



Product Features

Geometric model creation

- ▶ Model creation integrated within the most widely used CAD systems
- ▶ Fully associative and compatible with most CAD systems
- ▶ Geometry creation through the modeller functions and with Boolean operations
- ▶ Surfaces extraction. Projection feature for loads and boundary conditions definition
- ▶ 2-D sketching
 - sketches creation, dimensioning, constraints definition, parameters modification...
- ▶ 3-D technological functions
 - extrusion, revolution, sweeps, fillet radii, shell/surface...

Pressure Equipment Analysis Software According to Codap, ASME and EN 13445

Whether you need to 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, ANSYS Pressure Equipment Module offers a simple and efficient solution.

Application Range

The ANSYS Pressure Equipment Module allows you to 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. Additionally, this module handles load cases not directly covered by the design codes: load cases other than internal pressure such as 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 ...



Benefits

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 :

Nozzles can be created using ANSYS DesignModeler

- ▶ 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, and EN 13445

Product Features

Parametric modelling of pressure equipment components

- › Cylindrical and conical shells
- › Torispherical ends
- › Hemispherical ends
- › Elliptical ends
- › Flat ends

Creation of component nozzles with the following characteristics

- › Bottom face
- › Flange face
- › Protruding nozzle
- › Reinforcement around the nozzle on the main component
- › Reinforcement along the height of the nozzle

Conceptual modelling

- › Wireframes for beam structure analysis
- › Beam cross sections parametric data base
- › Mixed beam/shell models

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®
- › Autodesk® Inventor™
- › CATIA® V5



Retrieve geometry and parameters from your CAD

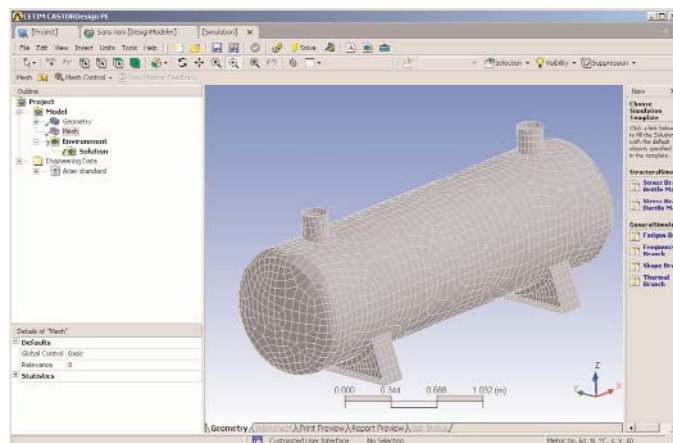
It allows you to make design changes to your CAD model without having to reapply loads and/or supports. You can either pick the CAD dimension to change directly or enhance your design iterations with the Parameter Manager.

The ANSYS Pressure Equipment Module can also be used with native Parasolid, ACIS or IGES files.

Nozzle creation tool

The ANSYS Pressure Equipment Module offers an efficient tool for creating nozzles associated with standard components. The following functions are available:

- › 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



Automatic mesh generator

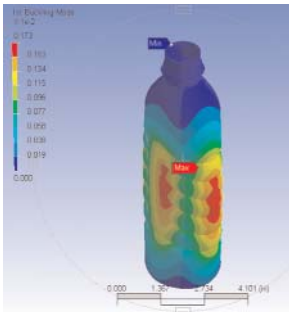
Surface model meshes are built with triangles and/or quadrilateral elements. The user has control of the following parameters:

- ▶ 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
- ▶ Choice of the element type

Materials and analysis data

Different types of data are included:

- ▶ 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



Stability analysis of pressure equipment

Stability Analysis

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

Product Features

Automatic mesh generator

- ▶ Shell, 3D volume and beam elements
- ▶ Automatic mesh generation
- ▶ Mesh quality checks

Analysis types

- ▶ Static and dynamic analysis
- ▶ Buckling analysis



Product Features

Results interpretation

- ▶ 3-D results animation
- ▶ 3-D dynamic cutting section
- ▶ Definition of results assessment criteria
- ▶ Iso-stresses and displacements
- ▶ Results query on nodes and elements
- ▶ Plot of Tresca stress criterion against the nominal calculated stresses
- ▶ Stress results analysis tool

Depending on the classification of the elements in local or global zones possibility to:

- ▶ Plot the global primary membrane stresses
- ▶ Plot the local primary membrane stresses
- ▶ Plot the total primary membrane stresses
- ▶ Plot the equivalent variation of the combined stresses
- ▶ Modify the definition of global and local zones
- ▶ Plot the bending stress alone
- ▶ Edit analysis report

Options

- ▶ CAD system integration Plug-ins
- ▶ CATIA models retrieval

Post-processing

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
- ▶ Colored scale iso-stresses
- ▶ Iso-stresses on section for beam elements
- ▶ Dynamic 3-D cutting section
- ▶ Alert warning when exceeding defined criterion
- ▶ Results animation, and AVI animation file creation

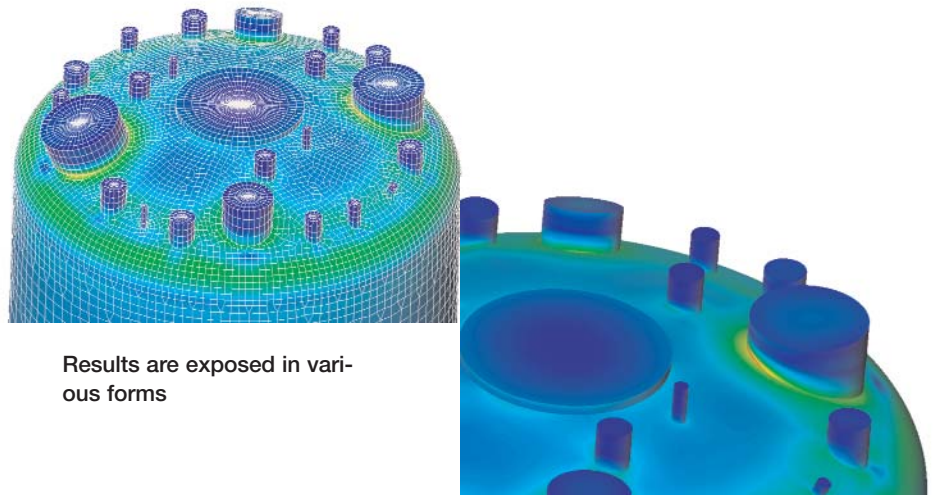
Stress results analysis tool

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, Microsoft® Word® and Microsoft® 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.



Results are exposed in various forms