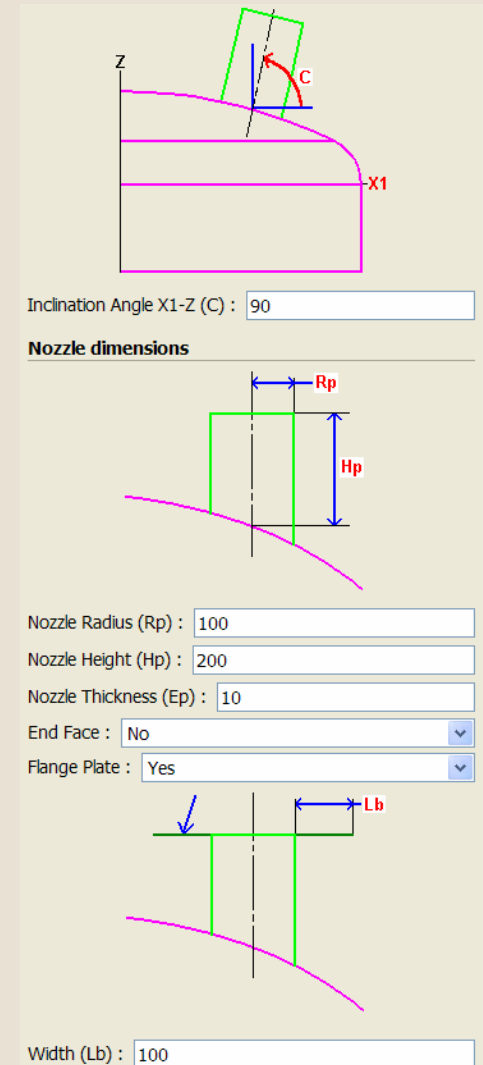


- **Direct link with your CAD system**
  - The ANSYS Pressure Equipment Module can be started from your CAD system interface and is fully associative with most of the CAD systems:
    - SolidWorks
    - Solid Edge
    - Pro/Engineer
    - Unigraphics,
    - Mechanical DeskTop
    - Inventor
    - CATIA V5
  - It allows you to make design changes to your CAD model without having to reapply loads and/or supports. You can either pick the CAD dimension to change directly, or enhance your design iterations with the Parameter Manager.
  - The ANSYS Pressure Equipment Module can also be used with native Parasolid, ACIS or IGES files.

- **Geometry creation:**
  - Surfaces extraction. Projection feature for loads and boundary conditions definition.
  - 2D sketching
  - sketches creation, dimensioning, constraints definition, parameters modification, ...
  - 3D technological functions
  - extrusion, revolution, sweeps, fillet radii, shell/surface, ...
- **Parametric modelling of pressure equipment components**
  - Cylindrical and conical shells
  - Torispherical ends
  - Hemispherical ends
  - Elliptical ends
  - Flat ends



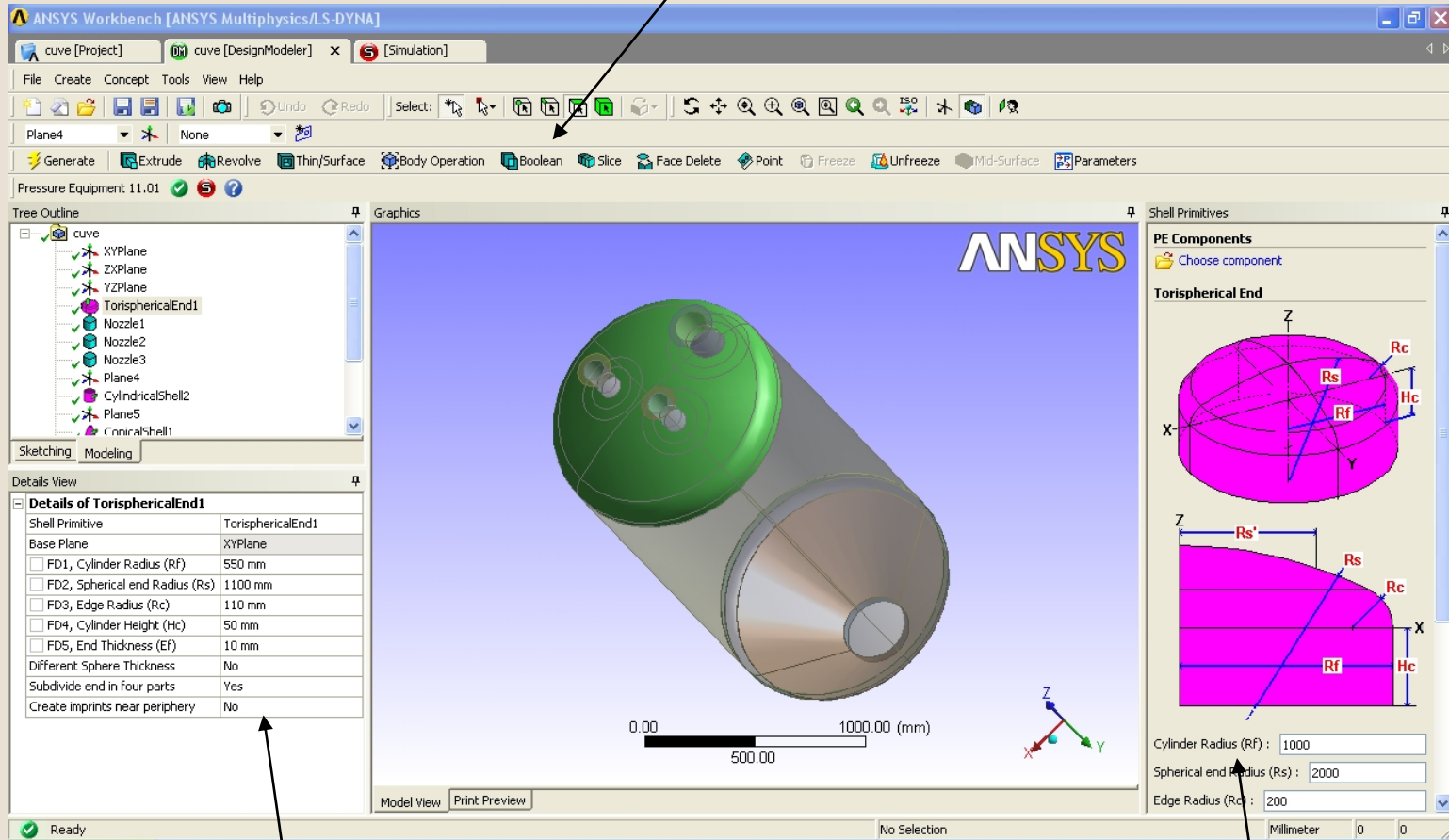
- **Nozzles:**
  - Modelling of the flange face
  - Modelling of the bottom face
  - Protruding nozzles
  - Reinforcement at the base of the nozzle along its height
  - Reinforcement around the nozzle on the main component
- **Conceptual modelling**
  - Wireframes for beam structure analysis
  - Beam cross sections parametric database
  - Mixed beam/shell models



# Modeling interface



## Standard modeling tools



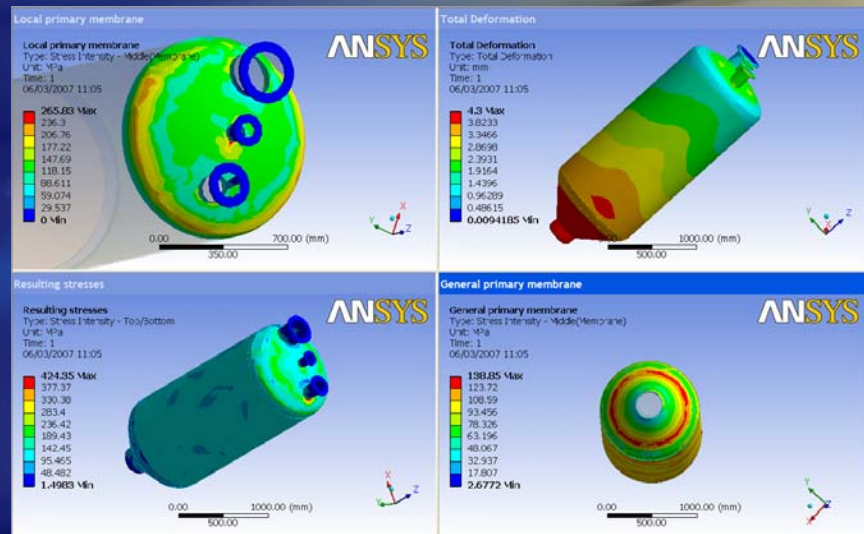
Models are fully parametric

Pressure equipment primitives



# ANSYS Pressure Equipment Module

## Analyzing the equipment



- Surface model meshes are built with triangles and/or quadrilateral elements.
- Multiple parameters can be controlled:
  - Refinement to control the overall density of meshes on selected geometry
  - User defined Defeaturing tolerance so the mesh skips small details of the geometry)
  - Initial sizing to control the mesh on edges or surfaces.
  - Local mesh refinement around points, edges and/or surfaces

- Materials and sections database, including the ability to define your own materials
- Mechanical and thermal load cases definition (point loads, distributed loads, pressure, hydrostatic pressure, dead-weight, prescribed displacements, centrifugal force, prescribed temperature)
- Boundary conditions like restraints, elastic supports, prescribed displacements
- Visualisation and control of all the data

- **Static Analysis**

- Deformations, stresses, strains, reaction forces...

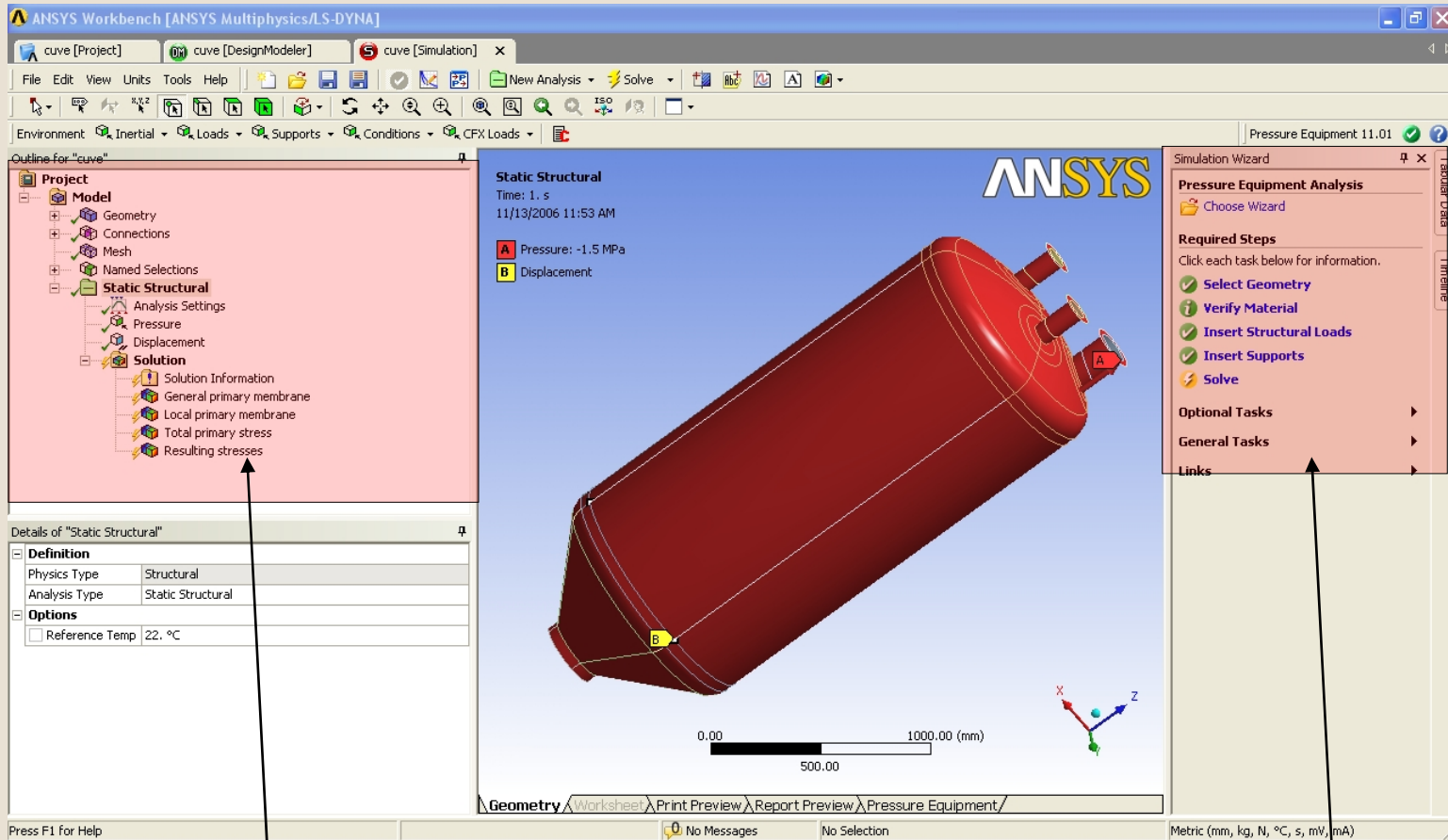
- **Stability Analysis**

- The study of the stability of structures is a major concern for engineering companies
- The buckling analysis of beam, plate or shell structures is incorporated within the ANSYS Pressure Equipment Module
- allows you to calculate the maximum pressure the equipment can sustain before buckling occurs.

- **Analysis of the results can be made in various forms:**
  - Deformed shape for static analyses, natural frequency and mode shapes for dynamic analyses, coloured scale iso-displacements,
  - Coloured scale iso-stresses,
  - Iso-stresses on section for beam elements,
  - Dynamic 3D cutting section,
  - Alert warning when exceeding defined criterion,
  - Results animation, and AVI animation file creation
  
- **Stress results analysis tool**
  - Depending on the classification of the elements in local or global zones it is possible to plot:
    - global primary membrane stresses.
    - local primary membrane stresses.
    - total primary membrane stresses.
    - equivalent variation of the combined stresses.

- You can also visualize the extension of local zone (zone where the membrane stresses are classified as PL) in order to check whether their extension or proximity is acceptable or not, according to the rules of Codap, ASME, and EN 13445. Global and local zones can be modified as well.
- The ANSYS Pressure Equipment Module includes an analysis report generator, available in HTML, Word and PowerPoint format. The report provides a real time list of all calculated analysis scenarios and enables the user to keep track of the design and analysis process.

# Simulation interface



Simulation tree

Simulation wizard

# Sample Stress Analysis



## Stress analysis for pressurised equipment

Model : Model, Environment : Static Structural

### Local and global areas

Local areas :   
General areas :

Areas identification

### Select a norm

Norm Selection

### Definition of nominal design stress for materials

Click to select material and apply formula  
Double-click to edit a value

Materials	Re	Rm	Nominal design stresses
Structural Steel	250.00	460.00	166.67

Material data

### Stress admissibility criteria

Double-click to show details for each material.

Criteria	Coefficient	Max - C.f
General primary membrane	1	
<b>Material</b>	<b>Value</b>	<b>C.f</b>
Structural Steel	138.85	166.67
Local primary membrane	1.5	265.83 > 250.00
<b>Material</b>	<b>Value</b>	<b>C.f</b>
Structural Steel	265.83	250.00
Total primary stress	1.5	424.35 > 250.00
<b>Material</b>	<b>Value</b>	<b>C.f</b>
Structural Steel	424.35	250.00

Violations are highlighted

- CODAP
  - Primary membrane stress: C10.1.7.1.a
  - Primary local membrane stress : C10.1.7.1.b1
  - Total primary stress: C10.1.7.1.c
  - Resulting stress (primary+secondary): C10.1.7.2.a
  
- EN 13445
  - Equivalent stress (Tresca or Mises) : C.4.1
  - Primary global membrane stress: C7.2-1
  - Primary local membrane stress: C7.2-2
  - Total primary stress: C7.2-3
  - Resulting stress (primary+secondary): C7.3-1
  
- ASME
  - Primary global membrane stress: 4-131
  - Primary local membrane stress: 4-132
  - Total primary stress: 4-133
  - Resulting stress (primary+secondary): 4-134



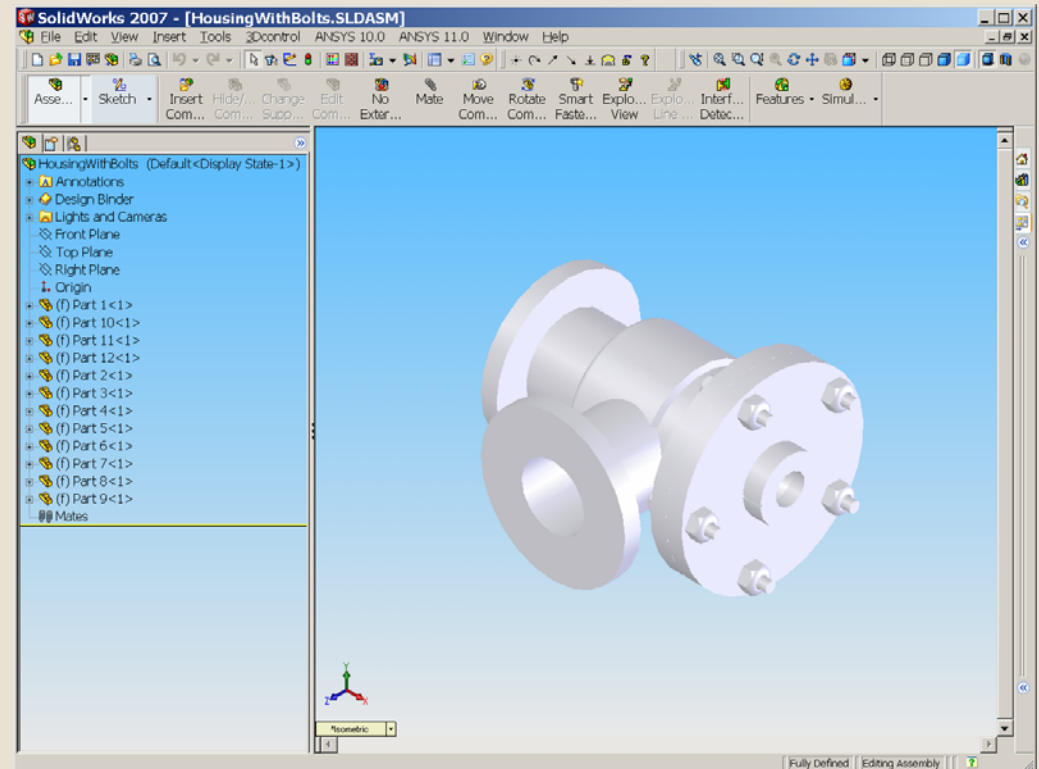
# Building the Geometry



# Importing from CAD



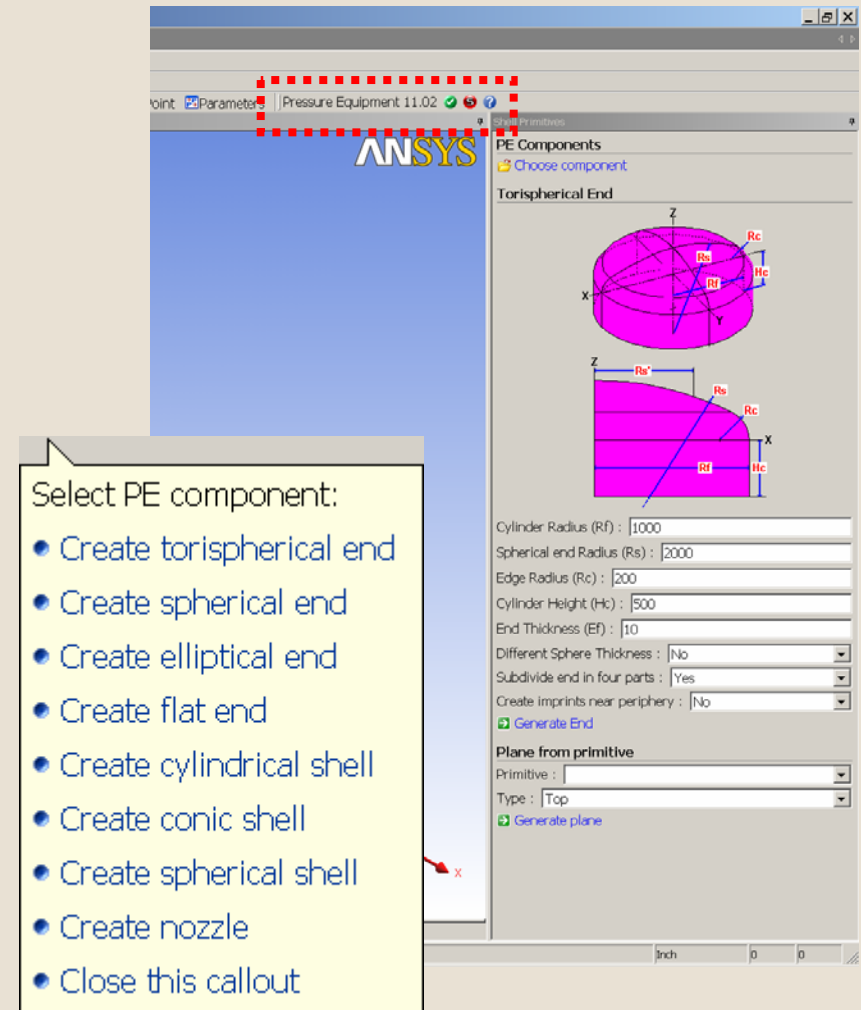
- Inventor, SolidEdge, SolidWorks, Pro/E, UG NX, Catia
- Also can import STEP, IGES, ACIS (\*.sat) and parasolid (\*.x\_t)
- Start Workbench and import model into Simulation
- Support solid and surface body (CAD dependent)



# Using DesignModeler



- Turn on Wizard
- Choose component
  - Torispherical end
  - Spherical end
  - Elliptical end
  - Flat end
  - Cylindrical shell
  - Conic shell
  - Nozzle



# Torispherical End



Shell Primitives

PE Components

Choose component

Torispherical End

Cylinder Radius (Rf) : 36

Spherical end Radius (Rs) : 72

Edge Radius (Rc) : 6

Cylinder Height (Hc) : 120

End Thickness (Ef) : 1

Different Sphere Thickness : Yes

Sphere Radius (Rs') : 24

Sphere Thickness (Es') : 1.5

Subdivide end in four parts : Yes

Create imprints near periphery : No

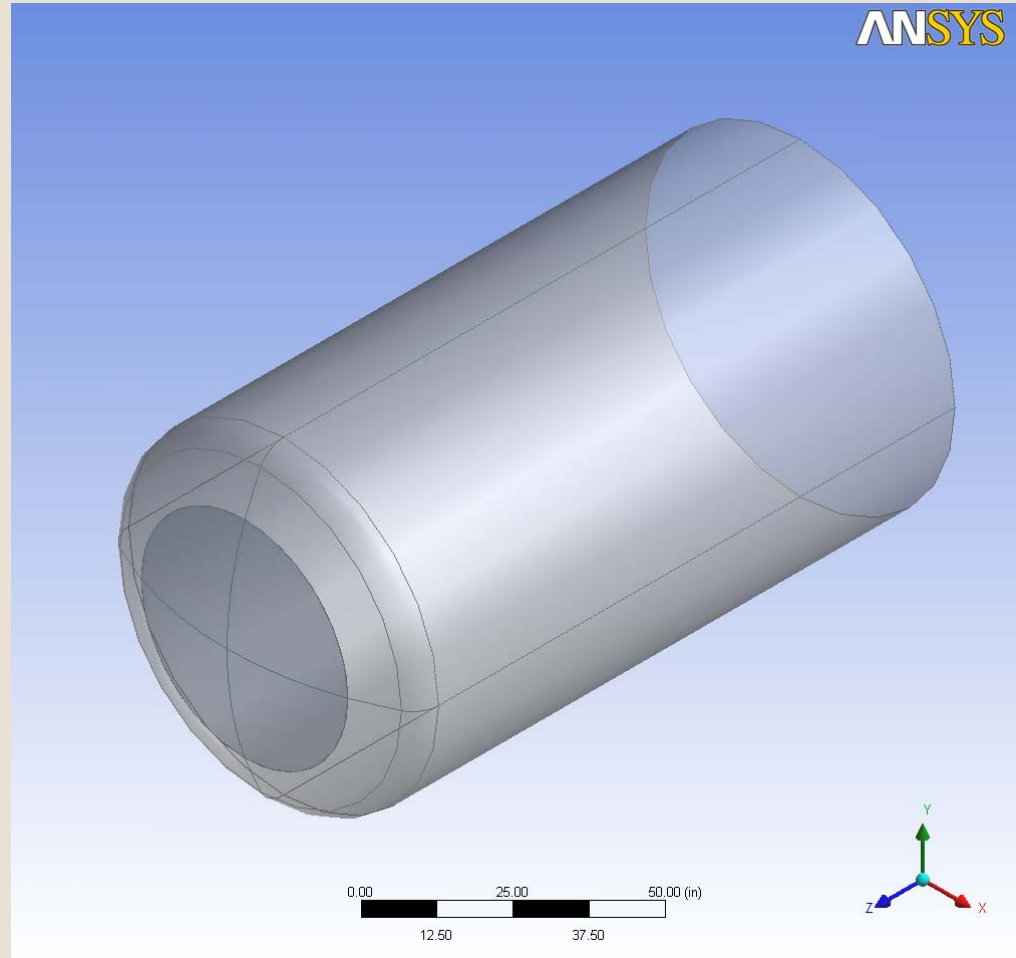
Generate End

Plane from primitive

Primitive :

Type : Top

Generate plane



# Spherical End



Shell Primitives

PE Components  
[Choose component](#)

Spherical End  
 End type : Hemispherical end

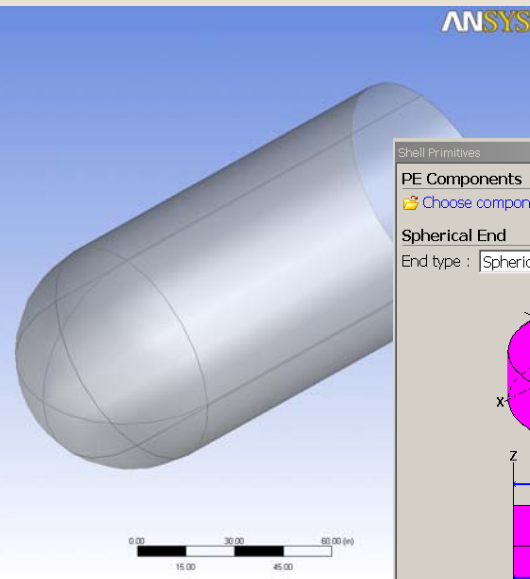
Spherical end Radius (Rs) : 36  
 Cylinder Height (Hc) : 120  
 End Thickness (Ef) : 1

Different Sphere Thickness :  No  
 Subdivide end in four parts :  Yes  
 Create imprints near periphery :  No

[Generate End](#)

Plane from primitive  
 Primitive : SphericalEnd1  
 Type : Top

[Generate plane](#)



Shell Primitives

PE Components  
[Choose component](#)

Spherical End  
 End type : Spherical end

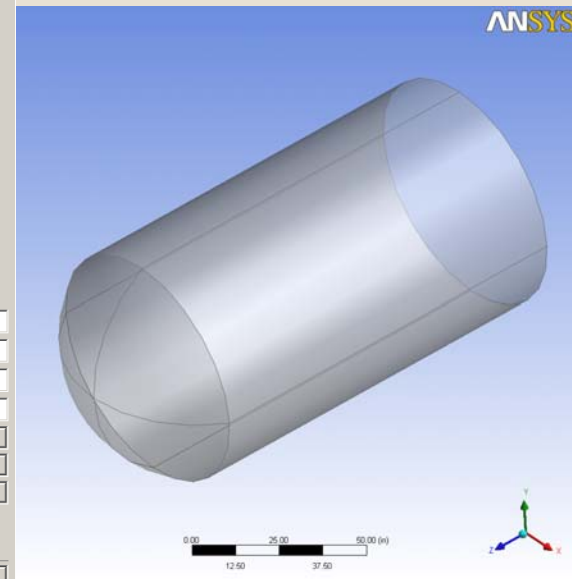
Cylinder Radius (Rf) : 36  
 Spherical end Radius (Rs) : 42  
 Cylinder Height (Hc) : 120  
 End Thickness (Ef) : 1

Different Sphere Thickness :  No  
 Subdivide end in four parts :  Yes  
 Create imprints near periphery :  No

[Generate End](#)

Plane from primitive  
 Primitive : SphericalEnd2  
 Type : Top

[Generate plane](#)



# Elliptical End



Shell Primitives

PE Components

Choose component

Elliptical End

Big Radius (R1) : 36

Small Radius (R2) : 18

Cylinder Height (Hc) : 120

End Thickness (Ef) : 1

Different Ellipse Thickness : No

Subdivide end in four parts : Yes

Create imprints near periphery : No

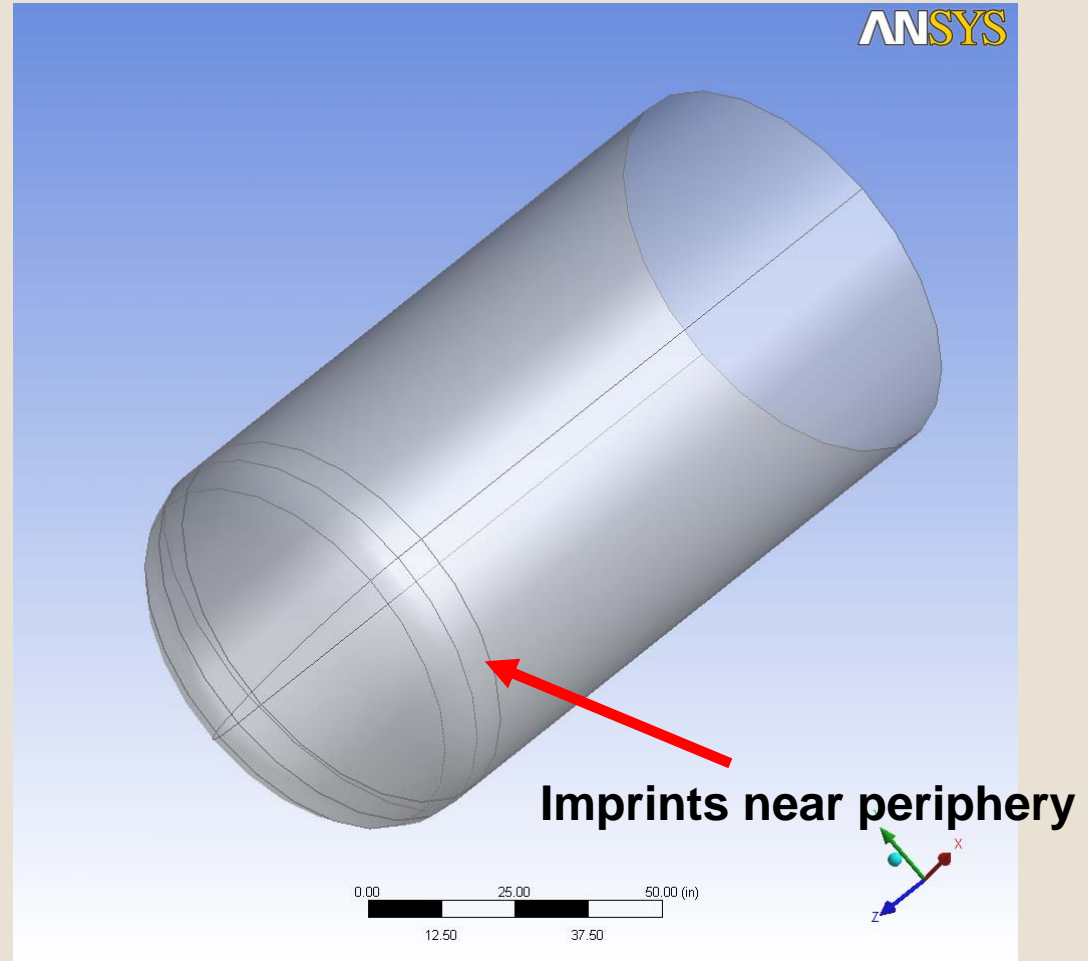
Generate End

Plane from primitive

Primitive : EllipticalEnd1

Type : Top

Generate plane



# Flat End



Shell Primitives

PE Components

Choose component

Flat End

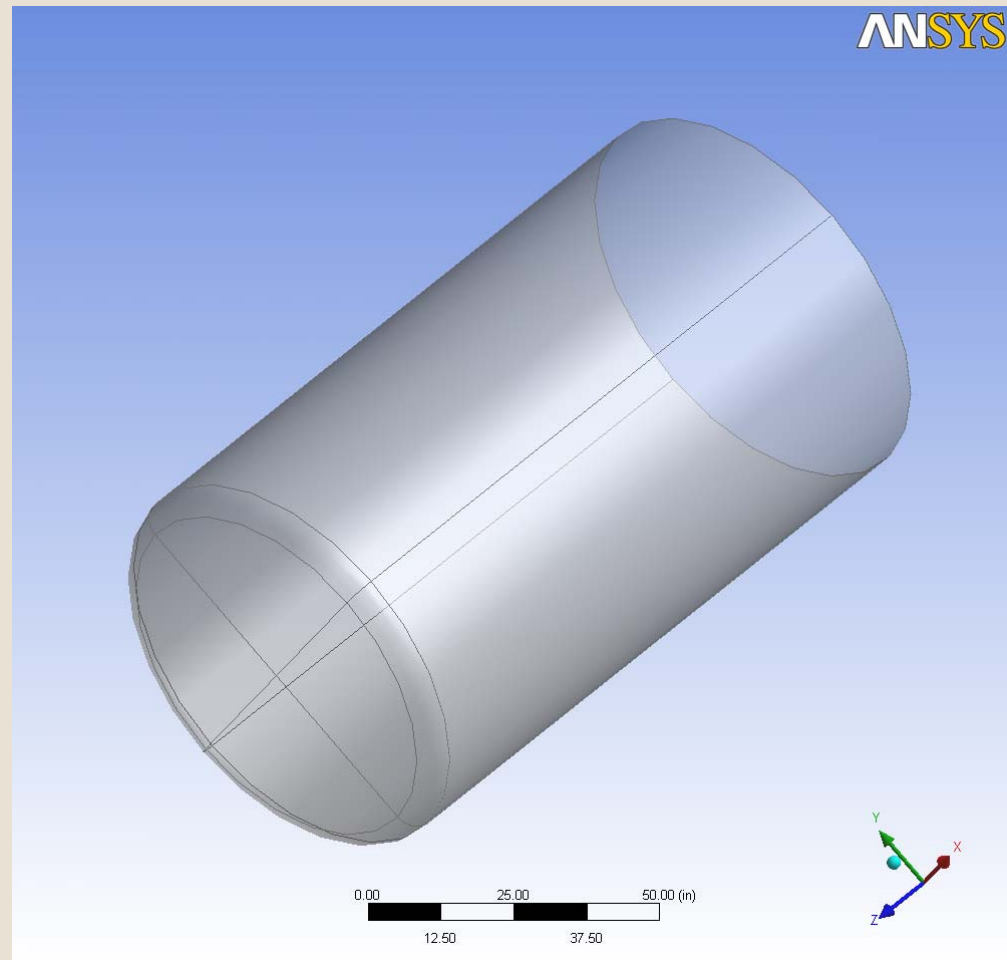
Cylinder Radius (Rf) : 36  
Edge Radius (Rc) : 4  
Cylinder Height (Hc) : 120  
End Thickness (Ef) : 1  
Different Sphere Thickness : No  
Subdivide end in four parts : Yes  
Create Imprints near periphery : No

Generate End

Plane from primitive

Primitive : FlatEnd1  
Type : Top

Generate plane



# Cylindrical Shell

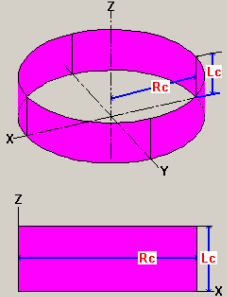


Shell Primitives

PE Components

Choose component

Cylindrical Shell



Cylinder Radius ( $R_c$ ):

Cylinder Length ( $L_c$ ):

Cylinder Thickness ( $E_c$ ):

Subdivide shell in four parts:

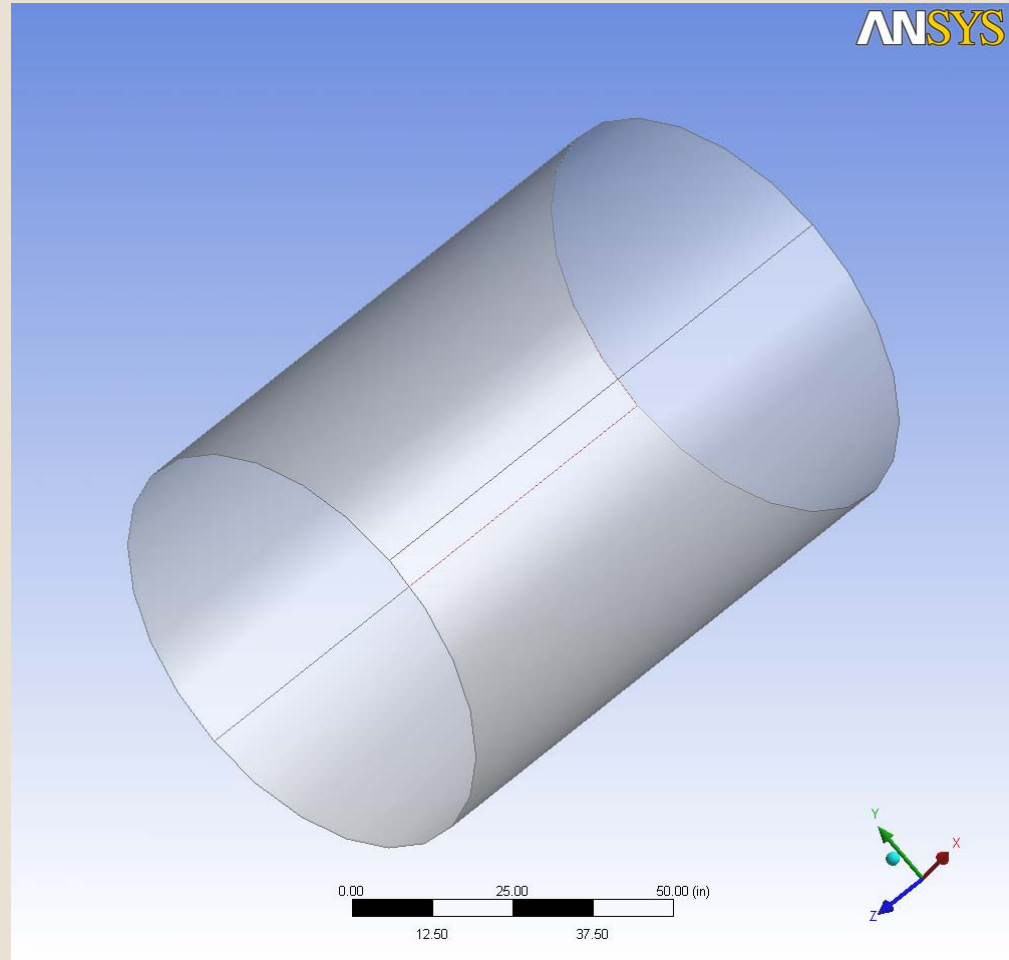
Generate Shell

Plane from primitive

Primitive:

Type:

Generate plane



# Conic Shell



Shell Primitives

PE Components  
Choose component

Conical Shell

Bottom radius (R<sub>gb</sub>) : 36  
Bottom length (L<sub>gb</sub>) : 120  
Bottom thickness (E<sub>gb</sub>) : 1  
Top radius (R<sub>pb</sub>) : 18  
Top length (L<sub>pb</sub>) : 36  
Top thickness (E<sub>pb</sub>) : 1.5  
Cone defined by : Cone half-angle  
Cone half-angle (Alpha) : 30  
Cone thickness (E<sub>c</sub>) : 1.5  
Slice end : Yes

Bottom - Cone junction  
Top - Cone junction

Create shell  
Generate Shell

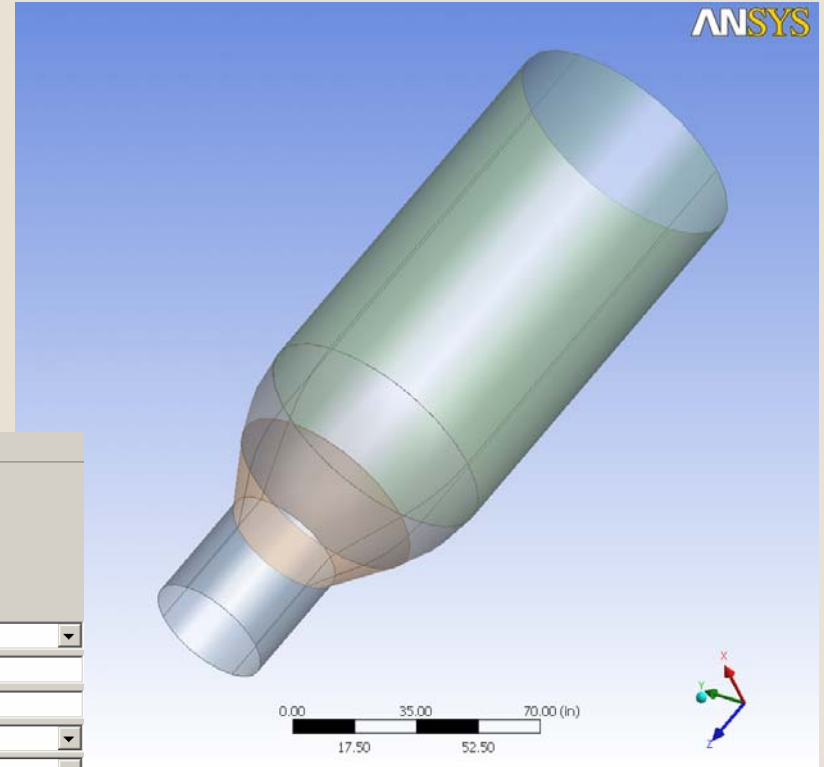
Plane from primitive  
Primitive : ConicalShell1

Bottom - Cone junction

Fittings : By toric part  
Torus radius (R<sub>tg</sub>) : 48  
Torus thickness (E<sub>tg</sub>) : 1.5  
Bottom additional thickness : No  
Cone additional thickness (Bottom) : No

Top - Cone junction

Fittings : With sharp angle  
Top additional thickness : No  
Cone additional thickness (Top) : No



# Spherical Shell

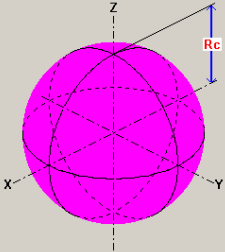


Shell Primitives

PE Components

Choose component

Spherical Shell



Sphere Radius (Rc) : 36

Sphere Thickness (Ec) : 1

Subdivide shell in eight parts : Yes

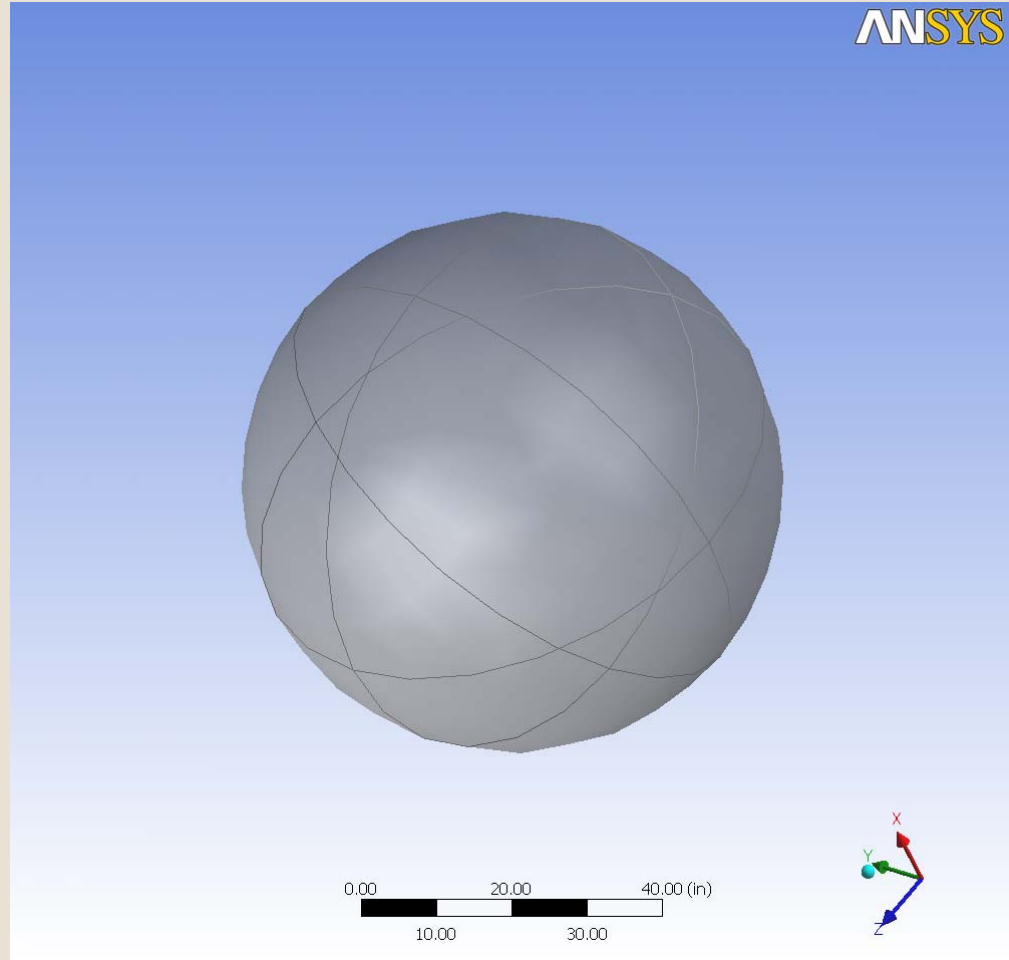
Generate Shell

Plane from primitive

Primitive : SphericalShell1

Type : Top

Generate plane



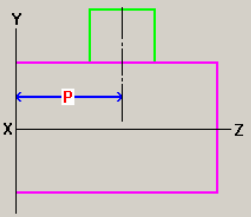
# Nozzle



Shell Primitives

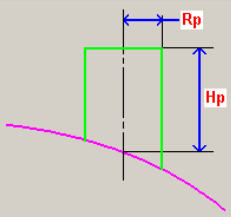
PE Macros  
Choose component

Nozzle position  
Primitive : CylindricalShell2  
Positioning Angle (A) : 36



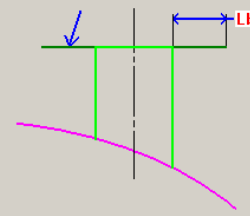
Position (P) : 100  
Orientation : Normal to the surface

Nozzle dimensions



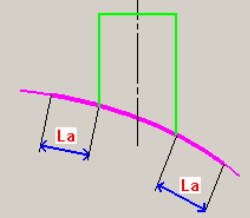
Nozzle Radius (Rp) : 9  
Nozzle Height (Hp) : 12  
Nozzle Thickness (Ep) : 1.5  
End Face : No  
Flange Plate : Yes

Flange Plate : Yes



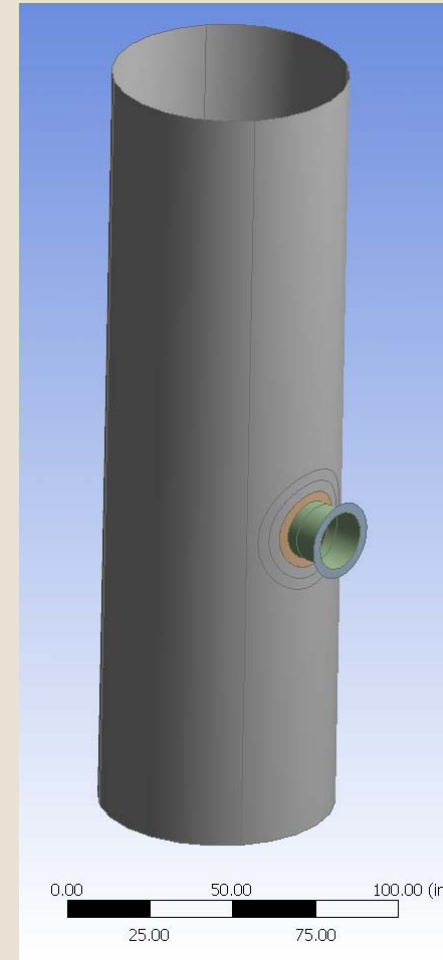
Width (Lb) : 3  
Thickness (Eb) : 1  
Nozzle Type : Flush Nozzle  
Slice nozzle : No  
Junction : Spherical

Reinforcements  
Nozzle Base Reinforcement : No  
Reinforcement Ring : Yes



Width (La) : 3  
Additional Thickness (Ea) : .5

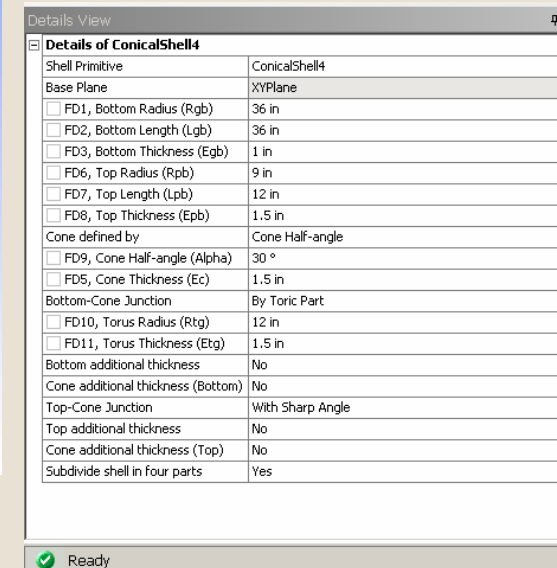
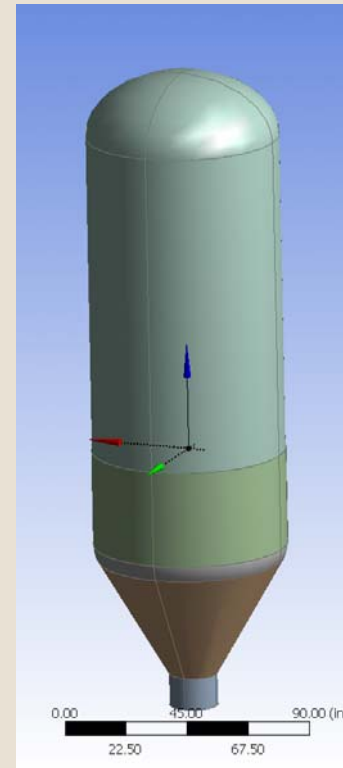
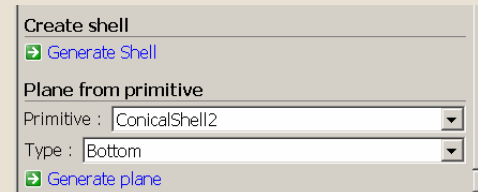
Create nozzle  
Generate Nozzle



# Putting it together



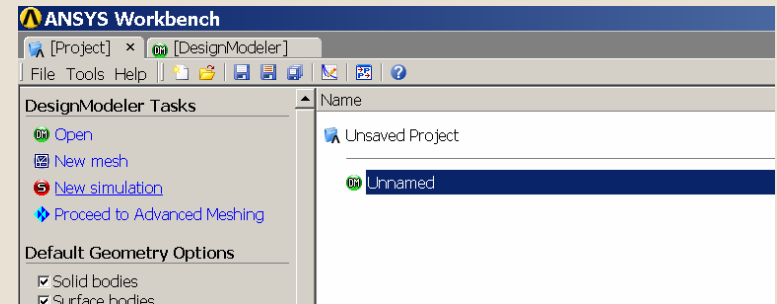
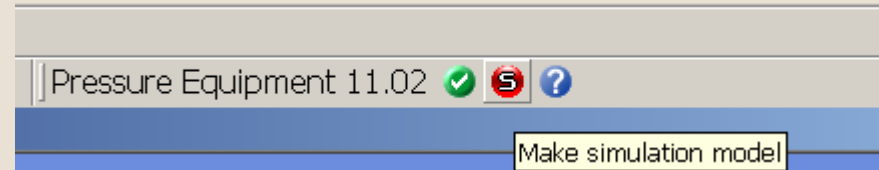
- Build on existing parts by creating a plane from existing parts (bottom of wizard)
- Check direction of plane— if necessary, create new plane with Z reversed
- Need to change something? All the dimensions and choices entered in the Wizard are in the Details View



# Prepare for Analysis



- Before exporting to Simulation, need to click Make Simulation Model
- Connects the parts together, so the mesh is continuous
- Go to Project Page and click New Simulation





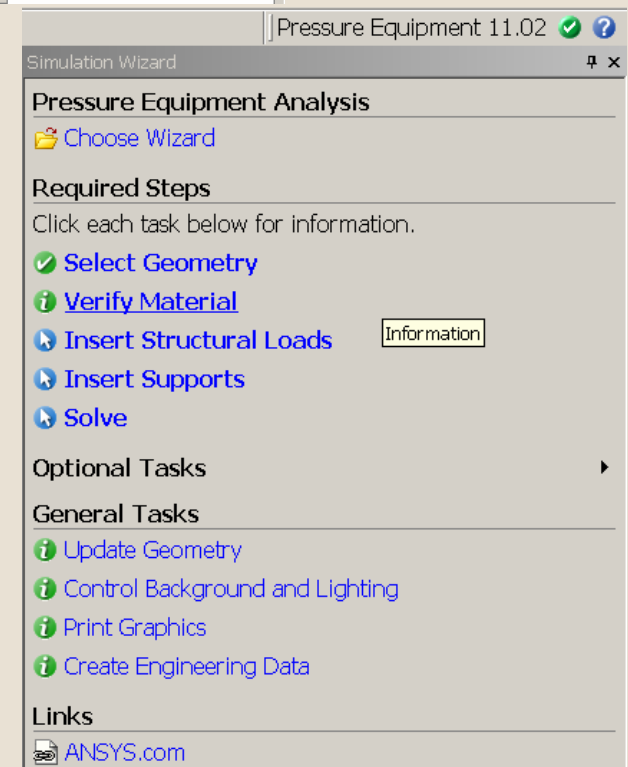
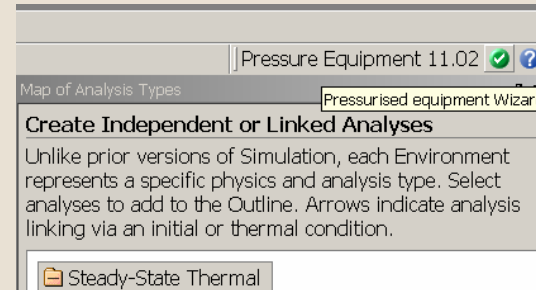
# Solving/Postprocessing



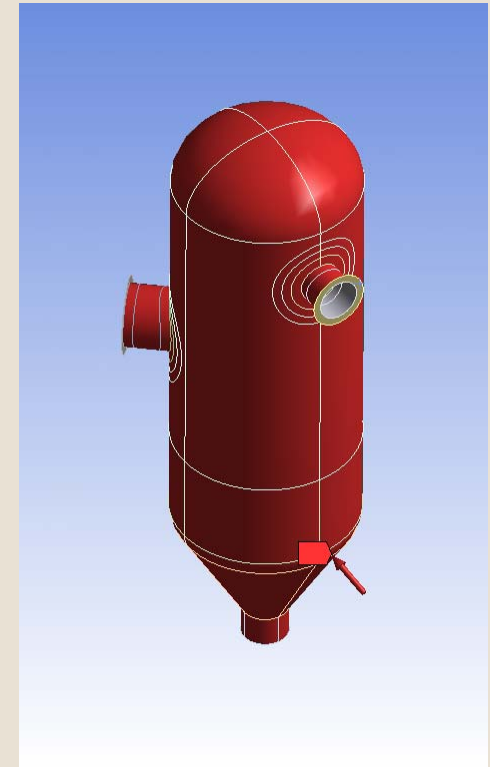
# Start the Wizard



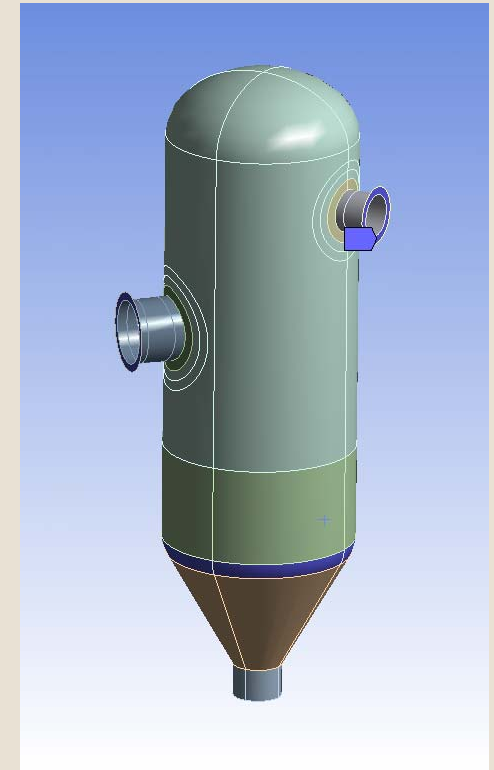
- Click the Pressure Equipment Wizard button
- Verify the material



- Insert Structural Loads
  - Gravity
  - Forces
  - Pressure (Hydrostatic)
  - Torque (Moment)
  - Specified Displacement
- Check direction
  - Internal pressure needs to be negative in this case
- Right-click to Select All
- Hold down Ctrl key for multiple select



- Insert Supports
  - Fixed
  - Frictionless support
    - Supported normal to face (flat or curved)
  - Remote displacement
  - Simply supported
  - Fixed rotation



- Solve
- Can solve on your machine or on a remote machine
- If meshing hasn't been done yet, it's done now
- Click Pressure Equipment Tab
- Select the Norm
- Set the nominal stress values
- Click Calculate

The screenshot displays the ANSYS Workbench interface for a stress analysis. The main window is titled "Stress analysis for pressurised equipment". The model is identified as "Model : Model, Environment : Static Structural".

**Local and global areas**

Local areas : [Local\_area]  
General areas : [General\_area]

**Select a norm**

[ASME]

**Definition of nominal design stress for materials**

Double-click to edit a value

Materials	kSm	Sps
Structural Steel	24000	24000

**Stress admissibility criteria**

Double-click to show details for each material.

Criteria	Coefficient	Max - C.f
General primary membrane	1	
Local primary membrane	1.5	
Total primary stress	1.5	
Resulting stresses	3	

Buttons: Calculate, Print report

Messages window: Warn: One or more contact regions may not be in initial contact. Ched Project>Model>Stz

- View code results on Pressure Equipment Tab
- View stress results in Tree
- View deformation (add to tree and Evaluate)
- View report

