HEXAGON
PPM

PV Elite

User's Guide

Version 2019 (21.0)
November 2018

Hexagon PPM
305 Intergraph Way
Madison, AL 35758

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

a. Use of a software product and Documentation is subject to the Software License Agreement ("SLA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.

b. For use of Documentation or Other Documentation where end user does not receive a SLA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee’s internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee’s internal, non-commercial use. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the SLA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

Contents

What's New in PV Elite and CodeCalc

The latest PV Elite and CodeCalc releases deliver a number of significant new and extended capabilities in response to current market requirements, as well as direct feedback from the growing PV Elite/CodeCalc user community. The following changes have been made to PV Elite and CodeCalc.

PV Elite and CodeCalc 2019 (Version 21.00.00.0000)

Code Updates

- Updated to support the KHK 2012 Japan seismic code. For more information, see *KHK 2012 Seismic Data* (page 496).
- Updated to support the IBC 2018 seismic code. For more information, see *IBC 2018 Seismic Data* (page 486).
- Updated to support the IBC 2018 wind data. For more information, see *IBC 2006, 2009, 2012, 2015, and 2018 Wind Data* (page 404).

Configuration

Input Processor & Analysis

- PV Elite now supports PD 5500:2018.
- Updated the documented table descriptions for the nozzle-to-shell weld **Fatigue Calculations**. (P3 TX:26475)
- You now have the option to force the VIII-1, 1-5 calculation when delta is greater than alpha and the cone is connected to a flange. This option is on the **General Input** tab when **Element Type** is set to **Conical**. (P4 TX:27162)

Export & Third-Party Integrations

Output Processor & Reports

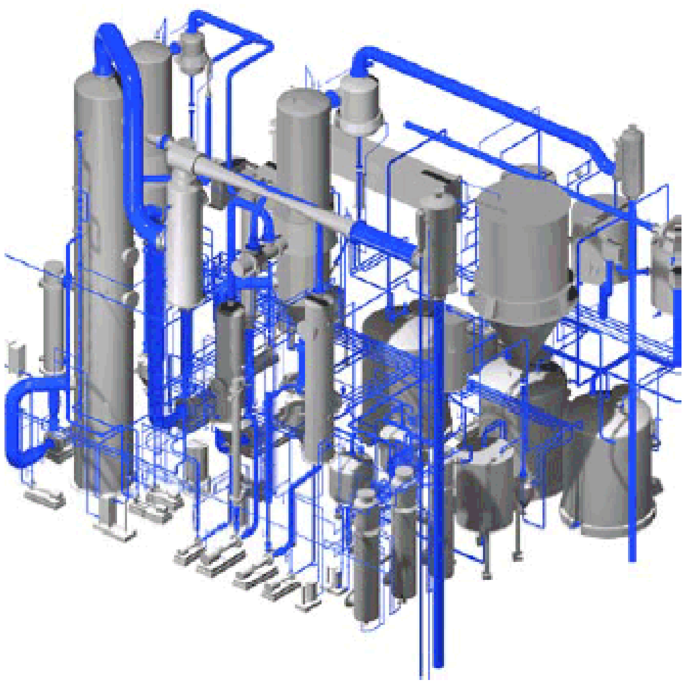
-

SECTION 1

PV Elite Overview

PV Elite consists of nineteen modules for the design and analysis of pressure vessels and heat exchangers, and assessment of fitness for service. The software provides the mechanical engineer with easy-to-use, technically sound, well-documented reports. The reports contain detailed calculations and supporting comments that speed and simplify the task of vessel design, re-rating, or fitness for service. The popularity of PV Elite is a reflection of Intergraph CADWorx and Analysis Solutions' expertise in programming and engineering, and dedication to service and quality.

Calculations in PV Elite are based on the latest editions of national codes such as the ASME Boiler and Pressure Vessel Code, or industry standards such as the Zick analysis method for horizontal drums. PV Elite offers exceptional ease of use that results in dramatic improvement in efficiency for both design and re-rating.



PV Elite features include:

- A graphical user interface allowing you to add model data while seeing the vessel elements as they are added.
- Horizontal and vertical vessels of cylinders, conical sections, and body flanges, as well as elliptical, torispherical, hemispherical, conical, and flat heads.
- Saddle supports for horizontal vessels.
- Leg and skirt supports at any location for vertical vessels.
- Extensive on-line help.

- Dead weight calculation from vessel details such as nozzles, lugs, rings, trays, insulation, packing, and lining.
- Wall thickness calculations for internal and external pressure according to the rules of ASME Section VIII Divisions 1 and 2, PD 5500, and EN-13445.
- Stiffener ring evaluation for external pressure.
- Wind and seismic data using the American Society of Civil Engineers (ASCE) standard, the Uniform Building Code (UBC), the National (Canadian) Building Code, India standards, as well as British, Mexican, Australian, Japanese, and European standards.
- A user-defined unit system.
- A complete examination of vessel structural loads, combining the effects of pressure, dead weight, and live loads in the empty, operating, and hydrotest conditions.
- Logic to automatically increase wall thickness to satisfy requirements for pressure and structural loads, and introduce stiffener rings to address external pressure rules.
- Structural load evaluation in terms of both tensile and compressive stress ratios to the allowable limits.
- Detailed analysis of nozzles, flanges, and base rings.
- Material libraries for all three design standards.
- Component libraries containing pipe diameter and wall thickness, ANSI B16.5 flange pressure vs. temperature charts, and section properties for AISC, British, Indian, Japanese, Korean, Australian and South African structural shapes.
- Thorough and complete printed analysis reports, with definable headings on each page. Comments and additions may be inserted at any point in the output.

What Distinguishes PV Elite From our Competitors?

Our staff of experienced pressure vessel engineers are involved in day-to-day software development, software support, and training. This approach has produced software that closely fits today's requirements of the pressure vessel industry. Data entry is simple and straightforward through annotated input fields. PV Elite provides the widest range of modeling and analysis capabilities without becoming too complicated for simple system analysis. You can tailor PV Elite through default settings and customized databases. Comprehensive input graphics confirm model construction before analysis is made. The software's interactive output processor presents results on the monitor for quick review or sends complete reports to a file, printer or Word document. PV Elite is an up-to-date package that not only uses standard analysis guidelines, but also provides the latest recognized opinions for these analyses.

PV Elite is a field-proven engineering analysis program and is a widely recognized product with a large customer base and an excellent support and development record.

What Can Be Designed?

PV Elite can design and analyze:

General Vessels

Enables users to perform wall thickness design and analysis of any vessel for realistic combinations of pressure, deadweight, nozzle, wind and seismic loads in accordance with ASME Section VIII Division 1 rules, Division 2 rules, PD 5500, and EN-13445. These calculations address minimum wall thickness for pressure and allowable longitudinal stress (both tension and compression) in the vessel wall for the expected structural load combinations.

Complete Vertical Vessels

Enables users to define vessels supported by skirts, legs or lugs for complete dead load and live load analysis. Stacked vessels with liquid are also addressed. Enables users to specify Hydrotest conditions for either vertical or horizontal test positions. Vessel MAWP includes hydrostatic head and ANSI B16.5 flange pressure limitations.

Complete Horizontal Vessels

Enables stress analysis of horizontal drums on saddle supports using the method of L. P. Zick. Results include stresses at the saddles, the midpoint of the vessel and in the heads.

SECTION 2

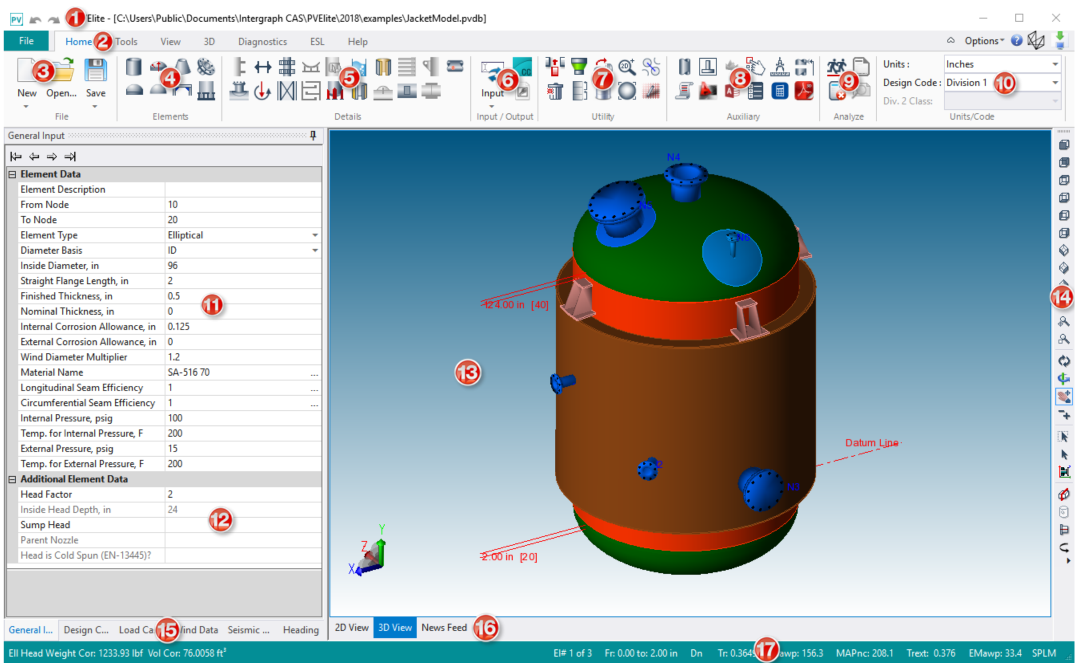
Getting Started and Workflows

This section focuses on getting started and the fundamental workflows PV Elite.

- **Input** - Enter information required to define the vessel, its service requirements, and its design guidelines.
- **Analysis** - Translate the input data with design and analysis algorithms, apply the rules of the appropriate code or standard, and generate results.
- **Output** - Present a comprehensive final report of the results.

Understanding the Interface

The main PV Elite window has a quick access toolbar in the top left corner (indicated with a red 1 in the picture below), which you can customize with the commands that you use most or use the **Undo** and **Redo** functionality. Directly below the quick access toolbar is a series of ribbons (2) and panels. On the Home tab, are data file commands (3 *File Panel* (page 50)). You can also add elements (4 *Elements Panel* (page 51)) and details (5 *Details Panel* (page 52)) to the current element.



PV Elite breaks a vessel into an assemblage of individual elements—heads, shells, cones, body flanges, and/or a skirt—and the components on these elements. A quick look at the screen above shows the data (11) defining one element in the graphic view (13). Except for **From Node** and **To Node**, the data is common to all vessel wall thickness calculations. The **From Node** and **To Node** inputs are necessary to assemble the individual elements into the complete vessel and are automatically assigned by PV Elite. A complete vessel is required if all dead and live loads are to be included in the design or analysis. However, PV Elite will run wall thickness calculations on elements without constructing the entire vessel.

The body of the screen contains either two or three areas - a table of the Element Data (11), a table of the Element Additional Data (12) and the graphic area which contains an image of the current status of the entire vessel or the current element (13). A status bar displays (17) across the bottom of the screen and displays the element count, the position and orientation of the current element, quick internal pressure calculations for the current element.

When you click in the data areas (11 and 12), the **Tab** key moves the highlight (and control) through its input cells. In most element data areas, press **Enter** to register the data and move the focus to the next field. The exception is at combo boxes where clicking the arrow displays the available choices. Throughout the software, **F1** displays help for the highlighted data item. After you are familiar with these screen controls, a combination of mouse and keystroke commands will provide the most efficient navigation through the software.

Some of the data input in PV Elite is controlled through a data grid (11 and 12). To enter the data, click the mouse on the data text, such as **Inside Diameter**, and type the input value. The cursor will not blink over the numeric/alphanumeric values until typing has begun. After the data is entered, press **Enter** or **Tab** to proceed. The arrow keys can also be used to navigate between the input fields.

NOTE The right mouse button is used to select vessel details on the vessel graphic. Combo boxes have the down arrow button at the right end of the input cell.

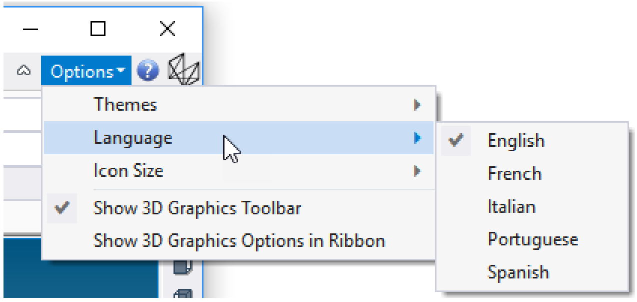
When the 3D View (13) is active, a few more keys are available. No special highlight appears, but the string PgUp/PgDn/Home/End displays at the bottom graphics area. This indicates that these keys are now active. The image in the graphics area shows the current state of the input for the vessel model with its elements and the details on these elements. Switch between 2D and 3D views using the tabs at the bottom of the screen (16).

When the 2D View is active, one of the elements is highlighted. The **Element Data** and **Element Additional Data** sections (11 and 12) define this element. By pressing **Page Up** or **Page Down**, the highlight changes from one element to the next through the vessel. Press the **Home** and **End** keys to move the highlight to the first and last elements in the vessel. Also, you can click the left mouse button while selecting the element to highlight it. After an element is highlighted, detail information for that element may be accessed. With the mouse, click the right mouse button for the existing detail image to display. To add details to the current element, click the appropriate detail on the **Details** panel (5) and enter the necessary data.

The News Feed contains product information, such as the latest product version. In addition, you can refer to the page for upcoming events, product training opportunities, and future webinars. You can use the quick icon links at the bottom to get to the product web pages, the latest newsletter/blog postings, and Intergraph CAS social media sites.

Set Interface Language

The PV Elite interface is available in several languages. Use **Options > Language** to select the language that you want to use.



La interfaz de PV Elite está disponible en varios idiomas. Utilice **Opciones> Idioma** para seleccionar el idioma que desea utilizar.

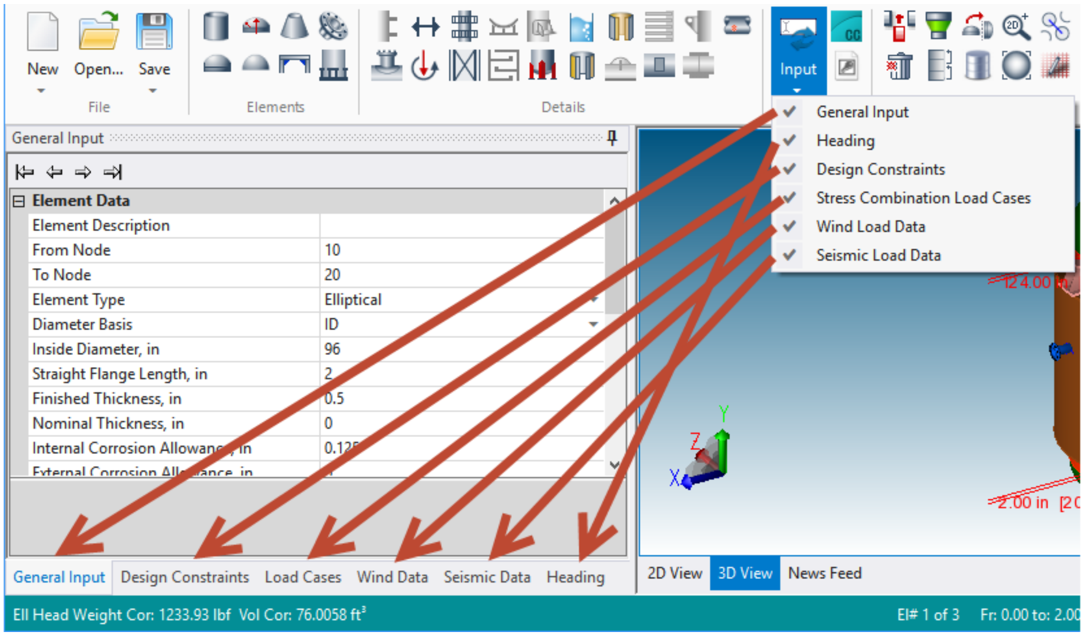
L'interface PV Elite est disponible en plusieurs langues. Utilisez **Options> Langue** pour sélectionner la langue que vous souhaitez utiliser.

L'interfaccia PV Elite è disponibile in diverse lingue. Utilizzare **Opzioni> Lingua** per selezionare la lingua che si desidera utilizzare.

O interface de PV Elite está disponível em vários idiomas. Use **Opções> Idioma** para selecionar o idioma que você deseja usar.

Input Processors

The **Input** items are used to define the other types of data that might be necessary for an analysis: design constraints, report headings, general input data, and live (**Wind Loads** and **Seismic Loads**) load definitions. Each command activates an input data tab in the bottom-left corner of the window. You can re-order the tabs as you like. The **Design Constraints** data is important because this is where the overall analysis for this vessel is defined and controlled.



The **Component Analysis** option allows you to enter data and analyze without building a vessel. These are Intergraph's CodeCalc analysis modules, some of which are not incorporated directly into PV Elite. CodeCalc, Intergraph's popular vessel component analysis package is included in PV Elite.

Status Bar

The status bar across the bottom of the window displays information about your vessel in real time. Values that display in red need attention. You can show or hide the status bar using the Toggle Status Bar option on the **View** tab.

Wgt Cor

Displays the corroded weight.

Vol Cor

Displays the corroded volume.

t/L

Displays the thickness to length ratio.

OutDepth

Displays the head depth for a torispherical head.

El#

Displays the current element location relative to the start of the vessel. For example, a value of **4 of 11** means the active element is the fourth element from the start of the vessel and that there are eleven elements in total that define the vessel.

Fr

Displays the starting and ending distances from the Datum point for the current element.

Left / Right / Up / Down

Displays the element orientation.

Tr

Displays the computed thickness for the internal pressure.

Tmin, as

Displays the retirement limit minimum thickness at the seam.

Tmin, afs

Displays the retirement limit minimum thickness away from the seam.

Mawp

Displays the maximum allowable working pressure of the vessel and flange.

MAPnc

Displays the maximum allowable pressure in a new and cold condition.

Trext

Displays the computed thickness for the external pressure.

EMawp

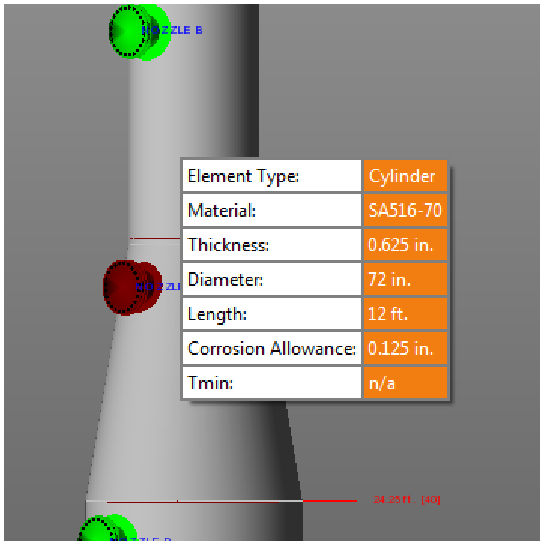
Displays the external maximum allowable working pressure.

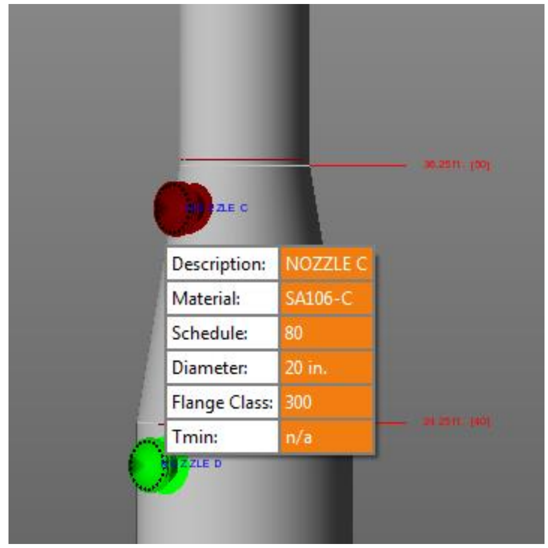
Slen

Displays the maximum length between ring stiffeners. When **>>>** displays, no reinforcing ring is required for this element for external pressure purposes. For more information, see Stiffening Ring.

Tooltip

When you press the CTRL key and hover your mouse over an element or detail on the 3D View tab, PV Elite displays a tooltip that contains a table with data about the element or detail. The tooltip table contains information such as, the element/detail type, material name, thickness, diameter, and so forth.



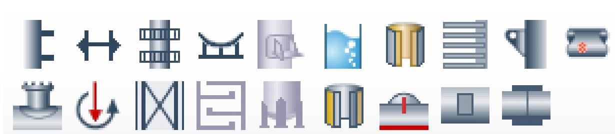


After analyzing a file, when you display the tooltip, the **Tmin** row displays the calculated minimum thickness of the element/detail as well as the element which governs the MAWP.

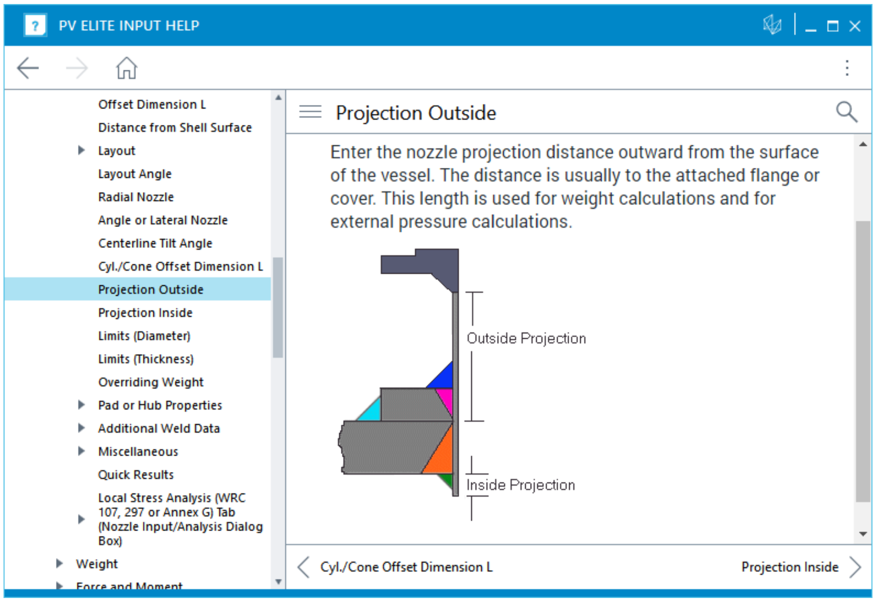
Adding Details

With the elements defined, enough information exists to run through the pressure calculations but the total vessel weight is not yet set. Much of this information is specified as element details. Nozzles, insulation, operating fluid, platforms and the like are all entered as details on the various elements. PV Elite calculates the weight of each of these items and account for them in the various analyses. Details such as saddles, lugs and legs are also used to locate support points on the vessel—important data for load calculations.

Details can only be specified on the current element. To enter the first detail, highlight (make current) the element that will hold the detail, and click the appropriate **DETAIL** command. Allowing the cursor to rest on the toolbar button displays a tool tip definition of the button. Select the detail and enter the data in the screen that follows. For more information, see *Details Panel* (page 52).



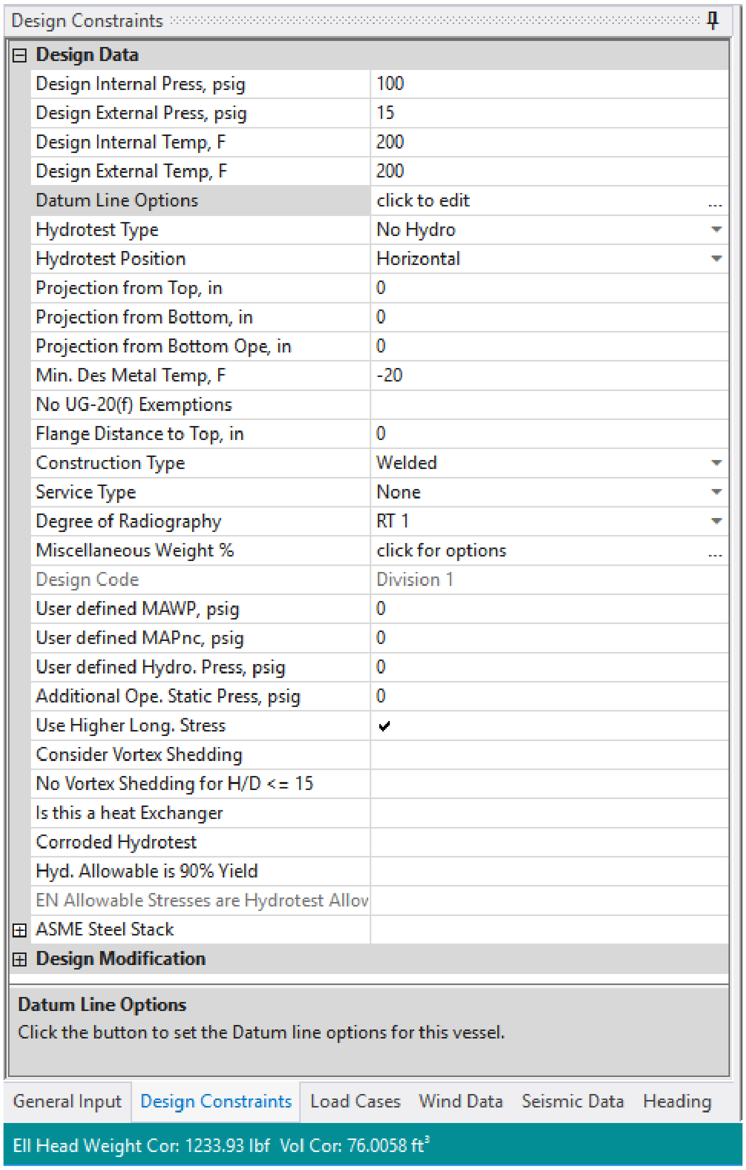
Use the **Help** button on the detail screen or press [**F1**] to learn more about the requested data. Define all details necessary to develop the proper total vessel load.



Specifying Global Data - Loads and Design Constraints

Although default values allow the analysis to proceed, other data should be set before the analysis continues. These data are the required live loads and design constraints, and the optional vessel identification and report headings. These data are accessed and entered through the **Input** panel on the **Home** tab. The **Headings** input allows the specification of three lines of data, which appears at the top of each page in the printed output. The heading data also includes title page entry, which appears at the beginning of the input echo report.

Select **Input > Design Constraints** and then select the **Design Constraints** tab to display the design data.



Design Data includes vessel identification along with items that affect the design and analysis of the vessel; items such as type of hydrostatic testing and degree of radiographic examination appear here. It is important to note that this is where the design code is set - Division 1, Division 2, PD 5500 or EN 13445.

The **Design Modification** area holds four inputs that control the redesign of the vessel should the user-entered wall thickness be insufficient for the analyzed loads. If a box is checked, the software increase the element's wall thickness so that it meets or exceeds the requirements for that load category. There are four boxes for three load types: one box for internal pressure, two boxes for external pressure (either increase the wall thickness or locate stiffener rings along the vessel to satisfy the buckling requirements), and one box for the variety of structural loads that develop longitudinal stresses in the vessel wall. The software provides the option of rounding up a required thickness to a nominal value (such as the next 1/16 inch or 1 mm) in the *Configuration* (page 224) dialog box.

The **Load Cases** tab displays nineteen default structural load cases for the analysis. These cases cover the extent of structural loads on the vessel wall. Each case contains a pressure component (axial) 1, a weight component (both axial and bending), and a live load component (bending). The axial stresses are combined with the bending stresses to produce a total stress in the vessel wall. Both tensile and compressive stresses are compared to their allowable limits. Refer to the table below for a definition of terms used in the **Load Case** input.

NOTE These pressure calculations should not be confused with those used for the wall thickness requirements defined in ASME Section VIII and PD 5500. Here, internal and hydrostatic pressures are used to calculate a longitudinal, tensile stress in the vessel wall and the external pressure a similar compressive stress in the wall.

Pressure	**Weight**	**Live Load**
NP - No Pressure	EW - Empty Weight	WI - Wind
IP - Internal Pressure	OW - Operating Weight	EQ - Earthquake
EP - External Pressure	HW - Hydrostatic Weight	HI - Wind at Hydrostatic Weight conditions
HP - Hydrostatic Pressure	CW - Empty Weight No CA	HE - Earthquake at Hydrostatic Weight conditions
		VF - Vortex Shedding Filled
		VO - Vortex Shedding Operating
		VE - Vortex Shedding Empty
		WE - Wind Bending Empty New and Cold
		WF - Wind Bend Filled New and Cold
		CW - Axial Weight Stress New and Cold
		FS - Axial Stress, Seismic
		PW - Axial Stress Wind

Nozzle Design Options are used to set the overall pressure requirements for the nozzles on this vessel and also to include the maximum allowable pressure - new and cold (MAP nc) case in the nozzle checks. The **Installation | Misc. Options** option displays a screen to specify where certain vessel details will be added - either at the fabrication shop or in the field. This data is used to properly set the detail weights for the empty and operating conditions.

Wind and earthquake information is entered on the **Wind Data** or **Seismic Data** tabs. PV Elite generates live loads based on the criteria established by one of many standards, including the American Society of Civil Engineers (ASCE), the Uniform Building Code (UBC), the (Canadian)

National Building Code (NBC), and the Indian National Standard. Wind loads can also be specified directly as a wind pressure profile. PV Elite references these codes for live loads only. ASME Section VIII or PD 5500 rules apply for all other calculations. The screen below shows the data required for the default codes. PV Elite uses these criteria to set the magnitude of the live load and bending moment on each element of the vessel.

After the element, detail, and global data is entered and checked, the model is ready for error processing and analysis.

Error Checking

The **Input Processor** makes many data consistency checks during the input session. For example, the processor creates an error message if you try to specify a nozzle 20 feet from the bottom of a 10-foot shell element. However, not all data can be confirmed on input so a general error processor is run prior to the analysis. This error processor can be run as a stand-alone from the **Analyze** panel, *Error Check Only* (page 221) .

In addition to the notes that are presented on the screen during error checking, these error messages appear in the output report and are accessible through the output review processor.

NOTE As with all engineering and designing, the vessel analyst must use common sense to insure the model is basically correct. This is a great advantage of the 3D graphics as it reveals obvious errors.

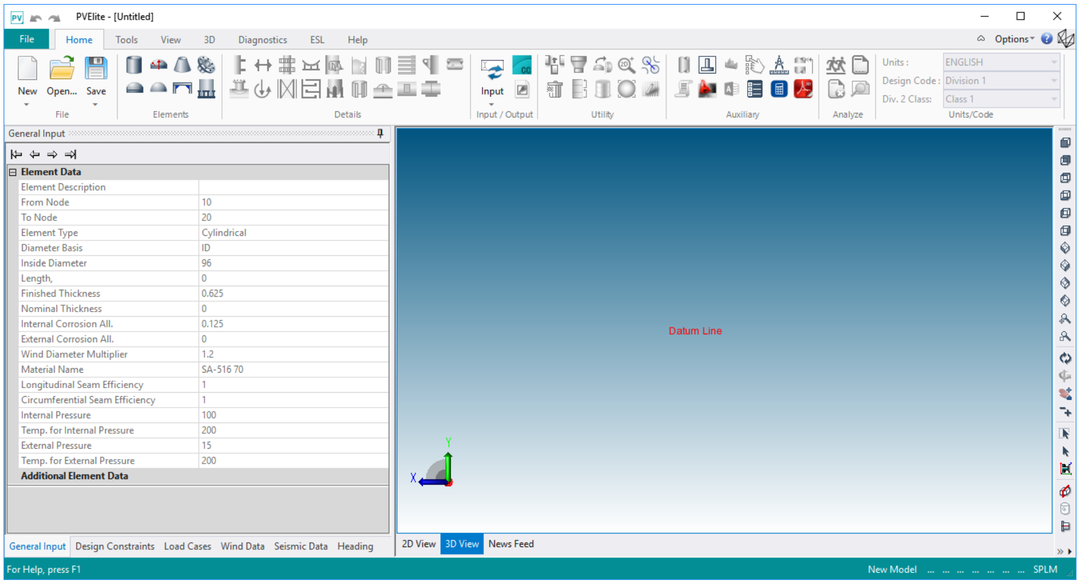
Modeling Basics

PV Elite breaks a vessel into an assemblage of individual elements—heads, shells, cones, body flanges, and/or a skirt—and the components on these elements. Vessels are defined one element to the next - from bottom to top for vertical vessels and from left to right for horizontal vessels.

Before starting PV Elite, most users collect the necessary data for the vessel design or analysis. Collecting data to define these elements before starting the program is not required but it will make the most efficient use of the designer's time. Typical input items include actual or proposed values for vessel material, inside diameter, operating temperatures and pressures, wind and seismic site data, nozzle and ring location to name a few. If necessary, the input processor can be terminated at any time and restarted later if any missing data need be collected. With the program's graphic display of the vessel input, it is easy to recall the current state of an unfinished model or identify where data is missing or incomplete.

1. Start PV Elite by clicking the icon on the desktop or selecting the item from **Start** > **All Programs > Intergraph CAS > PV Elite > PV Elite**.

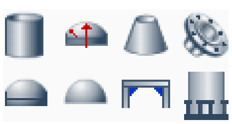
*PV Elite starts with a **Vessel Input** screen for the job called "Untitled."*



Defining the Basic Vessel

You should build vertical vessels from bottom to top and horizontal vessels from left to right. It is not necessary to build an entire vessel if only the thickness for pressure is needed. The Element Data (the grid on the left side of the interface) must be specified before the first element can be placed on the screen.

1. Start with the **Inside Diameter**, as both the **Node Numbers** and the **Element Type** are set by using the **Element** toolbar.



2. After the Element Data is entered, elements are quickly assembled one after another by clicking the **Element** toolbar and making any changes to the Element Data.

The complete vessel is created from the following elements (in their toolbar order):

- Shell
- Elliptical Head
- Torispherical Head
- Spherical Head
- Cone
- Welded Flat Head
- ANSI/Bolted Flange
- Skirt

If the vessel begins with a skirt element, it is a vertical vessel. Vertical vessels on legs and horizontal vessels start with a head element.

NOTE If that first head element is improperly oriented for the vessel in mind (horizontal or vertical), click **Flip Element Orientation** on the **Home** tab, **Utility** panel to correct the orientation. Later, if heads, body flanges, or cone elements show incorrect orientation, click **Flip Element Orientation** to fix the orientation.

After the second element is added, use the **Flip Model Orientation** on the **Tools** tab to flip the entire model flip between horizontal and vertical.

From Nodes and **To Nodes** values are automatically assigned by the software; they start with node 10 and are incremented by 10 throughout the model. The element data set at the beginning of the session carries forward from one element to the next. Any data changes on the last element carry forward onto any new elements that are added. The element data displayed belongs to the highlighted element in the vessel image.

1. Use the mouse to change the highlighted and displayed element by clicking on the element of interest.

   Data may be updated one element at a time but there are more efficient ways to change an item through several elements; for example, if the circumferential weld joint efficiency for the skirt (from node 10 to 20) is set at 0.7. If this value was not changed to 1.0 on the bottom head as it was created, this (incorrect) value is carried from one element to the next in the Build Mode to the top of the vessel element (such as, From Node 50 and To Node 60). In this situation, it is easiest to change the data on the bottom head element (20 to 30), and then click **Share** to "share" this item through the elements in the list with From Node 30 through From Node 50. Certain data is automatically "shared". Inside diameter, for example is automatically changed for all elements (stopping at cones) attached to the element where the change occurs. Some changes to the element data do not immediately appear on the vessel image.

2. To refresh the image, press **F5**.

Building a Heat Exchanger

This section provides the workflow of how to build a heat exchanger using PV Elite.

1. Launch **PV Elite**.

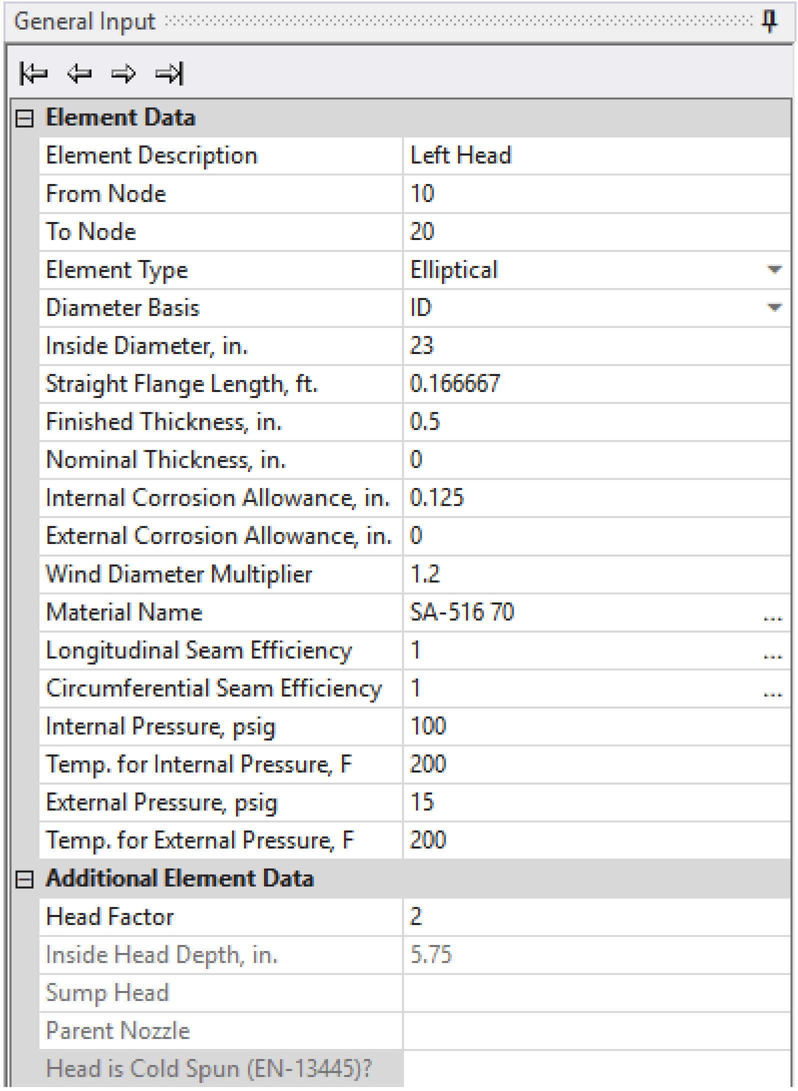
   *The **General Input** screen displays.*

2. Select an ellipsoidal head by clicking **Ellipse** .

3. Click **Flip Elements Orientation** on the **Home** tab to build the heat exchanger in the horizontal orientation.

4. Enter the information for the head exactly as shown below. Check your input before you move on. Remember to click on the text in the left column and then start typing. The cursor automatically moves to the right column for you to begin typing. Pressing **[F1]** displays the corresponding help information for that input.

The screen should then look exactly like this when you are finished with your input.

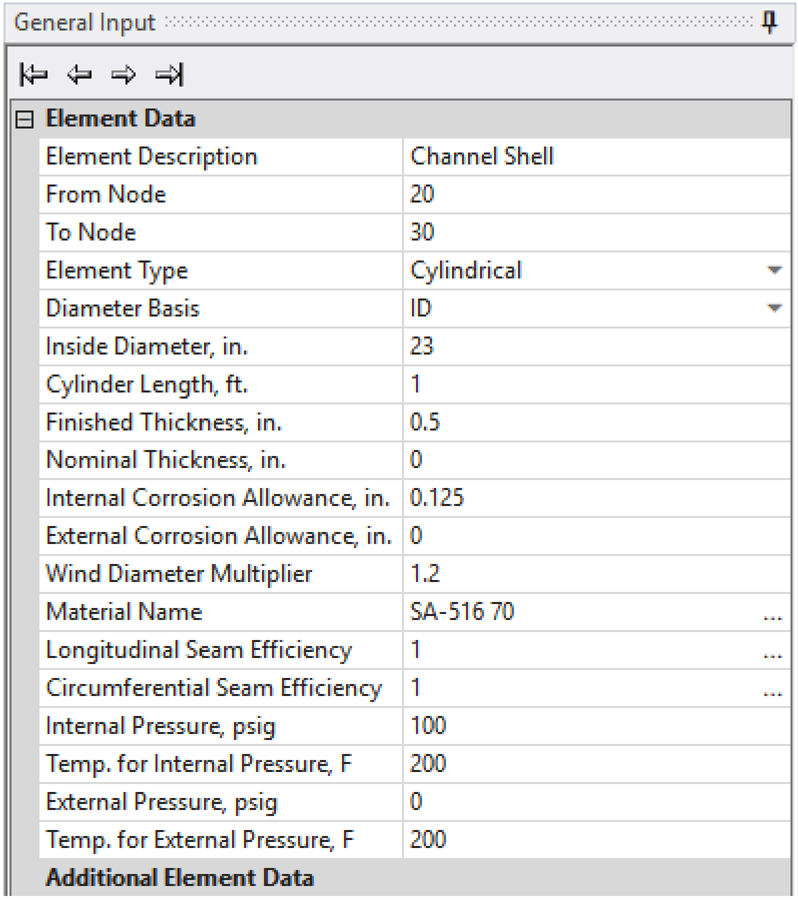


General Input

Element Data	
Element Description	Left Head
From Node	10
To Node	20
Element Type	Elliptical
Diameter Basis	ID
Inside Diameter, in.	23
Straight Flange Length, ft.	0.166667
Finished Thickness, in.	0.5
Nominal Thickness, in.	0
Internal Corrosion Allowance, in.	0.125
External Corrosion Allowance, in.	0
Wind Diameter Multiplier	1.2
Material Name	SA-516 70
Longitudinal Seam Efficiency	1
Circumferential Seam Efficiency	1
Internal Pressure, psig	100
Temp. for Internal Pressure, F	200
External Pressure, psig	15
Temp. for External Pressure, F	200
Additional Element Data	
Head Factor	2
Inside Head Depth, in.	5.75
Sump Head	
Parent Nozzle	
Head is Cold Spun (EN-13445)?	

5. When you are finished typing all the data, press **Enter** twice.
6. Click **Cylinder**  to add a cylinder to the head.

The screen should then look exactly like this when you are finished with your input.

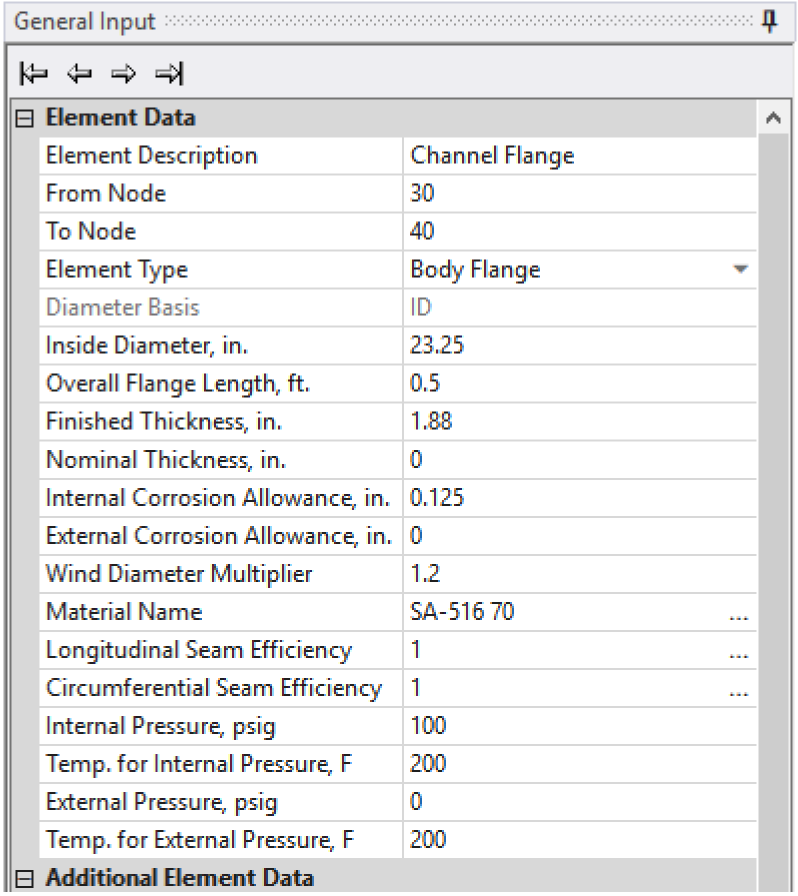


General Input

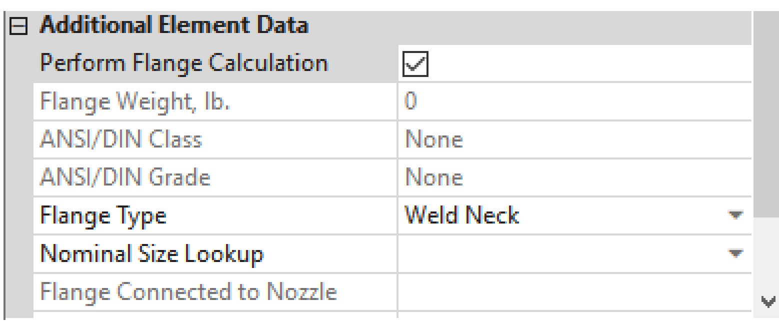
Element Data	
Element Description	Channel Shell
From Node	20
To Node	30
Element Type	Cylindrical
Diameter Basis	ID
Inside Diameter, in.	23
Cylinder Length, ft.	1
Finished Thickness, in.	0.5
Nominal Thickness, in.	0
Internal Corrosion Allowance, in.	0.125
External Corrosion Allowance, in.	0
Wind Diameter Multiplier	1.2
Material Name	SA-516 70
Longitudinal Seam Efficiency	1
Circumferential Seam Efficiency	1
Internal Pressure, psig	100
Temp. for Internal Pressure, F	200
External Pressure, psig	0
Temp. for External Pressure, F	200
Additional Element Data	

7. Click **ANSI/Bolted Flange**  to add a body flange to the right hand end of the channel shell.

8. After adding the flange, enter all the values exactly as shown below.



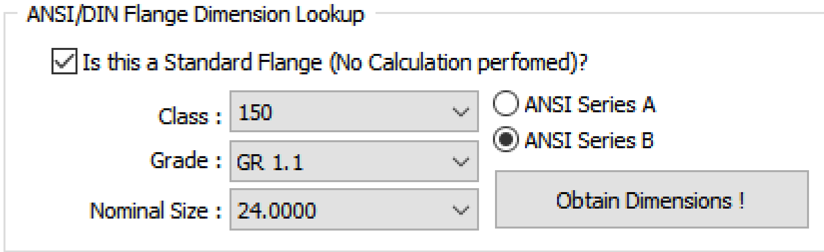
9. Select the **Perform Flange Calculation** check box.



*The **Flange** dialog box displays.*

You must correctly dimension the flange. To do this, change the flange as it appears to a 24 inch Class 150 flange, which will fit into the heat exchanger.

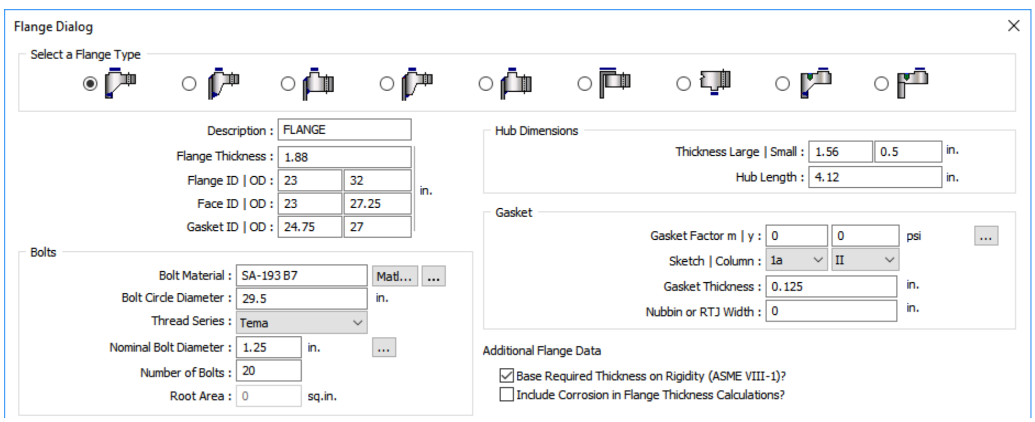
10. At the bottom of the **Flange** dialog box you see a section that resembles this:



11. Select the **Is this a Standard Flange (No Calculation performed)?** check box.
12. Select **150** as the **Class**
13. Select **24** for the **Nominal Size**.
14. Click **Obtain Dimensions !**.

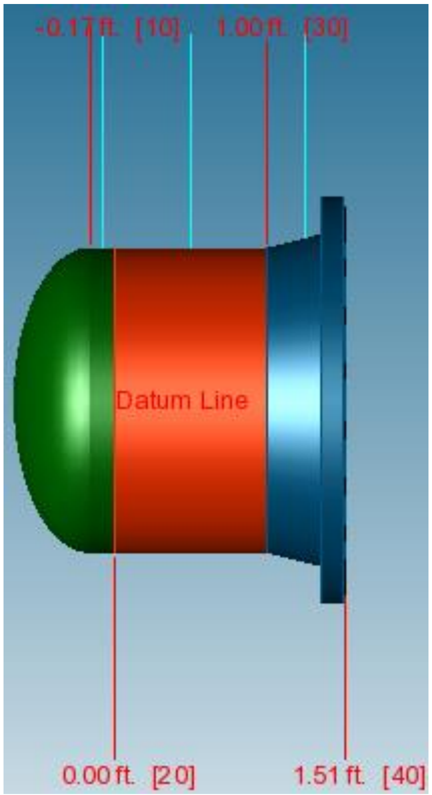
    The Flange screen is now set up for the 24 inch Class 150 dimensions and bolting.

15. Verify your screen looks like the following figure:



16. Click **OK** on the **Flange Dialog**.
17. Click **Save**  to save your work. Use **My First Model** or something similar for the file name.

18. Your model should look similar to this.



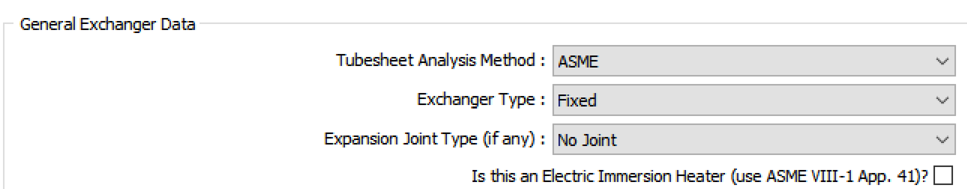
Up to this point, you have been using the normal PV Elite vessel building techniques for non-heat exchanger pressure vessels. You are now ready to start modeling the heat exchanger main elements, the tubesheets, tubes, and main shell that enclose the tube bundle.

1. Click **Tubesheet Analysis** .

   *The **Heat Exchanger Tubesheet Input** dialog displays.*

   You will now construct an ASME Division 1 exchanger, which requires a large amount of input data.

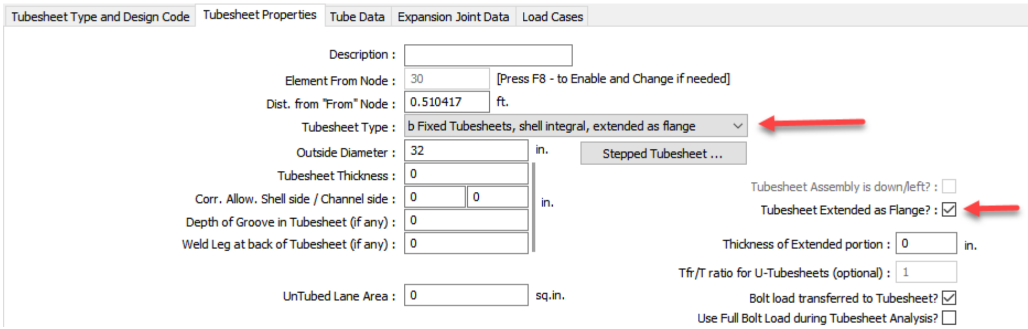
2. Select **Fixed** for **Exchanger Type**:



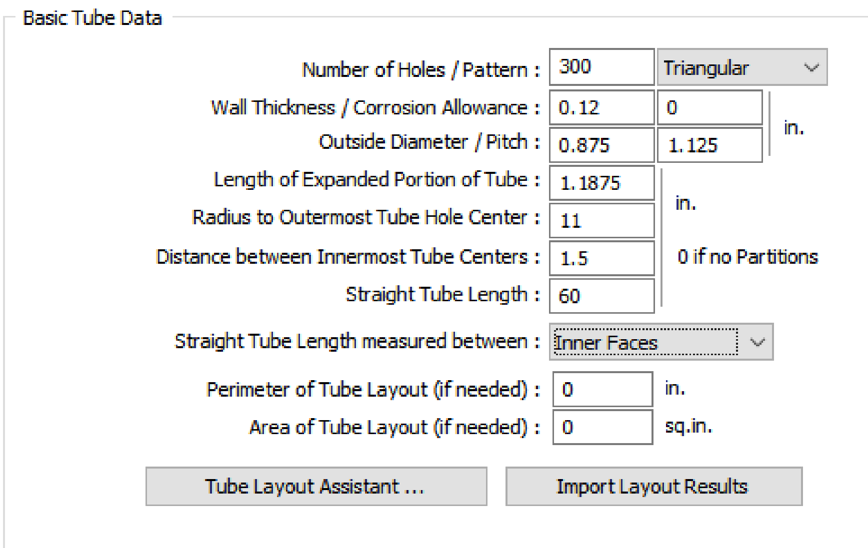
3. Select the **Tubesheet Properties** tab.

4. For **Tubesheet Type**, select **b Fixed Tubesheet, shell integral, extended as flange**.

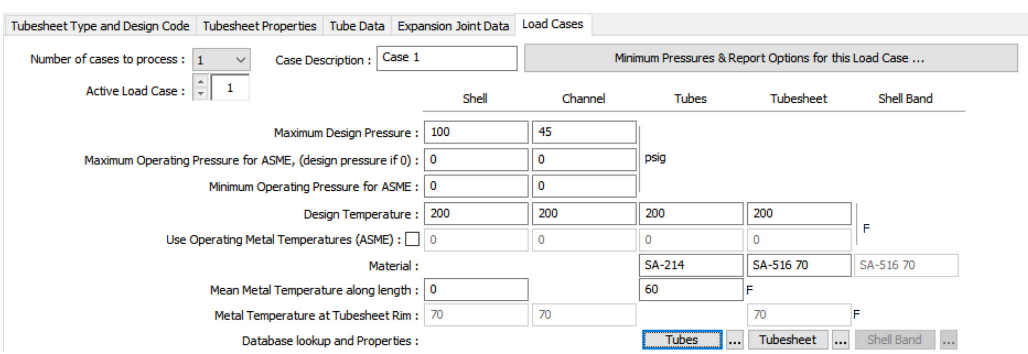
5. Select **Tubesheet Extended as Flange?**



6. Select the **Tube Data** tab and enter the information as shown below.



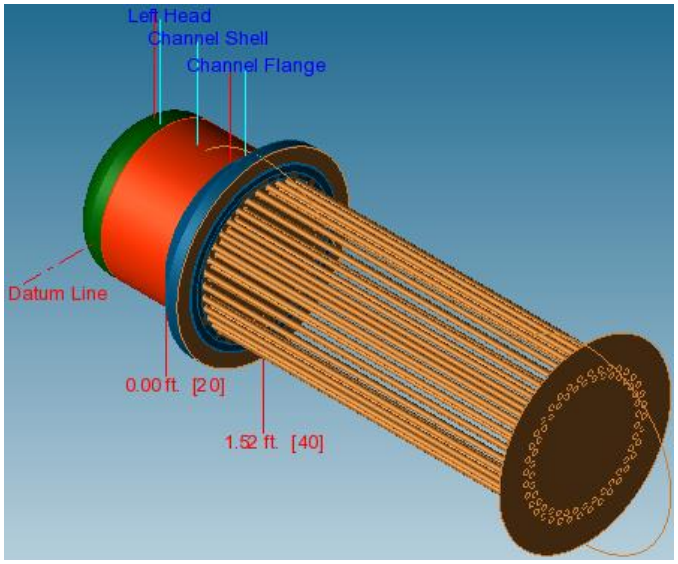
7. Select the **Load Cases** tab and enter the pressures and temperatures for the heat exchanger to complete the tube sheet and tubes data. You can enter multiple combinations of pressures and temperatures for heat exchangers.



8. Click **OK**.

9. Near the bottom-center of the window, select the **3D View** tab.

Look at the 3D model on your screen, and it should resemble this figure.



NOTE There are only two rows of tubes displayed. Because of the intensive nature of 3D graphics it is impractical to show hundreds of tubes.

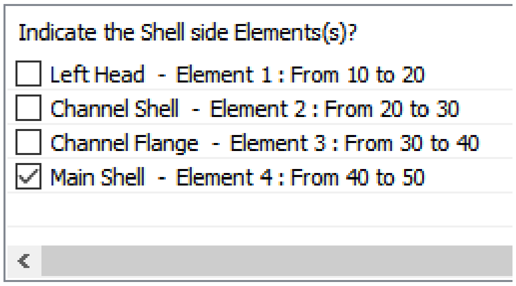
Notice that there is no cylindrical shell between the two tubesheets. PV Elite cannot perform tubesheet analysis unless the shell is present because the thermal load from the shell is needed to complete this analysis. You must add the shell between the tubesheets. Before you do this, recall that that the tubes are 60 inches long in the **Heat Exchanger** dialog. This is the distance between the tubesheets. So for a good match, the outer shell must also be 60 inches long.

1. Click **Cylindrical** to add a cylindrical shell to your model. This shell is 60 inches or 5 feet long as discussed above. Verify your entries match those below:

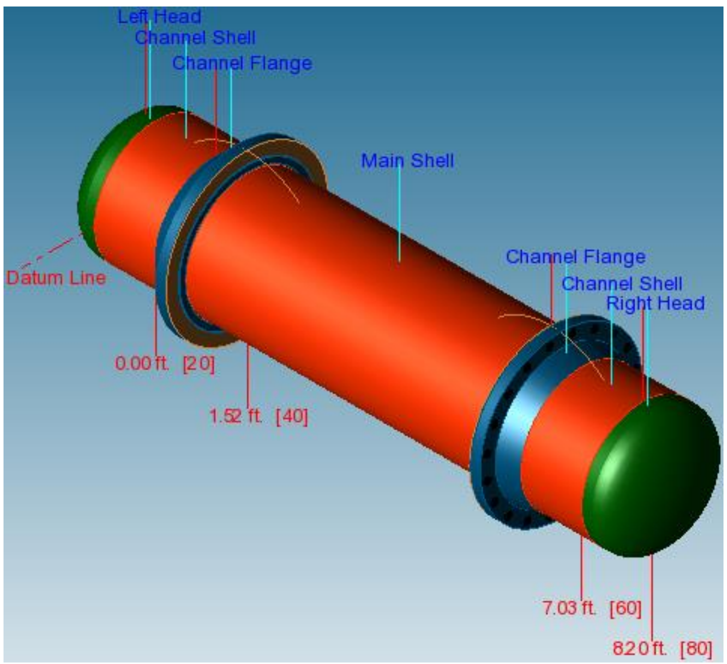
General Input

Element Data	
Element Description	Main Shell
From Node	40
To Node	50
Element Type	Cylindrical
Diameter Basis	ID
Inside Diameter, in.	23.25
Cylinder Length, ft.	5
Finished Thickness, in.	0.5
Nominal Thickness, in.	0
Internal Corrosion Allowance, in.	0.125
External Corrosion Allowance, in.	0
Wind Diameter Multiplier	1.2
Material Name	SA-516 70
Longitudinal Seam Efficiency	1
Circumferential Seam Efficiency	1
Internal Pressure, psig	100
Temp. for Internal Pressure, F	200
External Pressure, psig	0
Temp. for External Pressure, F	200
Additional Element Data	

2. Click **Tubesheet Analysis** again and go to the **Tubesheet Type and Design Code** tab.
3. Select the **Main Shell** check box, and then click **OK**.



4. All that remains is to add the body flange to the right end of the heat exchanger, then another channel shell and the final right channel head. By doing so, your model should look like this.



5. For the remaining steps, add other details to the elements using the commands on the **Detail** toolbar. You can add details such as saddle supports, nozzles, and loads.

Analysis

PV Elite can be used to confirm a safe design for a proposed or existing vessel. The program also provides direct design capabilities with which the wall thickness of individual elements is increased to meet the code requirements for internal and external pressure and longitudinal stress from a variety of dead and live loads. Whether or not the program changes wall thickness during the analysis is controlled through a **Design/Analysis Constraint** specification under **Design Modification**. For more information see *Design Constraints Tab* (page 352).

A simple analysis run (no design) occurs when the flags under **Design Modification** are all unchecked. If any of these boxes are checked, the program automatically increases the wall thickness until the constraint is satisfied. Your input in the resulting output report is automatically updated to reflect any changes made during the analysis. In addition to wall thickness, a fourth flag can be set - **Select Stiffener Rings for External Pressure**. In this case, rather than increasing the wall thickness, stiffener rings are located along the vessel to satisfy the external pressure requirements. As with the wall thickness changes, these stiffener rings are added to the model input for this analysis.

PV Elite analyzes each element to determine the required wall thickness for internal and external pressure based on the Section VIII Division 1 rules, Division 2, PD 5500 or EN-13445 rules. The program then calculates the longitudinal stresses in the wall due to four categories of vessel loads: pressure, deadweight, deadweight moments from vessel attachments or applied loads, and moments due to the live loads - wind and earthquake. These four categories are set for three different load conditions: empty, operating, and hydrotest. The sensible combination of these various categories and conditions produce the default set of 19 load cases that are found in the **Design/Analysis Constraints** processor. For each load case, PV Elite will calculate the maximum longitudinal stress around the circumference of the elements and compare these values to the allowable stress for the material, both tensile and compressive. If stresses in the vessel wall exceed the design limits, PV Elite proceeds according to the design modification settings in the input.

After the software finishes a pass through the analysis, a check is made for any program design modifications. If PV Elite changed any data, then the program automatically re-runs the complete analysis to review the impact of the changes.

There are several additional analysis controls that should be reviewed here. These controls, however, are more general in nature and are not defined for the individual job. Instead, these seven computational control directives are set for all jobs executed in the Data sub-directory. These controls are viewed and modified on the **Tools** tab, **Set Configuration Parameters** , for more information, see *Configuration* (page 224).

Performing the Analysis

In the **Analyze** panel on the **Home** tab, are two options: **Error Check Only** and **Analyze**. Use the **Error Check Only** option immediately after any questionable data is entered. **Analyze** automatically performs an error check before the analysis starts. Comments from an error check can be examined using *Review Reports* (page 222).

NOTE Errors must be corrected before the analysis can proceed.

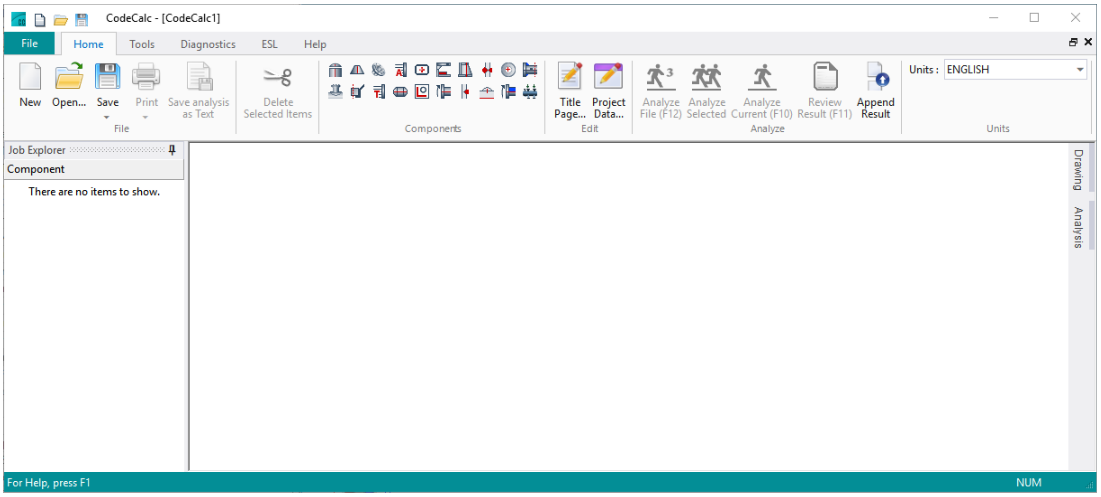
As the analysis proceeds, PV Elite displays the step or component being analyzed. If any *Design Modification (Design Constraints Tab)* (page 362) were set, PV Elite resets the thickness to the necessary value and exports these increased thicknesses to all output reports and in all other calculations. For example, if the user-entered wall thickness of 1/2 inch is insufficient for the load and the design flag is enabled, the program will calculate the required thickness (for example, 5/8 inch) and replace the user-entered input value (1/2) in the output report with the calculated required thickness (here, 5/8). The program does not change the original model data. PV Elite checks the element wall thicknesses for the various pressure cases (internal, external, and hydrostatic) and then assemble the axial and bending loads to construct each load case defined in the Global Design data. PV Elite also calculates the longitudinal stress on both sides of the vessel (for example, both *windward* and *leeward* for loads with wind) and compare the calculated stresses with the allowable stresses, both tensile and compressive. PV Elite displays the *windward* or *leeward* side stress, which is closest by ratio to the allowable limit, again either tension or compression.

After the analysis is complete, the *Review Reports* (page 222) processor displays the results of the analysis on the screen.

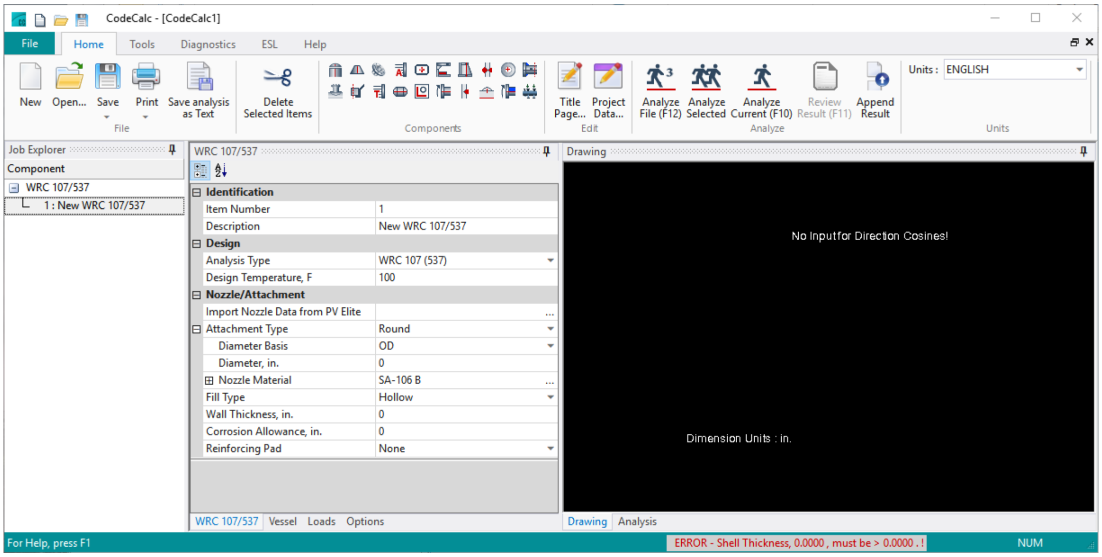
Design and Analysis of Vessel Details

At this point in the analysis the vessel details have been defined only so that their weights could be included in vessel calculations. With the structural analysis of the vessel complete and the wall thickness set, vessel details can be evaluated.

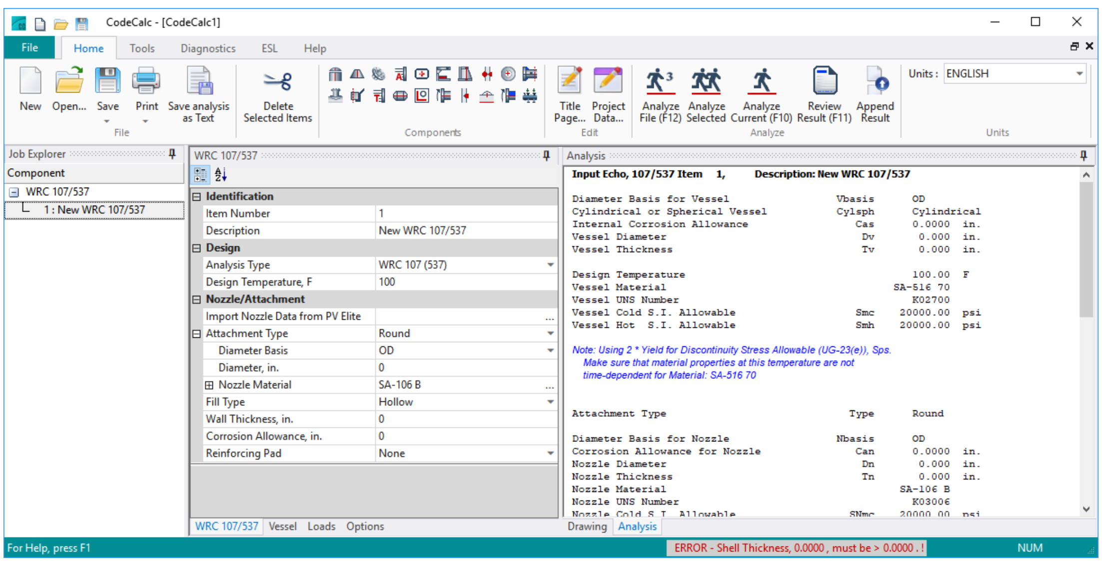
To access the input processor for these vessel details, click **Component Analysis** on the **Home** tab. This activates CodeCalc. The component selection is available on the **Home** tab.



WRC 107/537 Input Screen



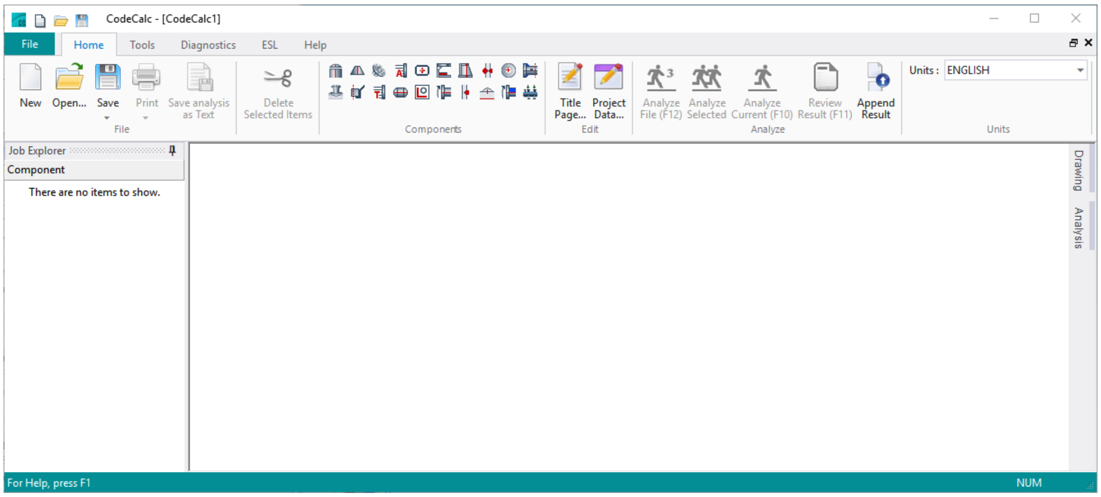
WRC 107/537 Results



Analyzing Individual Vessel Components Details

PV Elite provides for the analysis of a variety of vessel components that are not included in the overall vessel analysis: Appendix Y Flanges, Floating Heads, Lifting Lug, Pipe & Pad, WRC 107/537 and 297, Thin Joints, Thick Joints, ASME Tubesheets, TEMA Tubesheets, Halfpipe Jackets, Large Openings, and Rectangular Vessels.

To access the input processor for these vessel details, click **Component Analysis** on the **Home** tab. This activates CodeCalc. The component selection is available on the **Home** tab.

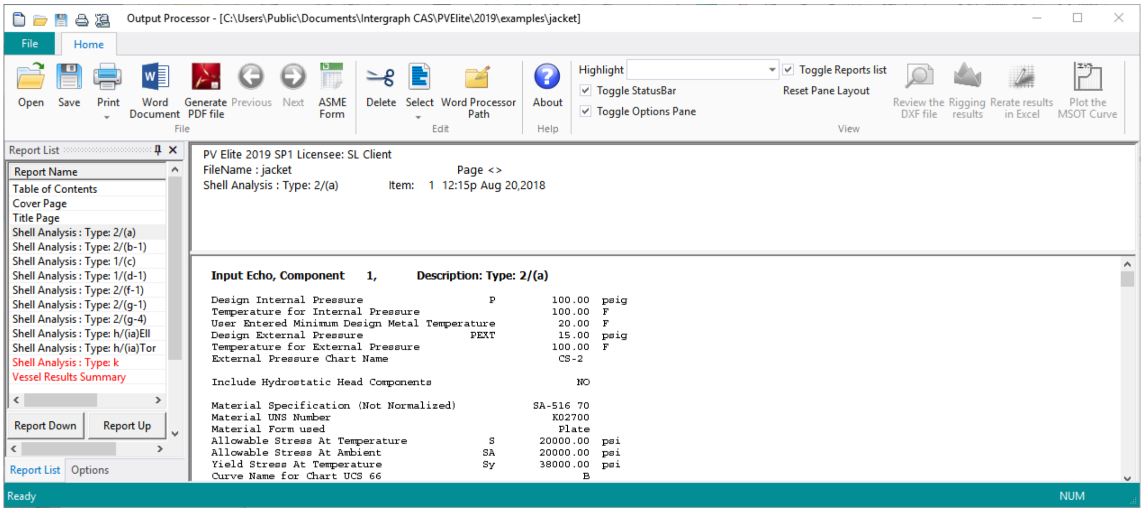


Output Review and Report Generation

Output from PV Elite analysis is stored in a binary data file that has the same name as the input file but with a ".T80" extension. Use *Review Reports* (page 222) to review every report contained in the output from input echo through stress reports.

Select the report to view from the **Report List**. Reports can be reviewed on the screen, sent to a printer, or sent to a file for review later.

Each analysis module creates its own report in the output data file. Most of the reports take the form of tables with the rows related to the elements and the columns holding the values such as thickness, MAWP, and stress.



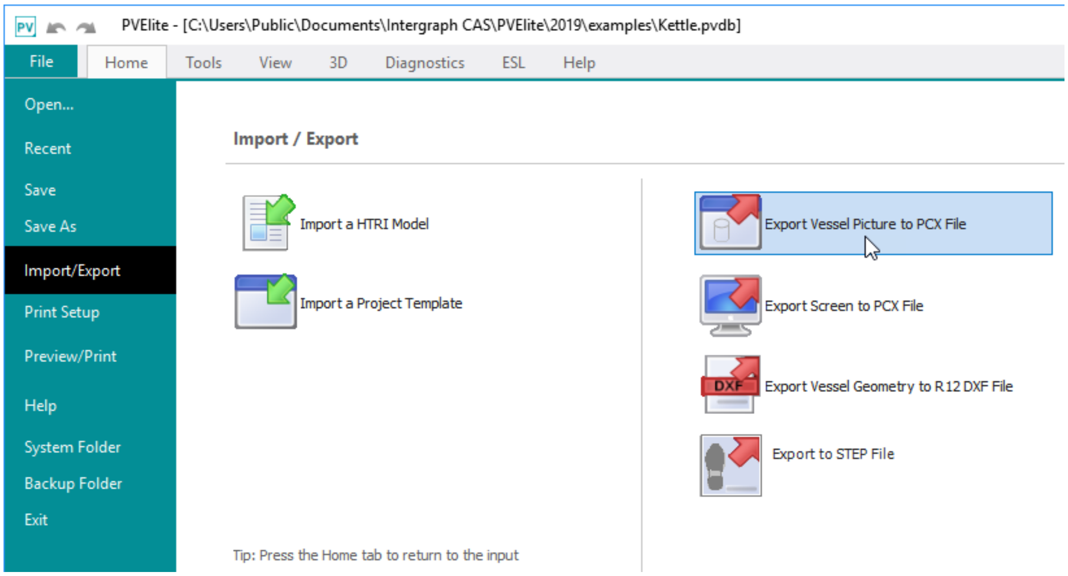
These are some reports available from PV Elite. Depending on the type, position, geometry and configuration settings the list of reports will vary.

Step 0	Vessel Element Error Checking
Cover	Cover Sheet
Title	Title Page
Step 1	Vessel Input Echo
Step 2	XY Coordinate Calculations
Step 3	Internal Pressure Calculations
Step 4	External Pressure Calculations
Step 5	Weight of Elements & Details
Step 6	ANSI Flange MAWP
Step 7	Natural Frequency Calculations
Step 8	Forces & Moments Applied to Vessel
Step 9	Wind Load Calculation
Step 10	Earthquake Load Calculation
Step 11	Wind and Earthquake Shear, Bending

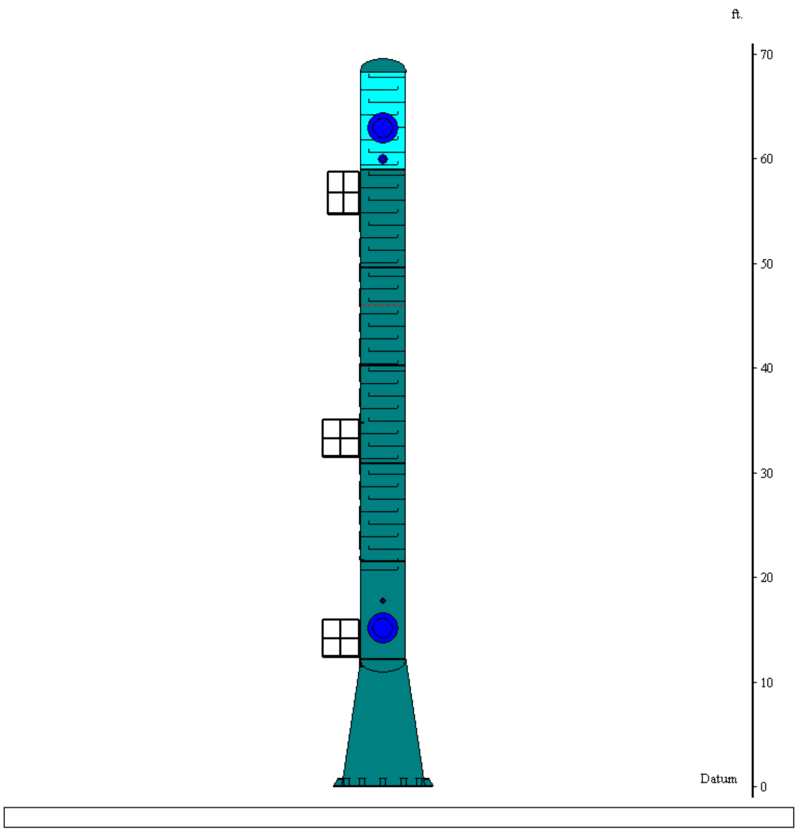
Step 12	Wind Deflection
Step 13	Longitudinal Stress Constants
Step 14	Longitudinal Allowable Stresses
Step 15	Longitudinal Stresses Due to Load Components
Step 16	Stress Due to Combined Loads
Step 17	Basering Calculations
Step 18	Center of Gravity Calculation
Cone 1-N	Conical Sections
Nozl 1-N	Nozzle Calculations
Step 20	Nozzle Schedule
Step 21	Nozzle Summary
Step 22	Vessel Design Summary

Recording the Model - Plotting the Vessel Image

At any point during the input process, a standard PCX file with the vessel image is available. This file can then be incorporated into reports or printed directly (on all printers) through most Microsoft Windows packages with graphics capabilities (such as Microsoft Word or Paintbrush). The 3D graphic can also be plotted. Click on the background of the 3D View tab before pressing the print button.



The vessel graphic can also be sent directly to the printer using **File > Preview/Print** .



Using the PV Elite Comparison Utility

When a hot fix or service pack is released for PV Elite, you may wish to reanalyze your files in the newer version of the software and compare the results with your original files. The PV Elite Comparison Utility allows you to easily perform this task. This utility provides an easy-to-read interface for visual comparisons, as well as the capability to export comparison data to a Microsoft Excel file for reference or record keeping.

NOTE You must have Microsoft Excel installed to use the PV Elite Comparison Utility.

What do you want to do?

- *Create file folders* (page 42)
- *Decompress benchmark files* (page 43)
- *Set PVE_TESTING variable to TRUE* (page 42)
- *Analyze models to create .pvr files* (page 43)
- *Perform file comparison* (page 44)
- *Review file comparison* (page 44)

Create file folders

Before running the PV Elite Comparison Utility, you must create separate folders for your original (benchmark) files and your reanalyzed (test) files.

1. Create two easily identifiable folders for your benchmark files and your test files in Microsoft Windows Explorer.
2. Copy the PV Elite files you wish to compare into both the benchmark folder and the test folder.

Set PVE_TESTING variable to TRUE

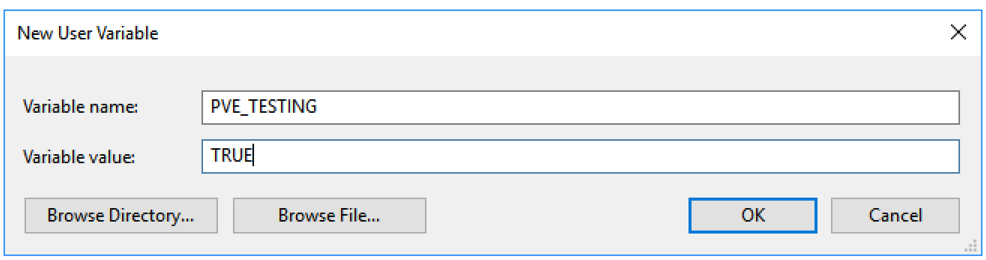
The PV Elite Comparison Utility uses PV Elite results (.pvr) files to perform the comparison. However, you must create and set the **PVE_TESTING** environment variable to **TRUE** for the software to leave the .pvr files in the directory after you analyze a PV Elite job.

To set the environment variable:

1. In File Explorer, right-click on the computer node. Select **Properties**.
2. Select **Advanced System Settings**.

   *The **System Properties** dialog box opens.*
3. On the **Advanced** tab, select **Environment Variables**.
4. Select **New** to create a new environment variable.

   *The **New/Edit User Variable** dialog box opens.*
5. Type the **Variable name** and **Variable value** as shown below, and click **OK**.



6. Click **OK** in the **Environment Variables** dialog box.
7. Click **OK** in the **System Properties** dialog box.

Decompress benchmark files

Beginning in PV Elite 2015, the software compresses the various input, intermediate, and output files into one .pv file. The PV Comparison Utility compares differences between PV Elite models, by comparing the values of the intermediate .pvr files.

Use the PV Elite File Extraction Utility to decompress the .pv files in the benchmark files folder and display the .pvr file in the folder.

1. Open the PV Elite File Extraction Utility.

   NOTE You can open the utility from the PV Elite Program folder in Windows Explorer, or from the **Tools** tab in the PV Elite application.

2. On the **Home** tab, click **Browse > Browse for Folder.**

   *The **Browse for Folder** dialog box displays.*

3. Select the file folder containing the file(s) you want to extract and click **OK**.
4. In the **File Type** field, select the file type of the file(s) you want to extract.
5. In the Compressed Files pane, select the file(s) to extract.

   TIP *Press and hold the CTRL key and click to select multiple files.*

6. Click **Extract** to extract from a single file. Click **Extract All** to extract from multiple files.

   The extracted file(s) display in the Extracted Files pane.

Analyze models to create .pvr files

When a file is analyzed in PV Elite, a .pvr file is created, but it disappears if the file compression is turned on. Before analyzing your test files in PV Elite, disable the file compression, so the .pvr files remain in the folder after analysis. Perform an analysis in the latest version of PV Elite for all the files in the test folder.

NOTE Perform these steps in the original version of PV Elite for any model in the benchmark folder that does not have a .pvr file.

1. In PV Elite, click **Tools** > **Set Configuration Parameters** to open the **Configuration** dialog box.
2. Select the **Default Value Settings** tab.
3. Select the **Do Not Compress the Input Files** option to stop PV Elite from compressing the input, intermediate and output files.
4. Drag and drop each test file from your test folder into the latest version of PV Elite.
5. Click the **Analyze** option to analyze the file.

   PV Elite saves the results file (.pvr) file for the job you analyzed in the directory. You can now compare this job with the comparable benchmark file.

6. Close the Output Processor.
7. Repeat these steps for each file in the folder.
8. When all the files have been analyzed, close the PV Elite application.

Perform file comparison

1. Locate and run the PV Elite Comparison Utility (pvqatest.exe) in the PV Elite program directory.
2. Click **Browse Bench Folder** to browse to and specify the folder in which your benchmark files are located.

   Click **Get Latest** if you want the software to default to the last Bench folder you compared.
3. Click **Browse Test Folder** to browse to and specify the folder in which your test files are located.

   Click **Get Latest** if you want the software to default to the last Test folder you compared.
4. To perform a comparison of individual files in the **PV Elite Binary Results Files** list in the left pane, click each individual file to move the file to the Results Pane on the right. To compare all files in the Bench and Test folders, click **Compare Files** at the bottom of the window.

   The results of this comparison appear in the Results Pane to the right. Test files with any difference to the benchmark files appear in red.

Review file comparison

Export the file comparison to Microsoft Excel to review the results.

1. Double-click on the failed comparison result, and the software opens an Excel file with details about the differences between the two files compared.
2. To see results of all comparisons in more detail, click **Excel Export** .

   The **GlobalDiff** tab shows all of the comparison results, including the paths to the two folders that were compared, which files were different and which were identical, and the number of differences between the files. The bottom of this tab shows all of the global details of the comparison, such as the number of files tested, passed, and missing as well as the time it took the comparison to run.

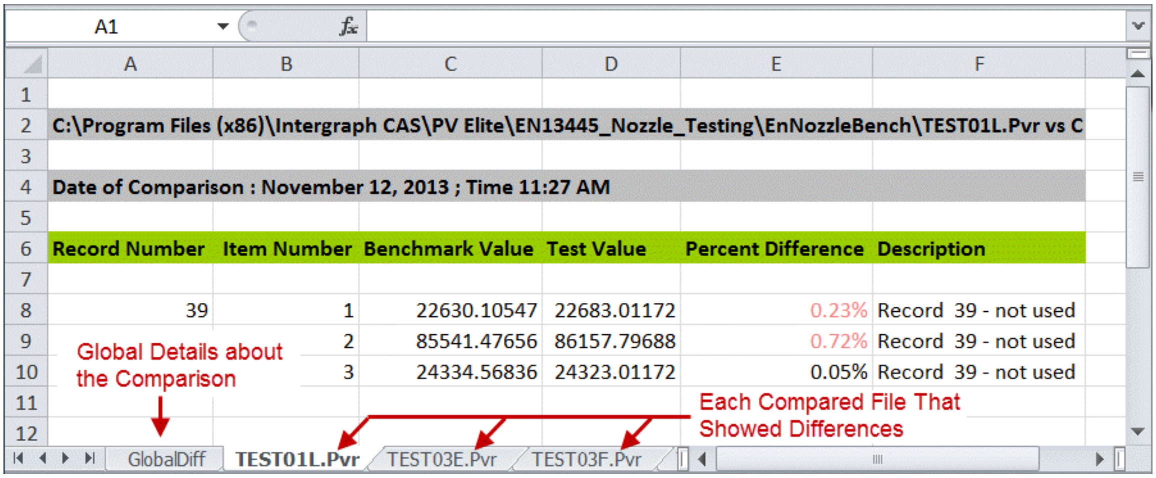
   Each tab that follows the **GlobalDiff** tab corresponds to the file comparisons that did not pass, which means the comparison between the bench and test files showed notable differences. The tab name correlates to the file name. Refer to each file comparison tab to see details about the comparison, such as the date and time the comparison was done, the values in compared files that were different, and the percentages of difference between the compared files.

   The **Percent Difference** column denotes the amount of difference between the test and the benchmark file. The following colors denote percentages of difference:

   Light Pink - Less than 1 percent difference between the files

   Dark Pink - 1 percent up to a 3 percent difference between the files

Red - 3 percent or greater difference between the files



DXF File Generation Option

PV Elite can write out Data Interchange Files (3 all together) using **File > Import/Export > Export Vessel Geometry to R12 DXF File**. This DXF file is a text file that contains commands for generating a 2D CAD drawing of the vessel that is on a one to one scale. The border and text are scaled by the diameter conversion constant and the scale factor generated by the program or defined by you. Many popular drawing programs such as AutoCad® and MicroStation® can read and process these files. The DXF files produced by **PV Elite** are release 12 compatible. Any version of AutoCad including release 12 and after should be able to read the DXF file.

Three files are produced: the vessel drawing, the nozzle schedule, and the Bill of Material. The files are written in the folder where the input file for the vessel file is located. These files are written at the end of the program's calculation execution. Nearly every individual has his/her own way of drafting. A conscious effort was made not to be too specific. This approach allows the drafter to take the vessel drawing file and edit it as necessary.

SECTION 3

File Tab

Controls general operations of PV Elite files.

Open - Opens an existing .pvdb, .pvi, or .pv file. For more information, see *Open* (page 47).
Recent - Displays recently-opened files and folder. Select a file to open.
Save - Saves the open file. For more information, see *Save* (page 47).
Save As - Saves the open file with a new name. For more information, see *Save As* (page 47).
Import/Export - Exports the open .pvdb or .pv file to a .pcx, .dxf, or .stp file. For more information, see *Import/Export* (page 47).
Print Setup - Selects a printer and printer options. For more information, see *Print Setup* (page 48).
Preview/Print - Prints the graphics in the open file. For more information, see *Preview/Print* (page 48).
Help - Displays help, getting started, contact information, and version number for PV Elite.
System Folder - Opens the System folder in Windows Explorer.
Backup Folder - Opens the Backup folder in Windows Explorer. The software saves a backup of your PV Elite files upon analysis.
Exit - Closes the software.

Open

File tab: **Open**

Activates the **Open** dialog box from which you can open a previously saved PV Elite file for editing. You can also open one of the many example files delivered with PV Elite.

Open is also available on the quick access toolbar in the top-left corner of the PV Elite window.

Save

File tab: **Save**

Saves the PV Elite file that you have open. When you save a file for the first time, the **Save As** dialog box appears so that you can name the file and select a folder location.

Save is also available on the quick access toolbar in the top-left corner of the PV Elite window.

NOTE The software automatically saves files into .pvdb format.

Save As

File tab: **Save As**

Saves the open file with a different name or to a different folder location.

NOTE The software automatically saves files into .pvdb format.

Import/Export

File tab: **Import/Export**

Select one of the following:

- **Import a HTRI Model -** This command is not available at this time.
- **Import a Project Template** - Imports a PV Elite project template into the current file.
- **Export Vessel Picture to PCX File -** Sends the vessel graphics to a .pcx file. This file can be printed at a later date or added to other documents. The .pcx file is created in the same folder and with the same name as the .pvdb or .pv file.
- **Export Screen to PCX File** - Sends a snapshot of the entire screen to a .pcx file. The .pcx file is created in the same folder and with the same name as the .pvdb or .pv file.
- **Export Vessel Geometry to R12 DXF File** - Exports the vessel to a .dxf file. For more information, see *Setting Up the Required Parameters* (page 219).
- **Export to STEP File** - Exports the vessel to a .stp file. The .stp file is created in the same folder and with the same name as the .pvdb or .pv file.

Print Setup

File tab: **Print Setup**

Selects a printer and defines printer options in the **Print Setup** dialog box.

Preview/Print

File tab: **Preview/Print**

Prints the model to a printer or to a PDF file. Options are available for print range, number of copies, and other printing characteristics.

NOTE You must have access to a printer, either locally or over your network, before you can use this command.

Exit

File tab: **Exit**

Closes the open file and exits the software. If you have changed data since the file was last saved, or if you have not saved a new file, the **Save As** dialog box appears and the software prompts you to save your changes.

SECTION 4

Home Tab

The **Home** tab contains the most common commands that you use in PV Elite.

Panel	Description
File	Provides file management commands, such as **Open**, **Save**, and **Print**. For more information, see *File Panel* (page 50).
Elements	Provides commands for elements used to create a vessel, such as cylinders, spheres, and cones. For more information, see *Elements Panel* (page 51).
Details	Provides commands to add details to a vessel element, such as stiffeners, nozzles, forces, moments, lining, half-pipe jackets, and tubesheets. For more information, see *Details Panel* (page 52).
Input/Output	Provides commands to add load and restraint information. For more information, see *Input/Output Panel* (page 212).
Utility	Provides miscellaneous element commands, for functions such as insert, delete, update, share, and flip. For more information, see *Utility Panel* (page 214).
Auxiliary	Provides miscellaneous model commands for functions such as manipulating of the model, creating drawings, and viewing properties. For more information, see *Auxiliary Panel* (page 216).
Analyze	Provides commands for analyzing the model. For more information, see *Analyze Panel* (page 221).
Units/Code	Provides commands for changing units and the design code. For more information, see *Units/Code Panel* (page 222).

File Panel

The following commands are available on the **File** panel on the **Home** tab.

	New - Creates a new .pvdb file. For more information, see *New* (page 50).
	Open - Opens an existing .pvdb, .pvi, or .pv file. The **Open** command is also available on the Quick Access bar at the top. For more information, see *Open* (page 47).
	Save - Saves the open file. The **Save** command is also available on the Quick Access bar at the top. For more information, see *Save* (page 47).

New

Home tab: **File > New**

Creates a new PV Elite input file (.pvdb). You can also press **Ctrl+N** on the keyboard to create a new input file.

You can select a specification (**ASME Section VIII-Division 1**, **ASME Section VIII-Division 2**, **British Standard PD 5500**, or **EN-13445**) or just create an empty .pvdb file. You can also select **From Project Template** (page 50) to create a new (.pvdb) file from a PV Elite template (.pvpt) file.

TIP Save time when creating PV Elite jobs by clicking the down arrow on the **New** box and selecting a code specification (such as **ASME Section VIII-Division 2** or **British Standard PD 5500).** The software automatically populates certain inputs in the job based on the code specification you selected.

From Project Template

Home tab: **File > New > From Project Template**

Creates a new PV Elite input file (.pvdb) based on the project template that you select. The software opens a window where you can browse and select any PV Elite template (.pvpt) file.

You can specify a default folder for project templates in the configuration settings. For more information, see *Configuration* (page 224).

Elements Panel

Provides commands for elements used to create a vessel, such as cylinders, spheres, and cones. The **Elements** panel is available on the **Home** tab.

	Cylinder - Cylindrical shell
	Ellipse - Elliptical head
	Torisphere - Torispherical head
	Sphere - Spherical head or shell
	Cone - Conical head or shell segment
	Welded Flat Head - Places a welded flat head.
	ANSI/Bolted Flange - Bolted body flange
	Skirt - Skirt support with base ring

The software does not require the complete construction of a vessel for analysis. Individual elements or groups of elements may be defined and partially analyzed. Only complete vessels with proper supports can be analyzed for dead weight and live loads.

All elements, except **Skirt**, can be used to create either horizontal or vertical vessels.

- Models for vertical vessels are built from bottom to top.
- Models for horizontal vessels are built from left to right.

The vessel orientation is established with the first element. If starting with a skirt, it is a vertical vessel. If starting with a head, the head can be flipped between a bottom head (vertical vessel) and a left head (horizontal vessel) by clicking **Flip Orientation** on the **Utilities** panel. After the second element is added to the model, the orientation is fixed. Skirts are the only vessel supports that are modeled as elements. Other supports, such as legs and lugs for vertical vessels and saddles, are modeled as details on the elements. For more information, see *Details Panel* (page 52).

Details Panel

Details can only be specified on the currently selected element (see *Elements Panel* (page 51) for more information on placing elements). Details, such as nozzles, insulation, operating fluid, platforms and the like, define the vessel's weight information for load calculations. These commands are available on the **Details** panel of the **Home** tab.

	**Stiffening Ring** - Add stiffening rings on the selected cylinder element. For more information, see *Stiffening Ring* (page 57).
	**Nozzle Input** - Add nozzles on the selected cylinder or head element. For more information, see *Nozzle* (page 60).
	**Weight/Piping Input** - Add piping and miscellaneous weight added to the selected cylinder or head element. For more information, see *Weight* (page 93).
	**Force/Moment Input** - Add external forces and moments to the selected cylinder or head element. For more information, see *Force and Moment* (page 96).
	**Platform Input** - Add a platform to the selected element. For more information, see *Platform* (page 97).
	**Packing Input** - Add packing to the selected element. For more information, see *Packing* (page 101).
	**Saddle Input** - Add saddles to the selected horizontal cylinder element. For more information, see *Saddle* (page 103).
	**Tray Input** - Add a set of equally spaced trays to the selected vertical cylinder element. For more information, see *Tray* (page 111).
	**Lug Input** - Add support lugs to the selected vertical cylinder element. For more information, see *Lug* (page 112).
	**Leg Input** - Add support legs to the selected vertical cylinder element. For more information, see *Legs* (page 116).
	**Liquid Input** - Add liquid data to the selected element. For more information, see *Liquid* (page 124).
	**Insulation Input** - Add insulation to the selected element. For more information, see *Insulation* (page 127).
	**Lining Input** - Add lining to the selected element. For more information, see *Lining* (page 128).

	Half-Pipe Jacket or Limpet Coil - Add half pipe jackets to the selected cylinder element. For more information, see *Halfpipe Jacket* (page 129).
	Tubesheet Analysis - Add a tubesheet to the selected element. For more information, see *Tubesheet* (page 133).
	Add a Clip - Add a clip to the selected element. For more information, see *Add a Clip* (page 181).
	Lifting Lug Data - Add lugs to the selected element. For more information, see *Lifting Lug Data* (page 185).
	Jacket or Vapor/Distribution Belt - Add a jacket or vapor/distribution belt to the selected element. For more information, see *Jacket or Vapor/Distribution Belt* (page 190).
	API-579 FFS - Add a flaw to the selected element. For more information, see *API-579 Flaw/Damage Input/Analysis* (page 199).

Add a Detail

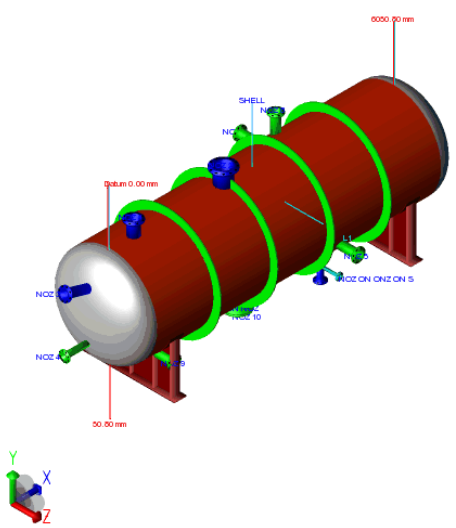
Details are assigned to elements using the commands on the **Details** panel on the **Home** tab.

1. Select an element in the graphics view.
2. Click the command on the **Details** panel on the **Home** tab.

   The dialog box for the command displays.

3. Enter the needed information, and click **OK**.

   The detail is added to the element and displays in the graphics view.



Modify a Detail

To modify one or more existing details:

1. In the graphic view, click the element having the existing detail that you want to modify.
2. Click the corresponding command on the **Details** panel.

   The dialog box for the command displays.
3. If there is only one detail, make the needed changes, and click **OK**.
4. If there is more than one detail, click **Previous** or **Go To Next** to modify the needed detail.

NOTE In the graphic view, you can also right-click a detail on the selected element to open the dialog box for that detail.

Common Detail Parameters

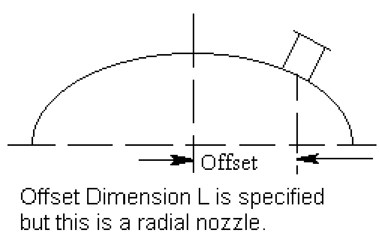
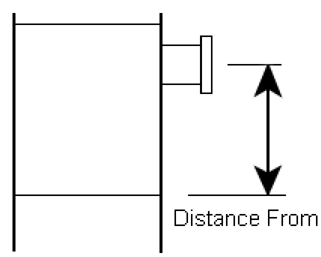
The following fields appear in the dialog box for most details.

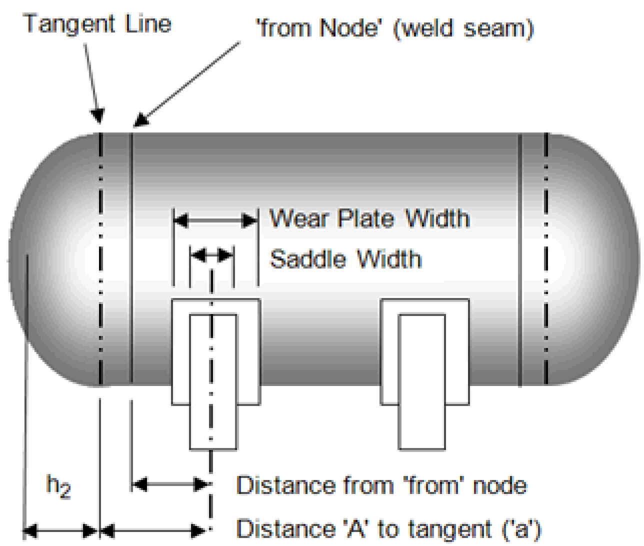
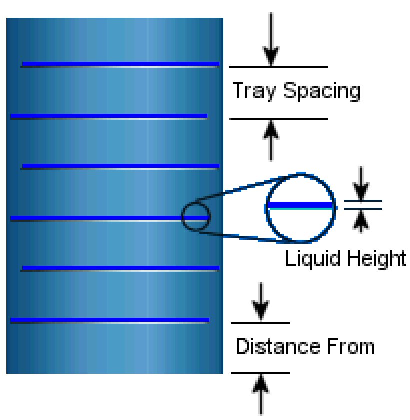
From Node

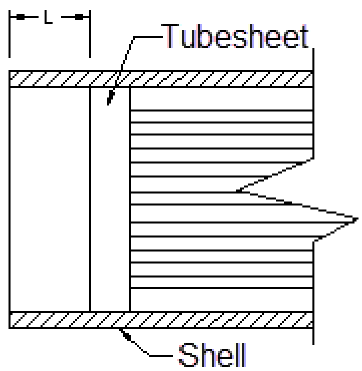
Displays the **From Node** for the selected element. The **From Node** is the software-generated node number describing the starting location of the element. The value cannot be modified.

Distance from "From" Node

Enter the axial or longitudinal distance from the From Node to the detail at the following locations:

Detail	Location
Stiffening Ring	Centerline of the first ring.
Nozzle	Centerline of the nozzle. If the nozzle is located in a head, then enter the distance from the head centerline. This dimension should always be entered if it is not zero. 
Weight	Point at which the weight acts.
Force and Moment	Point at which the force or moment acts.
Platform	Bottom of the platform.

Detail	Location
Packing	Start of the packed section.
Saddle	Vertical centerline of the saddle. 
Tray	Bottom of the lowest tray. 
Lug	Centroid of the lug attachment weld.
Legs	Centroid of the leg attachment weld. **NOTE** The software uses the value of **Overall Length of Legs** when performing AISC unity checks.
Liquid	Start of the liquid section.
Insulation	Start of the insulated section.

Detail	Location
Lining	Start of the lined section.
Halfpipe Jacket	The distance that the jacket starts from the circumferential seam of this element. This value, in conjunction with **Length Along Shell of Jacket Section**, is used to determine the weight of the jacket.
Tubesheet	The distance from the shell to the outer face of the nearer tubesheet. 
Generic Clip	Center of the clip.
Lifting Lug	For a vertical vessel, the centroid of the weld group. For example, the base of the lug + 1/2 the weld distance (*wl*) on the side of the lug. If the vessel is horizontal, enter the distance from the left end or tangent of the vessel to the center of each of the two lugs.
Jacket or Vapor/Distribution Belt	Bottom of the jacket.

Detail Description

Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

Stiffening Ring

Home tab: **Details** > **Stiffening Ring**

Adds one or more stiffening rings to the selected cylinder element.

As stiffening ring data is entered, the software automatically calculates the required inertias, except when the ring is at a cone-to-cylinder junction. For bar rings, the software sizes a new ring based on a default thickness of 0.375 inches or the element value entered for **Bar thickness to use when designing new rings** on the **Equipment Installation and Miscellaneous Options Dialog Box** of the **Load Cases** tab.

NOTE The maximum length between stiffeners is shown on the *Status Bar* (page 17) as **Slen**. When **>>>** displays, the element does not require a reinforcing ring for external pressure purposes.

Previous Ring

If you created more than one ring on the element, click to go back to the previous ring.

Go To Next Ring

If you created more than one ring on the element, click to go to the next ring.

Add New Ring

Click to add a new ring to the shell or head element.

Delete

Deletes all data for the current ring.

Common Detail Parameters (page 54)

Ring Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Ring Location

Select the location of the ring. Select **ID** if the ring is on the inside diameter of the cylinder. Select **OD** if the ring is on the outside diameter of the cylinder. **OD** is the most common selection.

Ring Type

Select **Bar Type** for a flat bar stiffener ring. Select **Section Type** for a more complex structural cross-section.

For **Bar Type**, click **Check "Standard" Bars**. The **Bar Selection** dialog box displays, where you select the flat bar size, and enter values for **Axial Thickness** and **Radial Width**. Bar type stiffeners have a simple rectangular cross section and the software calculates the ring properties based on the width and thickness of the ring. Most stiffeners are bar type.

For **Section Type**, select a standard cross-section in **Choose a Section** or click **Section Calculator** to create a custom fabricated section. You need the following properties for the section: **Moment of Inertia, Cross Sectional Area, and Distance to Ring Centroid.** The software provides these values for standard cross-sections. You must enter the values for a custom section, as defined in the AISC Steel Construction Manual.

Inside Diameter

Enter the inside diameter of the stiffening ring. This value is usually equal to the outside diameter of the vessel shell (when **OD** is selected for **Ring Location**), except for the less common case of a stiffening ring on the inside of the vessel (when **ID** is selected for **Ring Location**).This value is used both to calculate the weight of the ring and the stiffness of the ring for external pressure purposes. This entry is only available when **Bar Type** is selected for **Ring Type**.

Thickness

Enter the axial thickness of the stiffening ring. This value is used to calculate the weight of the ring and the stiffness of the ring for external pressure purposes. This entry is only available when **Bar Type** is selected for **Ring Type**.

Outside Diameter

Enter the outside diameter of the stiffening ring. This value is usually greater than the outside diameter of the vessel shell (when **OD** is selected for **Ring Location**), except for the less common case of a stiffening ring on the inside of the vessel (when **ID** is selected for **Ring Location**). This value is used to calculate both the weight of the ring and the stiffness of the ring for external pressure purposes. This entry is only available when **Bar Type** is selected for **Ring Type**.

Structural Database

Select the structural specification database to use for cross-sections. This entry is only available when **Section Type** is selected for **Ring Type**.

Moment of Inertia

Displays the moment of inertia of the stiffening ring about its neutral axis from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. The software uses this value to determine the adequacy of the ring for external pressure calculations, and for conical calculations according to Appendix 1 of the ASME Code. This entry is only available when **Section Type** is selected for **Ring Type**.

Cross Sectional Area

Displays the cross sectional area of the stiffening ring from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. This entry is only available when **Section Type** is selected for **Ring Type**.

Distance to Ring Centroid

Displays the distance from the surface of the shell to the ring centroid from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. This entry is only available when **Section Type** is selected for **Ring Type**.

Section Ring Height

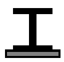
Displays the depth of the cross-section from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a value according to the selected specification. This entry is only available when **Section Type** is selected for **Ring Type**.

Section Name

Displays the section name from the specification selected for **Structural Database,** when a standard section is selected for **Choose a Section.** If you created a custom section using **Section Calculator**, enter a name. This entry is only available when **Section Type** is selected for **Ring Type**.

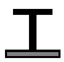
Choose a Section

Select a section type and a section size:

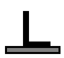


I - Beam
W Section

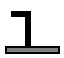
Channel



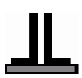
WT Section
MT Section
ST Section
(strong axis out)



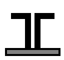
Angle
(weak axis out)



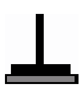
Angle
(strong axis out)



Double Angle
(weak axis out)



Double Angle
(strong axis out)



WT Section
MT Section
ST Section
(weak axis out)

Ring Fillet Weld Leg Size

Enter the size of the leg of the ring fillet weld.

Ring Attachment Style

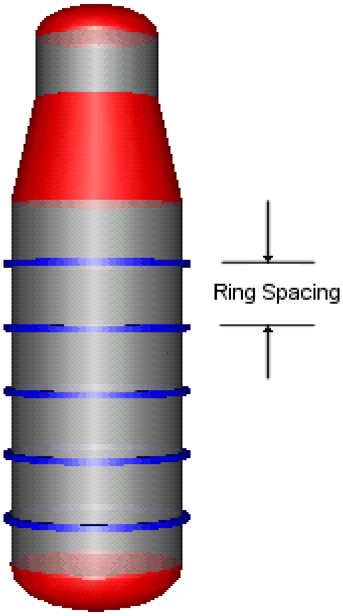
Select the type of fillet weld. Select **Intermittent**, **Continuous**, or **Both**.

Number of Rings to Add

Enter the number of rings to place on the cylinder element. The software resets this number to 0 after you close the **Stiffening Ring** dialog box, so it is possible to add multiple groups of rings.

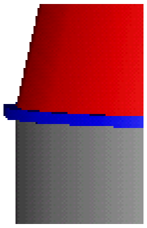
Ring Spacing

Enter the ring spacing. The first ring is placed at the distance entered for **Distance from "From" Node**. The rest of the rings are placed after the first ring by the **Ring Spacing** increment. If you are adding one ring, a value is not needed. The software resets this number to 0 after you close the **Stiffening Ring** dialog box, so it is possible to add multiple groups of rings.



Cone to Shell Junction Ring?

Select to attach the ring at the junction of the cone and cylinder elements. In this case, the software does not design the ring for external pressure considerations, but considers it for junction reinforcement according to App. 1-5 and 1-8 (Div. 1) or AD 360.3 (Div. 2).



Nozzle

Home tab: **Details** > **Nozzle**

Adds a nozzle to the selected cylinder or head element. Nozzles add to the total dead weight of the vessel and are used to evaluate the maximum allowable working pressure (MAWP) of the vessel and flange.

Nozzle Main Tab (Nozzle Input/Analysis Dialog Box) (page 61)

Local Stress Analysis (WRC 107, 297 or Annex G) Tab (Nozzle Input/Analysis Dialog Box) (page 85)

Nozzle Main Tab (Nozzle Input/Analysis Dialog Box)

Defines physical and weld parameters for the nozzle.

Common Detail Parameters (page 54)

Previous Nozzle

If you created more than one nozzle on the element, click to go back to the previous nozzle.

Go To Next Nozzle

If you created more than one nozzle on the element, click to go to the next nozzle.

Add New Nozzle

Click to add a new nozzle to the shell or head element.

Delete

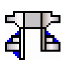
Deletes all data for the current nozzle.

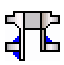
Plot

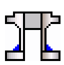
Click to open the **Nozzle Graphics** dialog box. A cross-section view of the nozzle design is shown.

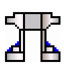
Nozzle Attachment

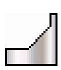
Select the type of nozzle:

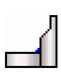
 Inserted nozzle with reinforcing pad

 Inserted nozzle without reinforcing pad

 Abutting nozzle with reinforcing pad

 Abutting nozzle without reinforcing pad

 Heavy Barrel Type, HB

 Type “F” connection

FVC Catalog

Select **FVC Catalogue** to select a nozzle from the Forged Vessel Connections catalog. For more information about FVC, see *http://www.forgedvesselconn.com/* (*http://www.forgedvesselconn.com/*).

- **Nominal Diameter** - Select the nominal diameter that you need.
- **Flange Class** - Select the pressure rating that you need.
- **Connection Type** - Select the connection type that you need.

- **Overall Length Oal** - Enter the length that you need from the surface of the vessel to the flange face.
- **Nozzle does not have a "Nut Relief"** - Select if the nozzle does not have a nut relief.
- **Select Now** - Loads the FVC nozzle data from the catalog into the main **Nozzle Input** dialog box.

Coupling Lookup

Select **Coupling Lookup** to find coupling properties for the nozzle. Select the pressure rating and the diameter, and then click **Select Now**.

Just Like

Select to place another nozzle just like an existing nozzle.

Existing Nozzle Description

Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

NOTE When using ASME VIII - 1, a special directive is available for small nozzles. If the text directive **#SN** is placed anywhere in the description, the software calculates the areas and MAWP of this nozzle connection. This directive overrides the global directive set in **Tools** > **Configuration**. It may be necessary to use this directive when required by UG-36. This paragraph in the code defines a small nozzle.

Nozzle Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Schedule

Specify the thickness of the nozzle by selecting the schedule of the nozzle neck pipe. Acceptable schedules are:

SCH 5S	SCH 30
SCH 10	SCH 40
SCH 10S	SCH 40S
SCH 20	SCH 60
SCH 30	SCH 80
SCH 40	SCH 80S
SCH 40S	SCH 100
SCH 60	SCH 120
SCH 80	SCH 140
SCH 5S	SCH 160
SCH 10	SCH STD
SCH 10S	SCH X-STG
SCH 20	SCH XX-STG

NOTES

- DIN schedules are also available.
- All schedules of pipe might not have a corresponding diameter associated. In this case, the software displays an error stating the thickness of the nozzle was not found.

Diameter

Enter the actual diameter of the nozzle when **Actual** is selected for **Thickness Basis**. Enter the nominal diameter, or click ... to select a diameter when **Nominal** or **Minimum** is selected for **Thickness Basis**. Values are given in English units and must be multiplied by the diameter conversion constant so that the software arrives at the correct diameter when it calculates the English value of the diameter. Valid English-unit nominal diameters range from 0.125 to 30.0 inches.

Dia. Basis

Select the type of diameter to use for the element. Select **ID** for the inside diameter. Select **OD** for the outside diameter. **ID** and **OD** are available for all design codes.

NOTES

- The ASME code provides different equations for required thickness based on whether the geometry is specified on **ID** or **OD**. By using the **ID** basis, the software computes a thinner required thickness, Tr, for the nozzle, such as in high-pressure, thick-wall geometries.
- If you are modeling a cylinder with welded flat heads on either end, and the welded flat heads sit just inside the cylinder shell, set **Diameter Basis** to **ID** and specify the **Inside Diameter** value on the welded flat heads to be the same size as the **Inside Diameter** of the cylinder. After you make these changes, if the flat head element still displays as sitting on the cylinder shell (instead of inside of the shell), select **Flip Orientation** twice. The software refreshes the model display to show the welded flat head inside the cylinder shell.

Thickness Basis

Select the thickness basis:

- **Actual** - The software uses the actual values entered for **Diameter and Actual Thk**. Select this value if the nozzle is fabricated from plate.
- **Nominal** - The software uses **Diameter** as the nominal diameter and looks up the actual diameter. The software also looks up the nominal thickness based on the selection for **Schedule**.
- **Minimum** - The software uses **Diameter** as the nominal diameter and looks up the actual diameter. The software also looks up the nominal thickness based on the selection for **Schedule**. It then multiplies the nominal thickness by a factor of 0.875.

Total CA.

Enter the corrosion allowance. The software adjusts both the actual thickness and the inside diameter for the corrosion allowance. For ASME VIII-1, if the nozzle has an external corrosion allowance, add the external corrosion allowance to the internal corrosion allowance and enter the total value.

Actual Thk

If you selected **Actual** for **Thickness Basis**, enter the minimum actual thickness of the nozzle wall.

Nozzle Orientation (Nozzle Main Tab) (page 64)
Pad or Hub Properties (Nozzle Main Tab) (page 73)
Additional Weld Data (Nozzle Main Tab) (page 75)
Miscellaneous (Nozzle Main Tab) (page 76)

Nozzle Orientation (Nozzle Main Tab)

Defines orientation parameters for the nozzle.

Is this nozzle connected to another nozzle?

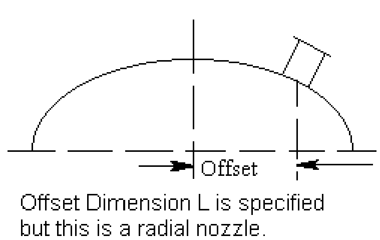
Select this option if the nozzle is connected to another nozzle.

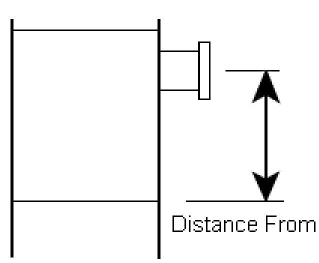
Parent Nozzle

Select the parent nozzle for this nozzle.

Offset Dimension L

Enter the axial distance from the **From Node** to the centerline of the nozzle. If the nozzle is located in a head, then enter the distance from the head centerline. This dimension should always be entered if it is not zero. This option is similar to **Distance from "From" Node** and is available when the nozzle is on a head element.





Distance from Shell Surface

Enter the axial distance from the **From Node** to the centerline of the nozzle. If the nozzle is located in a head, then enter the distance from the head centerline. This dimension should always be entered if it is not zero. This option is similar to **Distance from "From" Node** and is available when **Is this nozzle connected to another nozzle?** is selected.

Layout

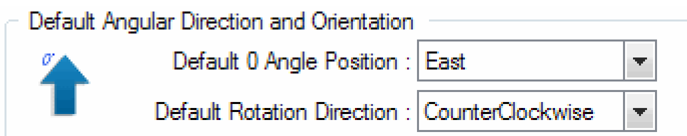
Click to open the **Nozzle Layout and Placement Dialog Box (page 70)** and set the nozzle orientation independent of the coordinate system of the model.

Layout Angle

Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the weight at the point where it is attached to the shell.

For a horizontal vessel, the zero degree line is at 12:00 (looking at a clock); 90 degrees is at 3:00. Entering these layout angles is important if the horizontal vessel has a liquid level and the nozzles are being designed using **Design P + Static Head**, selected for **Nozzle/Clip Design Pressure Options** on the **Load Cases** tab. For a vertical vessel, the angle is more arbitrary. For purposes of rendering the graphics, the assumption is that the zero degree line is at 3:00 and 90 degrees is 12:00.

The position of the zero degree reference direction (North, West or East) and the angular rotation (clockwise or counterclockwise) are set in **Tools** tab, **Set Configuration Parameters**, *Default Value Settings Tab (Configuration Dialog)* (page 233).



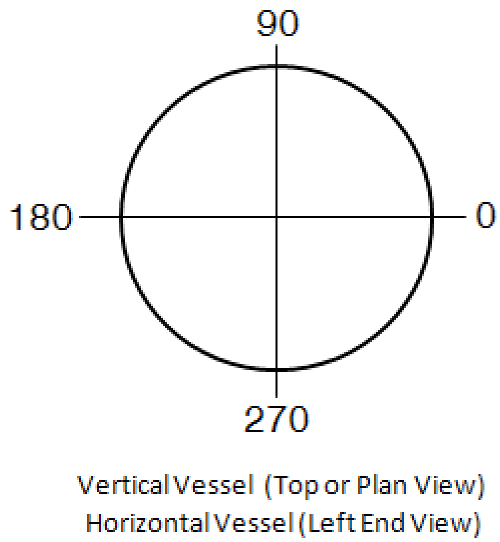
If the new orientation is different from the orientation of the current model, the software asks to update the model with the new settings. If the settings are updated, the weights and areas of platforms must be recalculated:

- Click **List Dialog** (page 218). On the **Detail Listing** dialog box, click the **Platform** tab so that platform data is visible. Click **Platform Wind Area** and **Platform Weight** to recalculate.
- Click **Platform Input** for each platform. Tab through the fields in the **Platform** dialog box, causing the weight and area to be recalculated

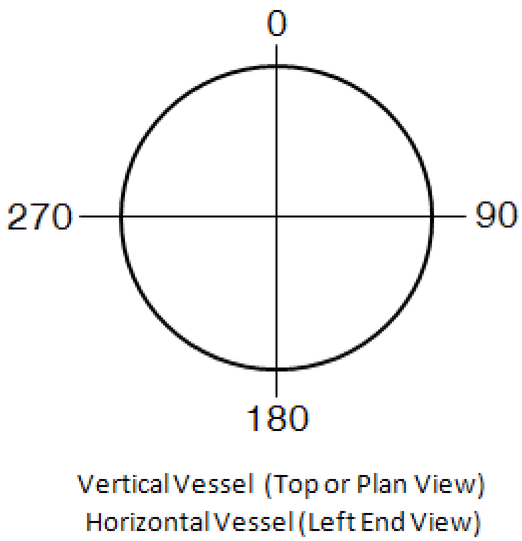
NOTE The angular settings apply to nozzles, clips, legs, lifting lugs, support lugs, base rings, platforms, weights and half-pipe jackets.

Examples

Default Orientation

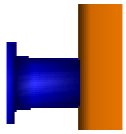


Angle Position: North
Rotation Direction: Clockwise



Radial Nozzle

Select to specify a radial nozzle.

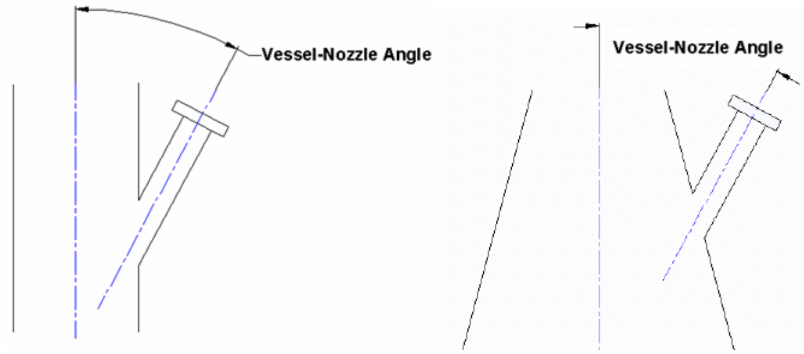


Angle or Lateral Nozzle

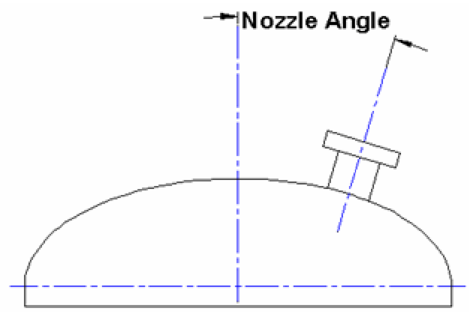
Select to specify an angled or lateral nozzle. Also enter a value for **Centerline Tilt Angle**, and — if needed — for **Cyl./Cone Offset Dimension L**.

Centerline Tilt Angle

Enter the angle for a non-radial nozzle when Angle or Lateral Nozzle is selected.



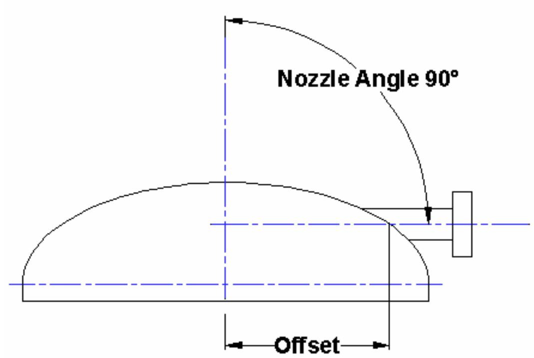
Non-radial nozzles can be specified by entering the angle between the vessel and nozzle centerlines and the offset from vessel centerline. This vessel-nozzle centerline angle can vary from 0 to a limiting value depending upon specific geometry. For nozzles on top heads, this value will generally range between 0 and 90 degrees. On bottom heads, this value would be between 90 and 180 degrees:

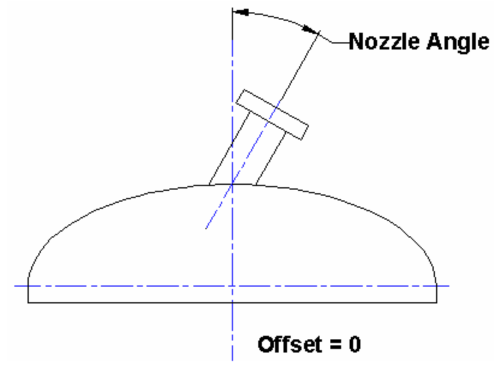


NOTE The input specification for non-radial and non-hillside nozzles changed starting with version 2008. The angle is measured between the centerline of the nozzle and the centerline of the vessel. This value can be determined from an electronic drawing of the model.

Cyl./Cone Offset Dimension L

Enter an offset dimension from the vessel centerline for hillside nozzles (neither **Radial Nozzle** or **Centerline Tilt Angle** are selected) and angled nozzles (**Centerline Tilt Angle** is selected).

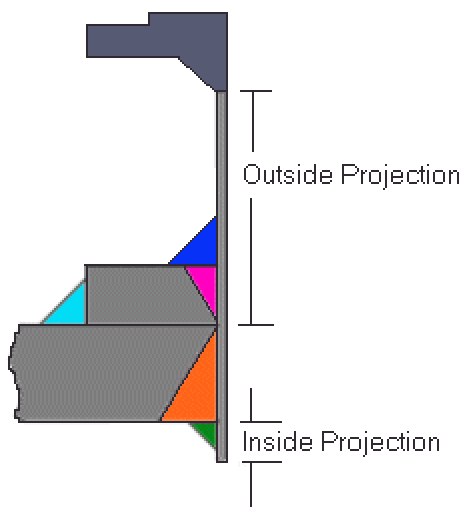




Hillside nozzles and some tilted nozzles are subject to calculations to meet ASME area requirements in both planes of reinforcement. In these cases, the software automatically checks area requirements in both planes using the corresponding lengths of the nozzle opening. For integral construction, the Code F correction factor of 0.5 is automatically applied in the hillside direction. If the connection is pad reinforced, a value of 1.0 is used. The F factor is used to account for the fact that the longitudinal stress is one-half of the hoop stress. The use of the F factor is limited to nozzles located on cylindrical and conical sections under internal pressure.

Projection Outside

Enter the nozzle projection distance outward from the surface of the vessel. The distance is usually to the attached flange or cover. This length is used for weight calculations and for external pressure calculations.



Projection Inside

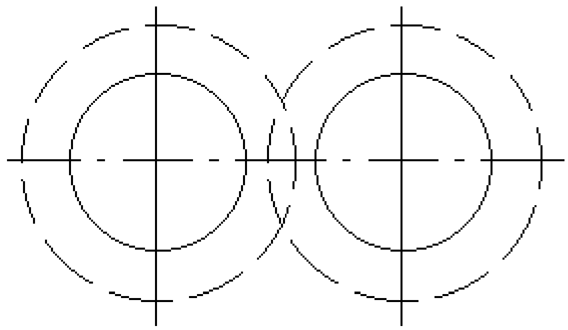
Enter the nozzle projection distance into the vessel. The software uses the least of **Projection Inside** and **Limits (Thickness)** with no pad to calculate the area available in the inward nozzle. Therefore, you may safely enter a large number such as six or twelve inches if the nozzle continues into the vessel a long distance.

For some nozzle types such as those shown in ASME VIII-1 UW-16.1 sketch K, the inside projection is the distance from the inside surface of the vessel to the inside edge of the nozzle. This distance must be less than the shell thickness.

Limits (Diameter)

Enter the maximum diameter for material contributing to nozzle reinforcement. For example:

- Where two nozzles are close together and the reinforcements overlap.



- A vessel seam for which you do not want to take an available area reduction.

Limits (Thickness)

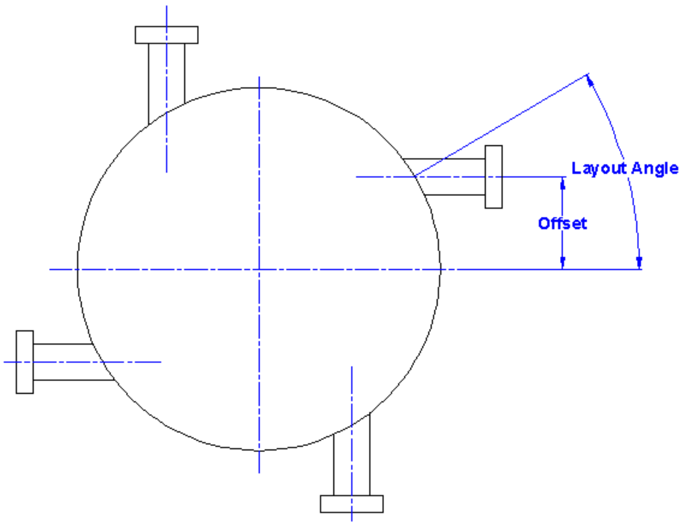
Enter the maximum thickness for material contributing to nozzle reinforcement. For example, where a studding pad or nozzle stub do not extend normal to the vessel wall as far as the thickness limit of the nozzle calculation.

Overriding Weight

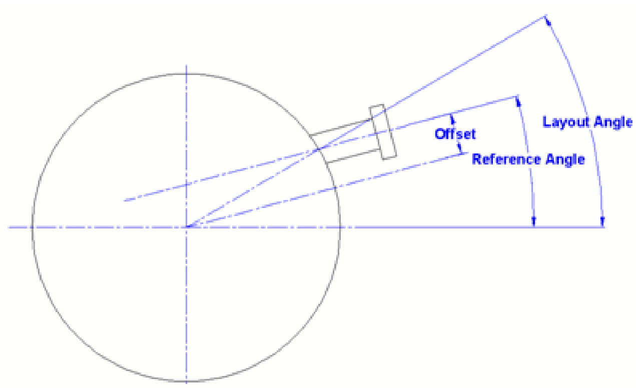
Enter a weight if the nozzle is significantly different from a standard weight nozzle. The weight overrides the software-calculated weight that is based on other entered values and internal tables of typical weights.

Nozzle Layout and Placement Dialog Box

Provides an alternative and more versatile method of nozzle orientation. You are not confined to having nozzles point in the directions of the coordinate system axes of the model. For example, when using the standard parameters (such as **Offset Dimension L** and **Layout Angle**) on the **Nozzle Input/Analysis** dialog box, hillside nozzles may only point in the X, Y and Z directions:

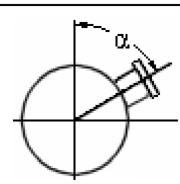
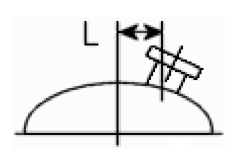


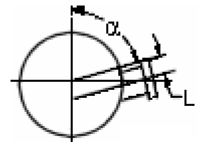
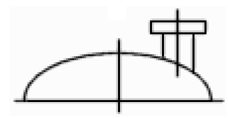
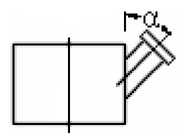
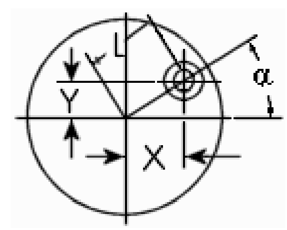
When using the **Layout** command and the **Nozzle Layout and Placement** dialog box, a hillside nozzle can point in any direction by using **Reference Angle alpha** instead of **Layout Angle**:



Nozzle Style

Select a nozzle orientation style:

Nozzle Style	Cylinder Element	Head Element
Radial Nozzle		

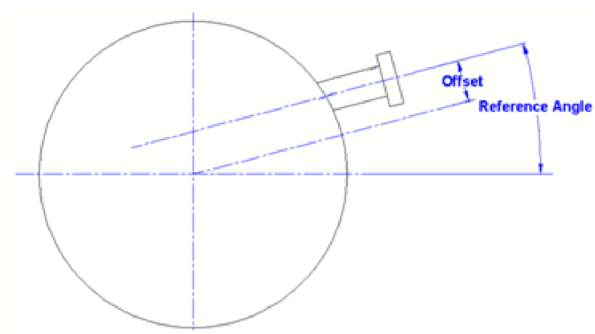
Hillside Nozzle		
Lateral Nozzle		

Layout, Reference or Tilt Angle alpha

Enter the angle of the line to use as reference for **Nozzle Offset Dimension L**.

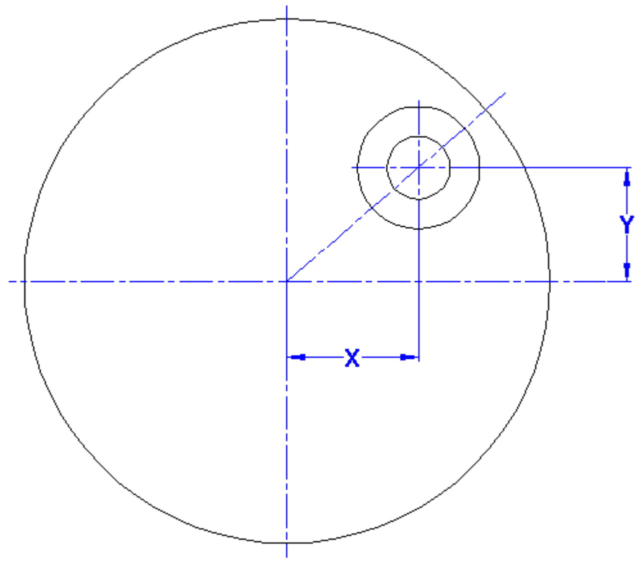
Nozzle Offset Dimension L

Enter the distance from the reference line to the centerline of the nozzle.



Nozzle "X" Dimension and **Nozzle "Y" Dimension**

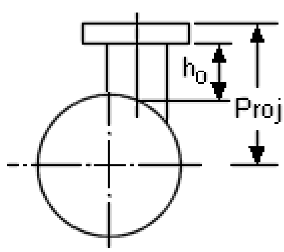
For nozzles on heads, enter values for **Nozzle "X" Dimension** and **Nozzle "Y" Dimension** to calculate **Reference Angle alpha** and **Nozzle Offset Dimension L** from the X-Y coordinate location. Nozzle locations on heads are often given in the X-Y coordinate system.



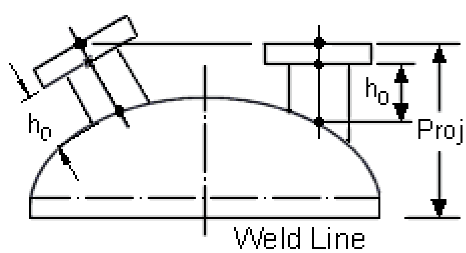
Projection Dimension "Proj"

For a radial or hillside nozzle, enter the projection from the centerline of the cylinder to the end of the nozzle. The software calculates the projection *ho*.

For a nozzle on a cylinder:

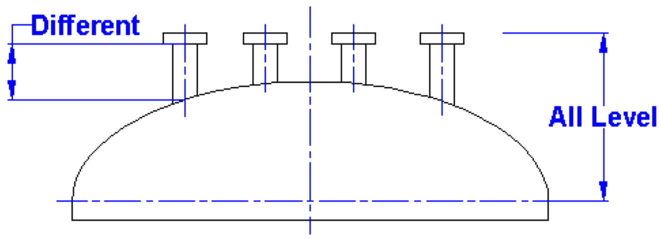


For a nozzle on a head:



Hillside Nozzles

Hillside nozzles in heads are often arranged with flange faces level in the same plane. Due to the curvature of the head, each nozzle has a *different* value for outside projection *ho*:



Because the value of **Projection Dimension "Proj"** is the *same* for each nozzle, this arrangement is simple.

NOTE Click **OK** to save parameter values and return to the **Nozzle Input/Analysis** dialog box with new orientation values displayed.

Pad or Hub Properties (Nozzle Main Tab)

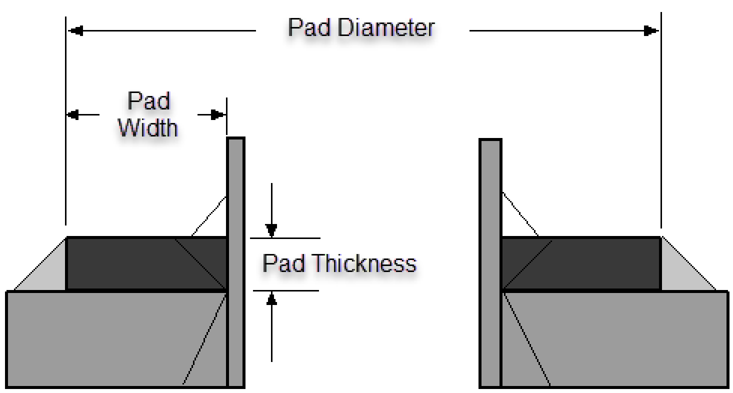
Pad Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Pad Diameter / Width

Enter the diameter of the pad. The diameter of the pad is the length along the vessel shell, *not* the projected diameter around the nozzle.

Alternatively, you can enter in the width of the pad. The software then calculates the pad diameter.

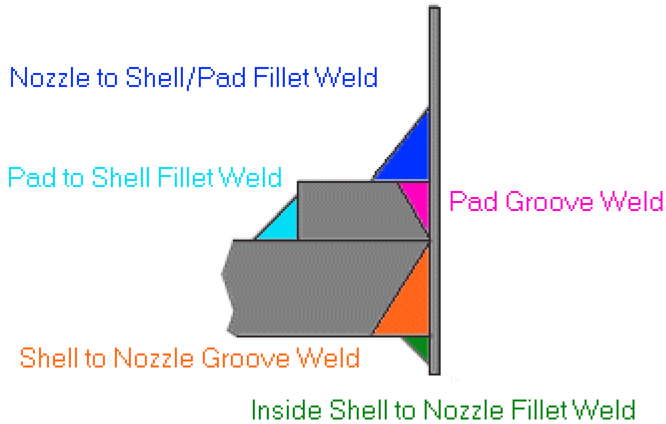


Pad Thickness

Enter the thickness of the pad. Any allowances for external corrosion should be taken into account for the pad thickness.

Groove Weld Depth

Enter the total depth of the groove weld between the pad and the nozzle neck. Most groove welds are full penetration welds. The depth of the weld is the same as the thickness of the pad. If the pad is attached with a partial penetration weld, enter the depth of the partial penetration. If the pad is attached with a fillet weld, enter zero.



Weld Leg at Pad OD

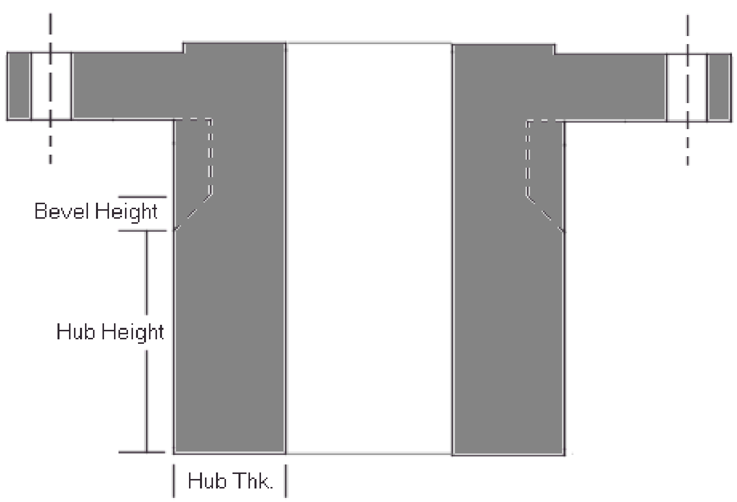
Enter the size of one leg of the fillet weld between the pad outside diameter and the shell. If any part of this weld falls outside the diameter limit, only the part of the weld inside the diameter limit is included in the available area.

For VIII-1 split pads, multiply A5 by 0.75 per UG-37(h)

Select to indicate that, with ASME VIII-1 split pads, the software multiplies area A5 by .75 for every UG-37 (h) used. ASME VIII-1 Nozzle F Factor specifies the nozzle F factor in paragraph UG-37 of ASME VIII-1.

Hub Thickness

Enter the thickness of the thicker part of the nozzle at the base. This value is equal to (Nozzle OD at the base - Nozzle ID at the base)/2.



Hub Height

Enter the hub height. This value is equal to (overall length - nut relief height - flange thickness - raised face dimension).

Bevel Height

Enter the bevel height.

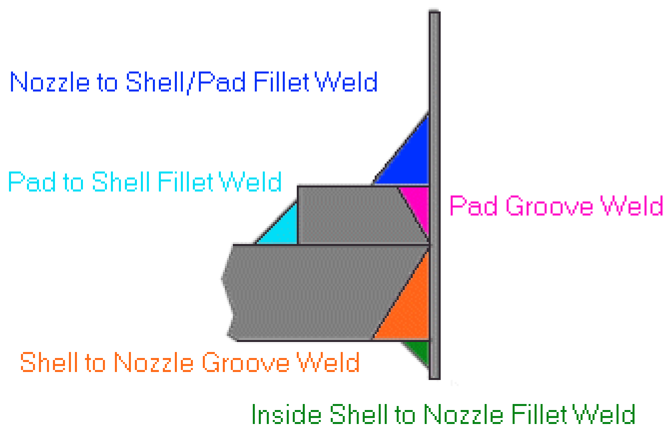
Obtain ASME Bevel Angle

Obtain the ASME bevel Angle.

Additional Weld Data (Nozzle Main Tab)

Nozzle to Pad/Shell Outside Fillet Weld Leg

Enter the size of one leg of the fillet weld between the nozzle and the pad or the nozzle and the outside shell.



Nozzle to Shell Inside Fillet Weld Leg

Enter the size of one leg of the fillet weld between the inward nozzle and the inside shell.

Nozzle to Shell Groove Weld Depth

Enter the total depth of the groove weld between the nozzle and the vessel. Most groove welds are full penetration welds. The depth of the weld is the same as the thickness of the pad. If the pad is attached with a partial penetration weld, enter the depth of the partial penetration. If the pad is attached with a fillet weld, enter zero.

ASME VIII-1 Weld Type

Select one of the following ASME VIII Division 1 weld types:

(a), **(b), (c)**, **(d)**, **(e)**, **(f-1)**, **(f-2)**, **(f-3)**, **(f-4)**, **(g)**, **(x-1)**, **(y-1)**, **(z-1)** - For these weld detail sketches, according to UW-16.1, the software does not perform the weld strength calculation. In these cases, the code does not require weld strength/path calculations for full penetration groove welds for pressure loadings.

(a-1), (i), **(j), (k)**, **(l)**, **(q)**, **(x-2)**, **(y-2)**, **(z-2)** - For these weld types, the software performs the additional weld size calculations according to UW-16(d)(1).

None - The software performs the calculation regardless of the type of weld.

NOTES

- Some sketches, such as UW-16.1 (k) or UG-40 (n), show that the nozzle does not completely extend to the inner surface of the shell. In these cases, be sure to enter the correct value for **Projection Inside**.
- These options are only available when you select **Division 1** as the **Design Code**.

Miscellaneous (Nozzle Main Tab)

Flange Class

Select the pressure rating class for the ANSI B16.5 or DIN flange, based on the pressure rating class for the attached nozzle. Select **150**, **300**, **400**, **600**, **900**, **1500**, **2500**, **NP6**, **NP10**, **NP24**, **NP40**, **NP63**, **NP100**, or **None**.

Flange Grade

Select the flange material grade (group). Please note that there are certain advisories on the use of certain material grades. Please review those cautionary notes in the ANSI B16.5 code. ASME B16.5-2003 and ASME B16.5-1996 flange grades are available:

Table 1A List of Material Specifications (ASME B16.5-2003)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V 3½ Ni	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1 A 350 Gr. LF3	A 216 Gr. WCB	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 2½Ni 3½Ni	A 350 Gr. LF 6 Cl.2	A 316 Gr. WCC A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E
1.3	C-Si C-Mn-Si 2 ½Ni 3 ½Ni C-½Mo		A 352 Gr. LCB A 217 Gr. WC1 A 352 Gr. LC1	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1		A 204 Gr. A A 204 Gr. B
1.7	½C-½Mo Ni-½Cr-½Mo ¾Ni-¾Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1¼Cr-½Mo 1¼Cr-½Mo-Si	A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	2¼Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2

Material Group	Nominal Designation	Forgings	Castings	Plates
1.11	Cr-½Mo			A 204 Gr. C
1.13	5Cr-½Mo	A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
1.17	1Cr-½Mo 5Cr-½Mo	A 182 Gr. F12 Cl.2 A 182 Gr. F5		
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H A 182 Gr. F317	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H		A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	23Cr-12Ni			A 240 Gr. 309H
2.7	25Cr-20Ni	A 182 Gr. F310		A 240 Gr. 310H
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
2.9	23Cr-12Ni 25Cr-20Ni			A 240 Gr. 309S A 240 Gr. 310S
2.10	25Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	
2.11	18Cr-10Ni-Cb		A 351 Gr. CF8C	
2.12	25Cr-20Ni		A 351 Gr. CK20	
3.1	35Ni-35Fe-10Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020

Material Group	Nominal Designation	Forgings	Castings	Plates
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe 64Ni-29.5Mo-2Cr-2Fe-Mn-W	B 462 Gr. N10665 B 462 Gr. N10675		B 333 Gr. N10665 B 333 Gr. N10675
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu 55Ni-21Cr-13.5Mo 55Ni-23Cr-16Mo-1.6Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825 B 462 Gr. N06022 B 462 Gr. N06200		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825 B 575 Gr. N06022 B 575 Gr. N06200
3.9	47Ni-22Cr-9Mo-I8Fe	B 572 Gr. N06002		B 435 Gr. N06002
3.10	25Ni-46Fe-21Cr-5Mo	B 672 Gr. N08700		B 599 Gr. N08700
3.11	44Fe-25Ni-21Cr-Mo	B 649 Gr. N08904		B 625 Gr. N08904
3.12	26Ni-43Fe-22Cr-5Mo 47Ni-22Cr-20Fe-7Mo 46Fe-24Ni-21Cr-6Mo-Cu-N	B 621 Gr. N08320 B 581 Gr. N06985 B 462 Gr. N08367	 A 351 Gr. CN3MN	B 620 Gr. N08320 B 582 Gr. N06985 B 688 Gr. N08367
3.13	49Ni-25Cr-18Fe-6Mo Ni-Fe-Cr-Mo-Cu-Low C	B 581 Gr. N06975 B 462 Gr. N08031		B 582 Gr. N06975 B 625 Gr. N08031
3.14	47Ni-22Cr-19Fe-6Mo 40Ni-29Cr-15Fe-5Mo	B 581 Gr. N06007 B 462 Gr. N06030		B 582 Gr. N06007 B 582 Gr. N06030
3.15	33Ni-42Fe-21Cr	B 564 Gr. N08810		B 409 Gr. N08810
3.16	35Ni-19Cr-1¼Si	B 511 Gr. N08330		B 536 Gr. N08330
3.17	29Ni-20.5Cr-3.5Cu-2.5Mo		A 351 Gr. CN7M	

Table 1A List of Material Specifications (ASME B16.5-1996)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1	A 216 Gr. WCB A 216 Gr. WCC	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1

Material Group	**Nominal Designation**	**Forgings**	**Castings**	**Plates**
1.2	C-Mn-Si C-Mn-Si-V 21/2Ni 31/2Ni	A 350 Gr. LF 6 Cl.2 A 350 Gr. LF3	A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E
1.3	C-Si C-Mn-Si 21/2Ni 31/2Ni		A 352 Gr. LCB	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1	A 217 Gr. WC1 A 352 Gr. LC1	A 204 Gr. A A 204 Gr. B A 204 Gr. C
1.7	C-1/2Mo 1/2Cr-1/2Mo Ni-1/2Cr-1/2Mo 3/4Ni-3/4Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1Cr-1/2Mo 11/4Cr-1/2Mo 11/4Cr-1/2Mo-Si	A 182 Gr. F12 Cl.2 A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	21/4Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.13	5Cr-1/2Mo	A 182 Gr. F5 A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H	A 351 Gr. CF8C	A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	25Cr-12Ni 23Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	A 240 Gr. 309S A 240 Gr. 309H

Material Group	Nominal Designation	Forgings	Castings	Plates
2.7	25Cr-20Ni	A 182 Gr. F310	A 351 Gr. CK20	A 240 Gr. 310S A 240 Gr. 310H
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53 A 182 Gr. F55	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
3.1	35Ni-35Fe-20Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe	B 335 Gr. N10665		B 333 Gr. N10665
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825

Flange Material

Enter the name of the material. This software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. Alternatively, you can click **Matl...** to select a material directly from the **Material Database** dialog box.

Flange Type

Select the flange type: **Weld Neck**, **Long WN**, **Lap Joint**, **Slip On**, **Socket Weld**, **Threaded**, **Studding Outlet**, **FFWn**, **FFSo**, **FFThrd**, **RTJWn**, **Clpg-Thrd**, **Clpg-Sw**, or **None.** This value is not used in the analysis, but is printed in reports.

Neglect Areas

Select how the area contributed by the shell or nozzle is handled. Some vessel design specifications mandate that no credit be taken for the area contributed by the shell or nozzle. Select one of the following:

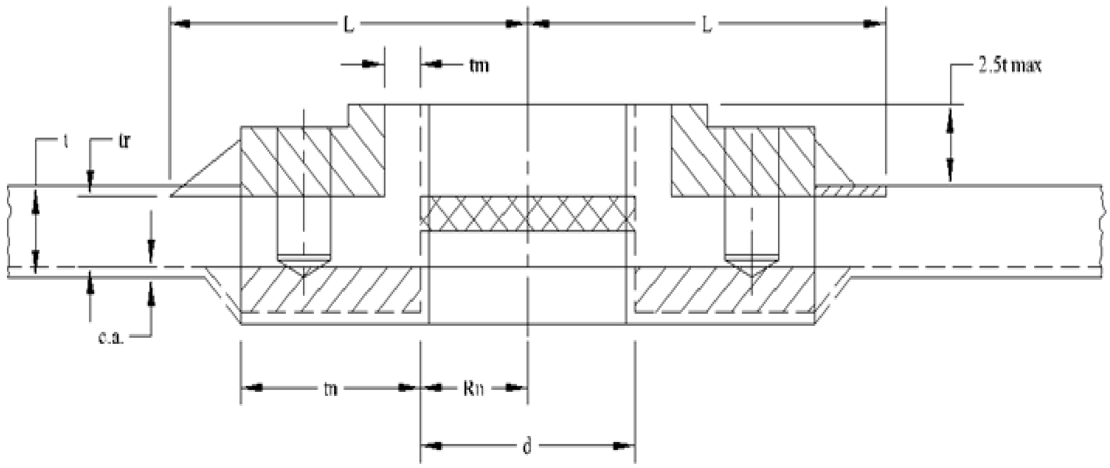
- **A1 -** Exclude the available area in the vessel wall.
- **A2 -** Exclude available area in the nozzle wall.
- **A1 A2 -** Exclude the available areas in both the vessel and nozzle walls.

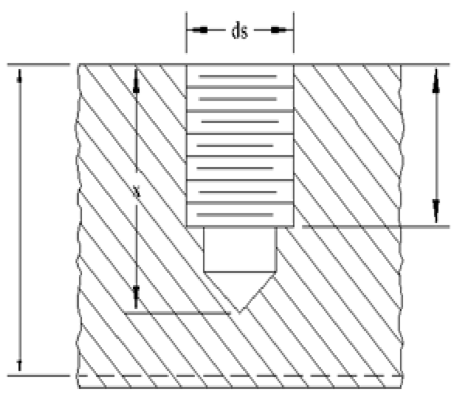
- **ACWLD** - Exclude the available area in the cover weld.
- **None** - Include all areas.

Tapped Hole Area Loss

Enter the area *S* to exclude when holes are tapped into studding outlets and other similar connection elements. The traditional industry standard is to increase the area required by the tapped hole area loss. Values for tapped area loss are shown in the table below, adapted from the *Pressure Vessel Design Manual:*

ds (in)	S (in^2)
5/8	**1.280**
3/4	**1.840**
7/8	**2.500**
1	**3.280**
1 1/8	**4.150**
1 1/4	**5.120**
1 3/8	**6.200**
1 1/2	**7.380**
1 5/8	**8.660**
1 3/4	**10.05**
1 7/8	**11.55**
2	**13.10**
2 1/4	**16.60**





Nozzle Efficiency

Enter the longitudinal seam efficiency of the nozzle neck. For nozzle wall thickness calculations, the seam efficiency is always **1.0**. For more information see the definition of E in the ASME Code, paragraph UG-37, and Interpretation VIII-89-171 of the A-90 addenda.

In the 2015 edition of the ASME VIII-1 Code, it was made clear that for wall thickness calculations, allowable stresses based on seamless material are to be used. For welded pipe and tube material that refers to note G24 in ASME VIII-1, the stress values will be divided by 0.85.

The nozzle material joint efficiency is used to reduce the area available in the nozzle wall per figure UG-37.

Shell Eff.

Enter the joint efficiency of the shell seam. The seam efficiency is used in "area available" calculations to reduce the area available in the shell. For shell wall thickness calculations, the seam efficiency is always **1.0**. The software accounts for the case where the nozzle passes through a weld by asking for the joint efficiency of the weld.

C Factor

Enter the PD 5500 C factor used in nozzle compensation calculations. This factor accounts for the possibility of external loads. When external loads are negligible, the C factor should not be more than **1.1**. When the nozzle is connected to a piping system, where forces and moments are considered, the C factor should not be greater than **1.0**. For vessels operating in the creep range, C should be less than or equal to **1**.

Local Shell Thk.

If your vessel has insert plates, enter the thickness of the plate. Use this value for vessels having insert plates that are thicker than the surrounding shell. This value is greater than the course thickness of the shell. If the area immediately adjacent to the opening is corroded to a greater degree or locally thinner than the rest of the shell, enter the thinner value. The greater of this value and the element thickness is used in nozzle reinforcement calculations.

User Tr

Enter the minimum required shell thickness *tr*, the actual thickness of the shell or head minus the corrosion allowance. For some vessel designs, the nozzle reinforcement is governed by bending and normal stresses in the local shell area. Under special conditions, project requirements specify that full area replacement for nozzle reinforcement is required. This value replaces the value that the software normally calculates for the required thickness based on internal or external pressure.

NOTE Optionally, for vertical vessels, select **Consider External Loads for Nozzle Tr** in the **Nozzle Design Options** section of the **Load Cases** tab. The software determines the maximum thickness based on the highest stress ratio and uses that value for *tr* if it governs over the required thickness based on internal or external pressure.

Blind Attached?

Select if there is a blind on the nozzle flange. This is used only to determine the weight of the nozzle. There is no structural effect.

Manway/Acs Ope?

Select if this is a manway, access, or inspection opening to bypass the UG-45 minimum nozzle neck thickness requirement. In these cases, paragraph UG-45 states that the minimum thickness requirement according to UG-45 is not required. This option is not used for PD 5500.

Fatigue Calc?

Select to perform a fatigue analysis of the nozzle-to-shell weld. Along with specification of **Weld Class** for PD 5500 and EN13445 models, the number and magnitude of fatigue pressure stress cycles must be specified in **Fatigue Analysis** on the **Load Cases** tab.

The table below provides descriptions for ASME VIII-2 2007 and later fatigue curves.

Table	Description
Table 3-F.1	Fatigue Curve for Carbon, Low Alloy, Series 4XX, High Alloy, and High Tensile Strength Steels for Temperatures Not Exceeding 700°F — σuts ≤ 80 ksi

Table	Description
Table 3-F.2	Fatigue Curve for Carbon, Low Alloy, Series 4XX, High Alloy, and High Tensile Strength Steels for Temperatures Not Exceeding 700°F — σuts = 115 ksi to 130 ksi
Table 3-F.3	Fatigue Curve for Series 3XX High Alloy Steels, Nickel–Chromium–Iron Alloy, Nickel–Iron–Chromium Alloy, and Nickel–Copper Alloy for Temperatures Not Exceeding 800°F
Table 3-F.4	Fatigue Curve for Wrought 70–30 Copper–Nickel for Temperatures Not Exceeding 700°F — σys $\leq$ 18 ksi
Table 3-F.5	Fatigue Curve for Wrought 70–30 Copper–Nickel for Temperatures Not Exceeding 700°F — σys = 30 ksi
Table 3-F.6	Fatigue Curve for Wrought 70–30 Copper–Nickel for Temperatures Not Exceeding 700°F — σys = 45 ksi
Table 3-F.7	Fatigue Curve for Nickel–Chromium–Molybdenum–Iron, Alloys X, G, C-4, and C-276 for Temperatures Not Exceeding 800°F

Weld Class

Select a weld class when **Fatigue Calc?** is selected. The weld classes, descriptions, and illustrations are found in PD 5500 Annex C. Class C is the least severe, while Class W is the most severe. This entry is only available for PD 5500 and EN 13445 models.

Piping Attached

Click to open the **Drain Piping Input Dialog Box** (page 85) to set the nozzle piping coordinates.

Derate Flange MAWP if Externally Loaded?

Check this option to allow a particular nozzle ANSI flange to have its MAWP derated if external loads are present. Use of the flange derating is up to the discretion and experience of the designer. Select the ANSI Flange Pressure Reduction Options method in the **Load Cases** tab under the **Nozzle Design Options**.

Quick Results

Click  to see a report of nozzle results.

Drain Piping Input Dialog Box

Specifies coordinates for the drain piping (nozzle) extending from the head element.

Piping Attached to this Nozzle?

Select to enable the drain piping input boxes on the dialog box.

Piping Exit Angle [Alpha]

Enter the angle (in degrees) at which the nozzle piping extends from the head. Use the Plan View diagram to determine the layout angle. For nozzles on the bottom of the vessel, the angle degrees increase in a clockwise direction. For nozzles on the top of the vessel, the angle degrees increase in a counter-clockwise direction.

NOTE The Plan View diagram is a top view of the vessel. The dashed-line nozzle indicates that the nozzle is on the bottom of the vessel.

Bend Radius Multiplier

Enter a value to indicate the amount of bend on the radius of the nozzle piping. The greater the value you enter, the greater the bend of the piping.

Horizontal Run Length [xR]

Enter the length of the nozzle piping as indicated in the Plan View diagram.

Centerline Distance to Tangent [xL]

Enter the distance from the centerline of the nozzle piping to the tangent of the head element as indicated in the Side View diagram.

Local Stress Analysis (WRC 107, 297 or Annex G) Tab (Nozzle Input/Analysis Dialog Box)

Defines loading information for nozzle stress analysis. Local loads are entered for stress analysis according to the British code PD 5500. Local or global loads are entered for stress analysis according to the WRC 107/537 and WRC 297 bulletins. The software also checks global loads against ASME VIII-2 allowables. Loads are categorized as sustained, expansion, and occasional.

NOTE In 2010 WRC bulletin 537 was released. The results of the local stress calculation of this bulletin are effectively identical to that of WRC bulletin 107. Bulletin 537 simply provides equations in place of the dimensionless curves found in bulletin 107. Please review the Forward in bulletin 537 for more information.

Calculation Method

Select the needed analysis method. The appropriate force and moment fields are available for the selected method. Select **PD 5500**, **WRC 107**, **WRC 297**, or **No Calc**. When you change the method, the software automatically converts entered loads into the respective coordinate systems used by each method.

Load Convention System

Select **Local** to defined loads locally with respect to the vessel and the nozzle, having the benefit of being independent of the orientation of the vessel. Select **Global** to use the global coordinate system. This entry is only available when **WRC 107/537** is selected for **Calculation Method**.

Quick Results

Click to see a report of local stress analysis quick results.

Nozzle Load Table

Click to open and edit a Microsoft Office Excel workbook with standard nozzle loads that can be imported into the software. You can modify the nozzle loads for Project A, Australian, and EN projects in this file, or create your own loading project. These loads are typically listed in the specifications of many engineering companies. After the needed nozzle loads are entered in the workbook, then these loads can be imported and applied to a nozzle depending upon its size and rating. This saves time and reduces chances of error. A default sample workbook is provided.

Loading

Click to select the loading project from the Excel load table workbook. Select **Project A Loadings** or **Australian User Loadings**, **EN-User Loadings,** or a user created loading project.

CAESAR II

Click to import nozzle loads from a CAESAR II .C2 file. This command is only available when **Global** is selected for **Load Convention System** and when CAESAR II is installed. CAESAR II is a separately-purchased Intergraph product.

Node #

Enter the node number of the nozzle used in CAESAR II.

Computed Stress Intensities/Ratios at the Nozzle Edge and Pad Edge

Displays the calculated stress intensities and stress ratios for **Vessel at Nozzle Edge**, **Vessel at Pad Edge**, and **In Nozzle at Vessel**. For example:

	Computed Stress Intensities/Ratios at the nozzle edge and pad edge	Pass/Fail Status
Vessel at Nozzle Edge :	Maximum calculated stress ratio: 0.323	Passed
Vessel at Pad Edge :	Maximum calculated stress ratio: 0.644	Passed

PD 5500 Annex G Analysis **(page 87)**

WRC 107/537 Analysis - Local Load Convention **(page 88)**

WRC 107/537 Analysis - Global Load Convention **(page 91)**

WRC 297 Analysis **(page 92)**

PD 5500 Annex G Analysis

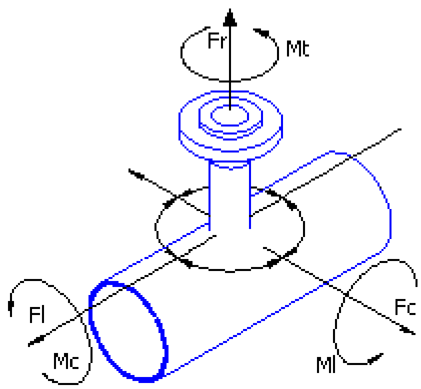
Enter values for the following options when **PD 5500** is selected for **Calculation Method**. The software calculates stresses in cylindrical or spherical geometries with or without reinforcing pads. Only stresses for round hollow nozzle geometries are calculated.

IMPORTANT The example in PD 5500 Annex W does not calculate the membrane stress at attachment edge; you must manually check the membrane stress when selecting the wall thickness. In addition, the membrane stress calculated at the attachment edge, according to Annex G, contains intensified stresses due to the presence of the hole.

Loads

Sustained loads (weight+pressure+forces) are used.

The following force/moment convention is used for PD 5500, Annex G:



Fr - Radial Force
Fc - Shear Force
Fl - Shear Force
Mc - Moment, Fl axis
Ml - Moment, Fc axis
Mt - Torsional moment

Length "L"

Enter in the length of the vessel. For vessels without stiffeners or cones this is the entire vessel length including heads. This value is used along with **Tangent Offset Distance** to calculate the equivalent length for off center loading.

Tangent Offset Distance

Enter in the distance of the centerline of the nozzle from the left tangent line or appropriate line of support. This value is used in conjunction with the **Length "L"** to calculate the equivalent length for off center loading.

Allowable Stress Intensity Factors at Nozzle Edge

Factor for Membrane Stresses

Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane stress intensity. These stresses are in rows 32, 33 and 34 in the printout samples in PD 5500 Annex W. This factor normally has a value of **1.2** or lower at the edge of the reinforcement pad.

NOTES

- This factor is higher than **Factor for Memb Stresses** at pad edge.
- This entry is only available when **Print Membrane Stress at Nozzle Edge** is selected.

Factor for Membrane + Bend Stresses

Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane plus bending stress intensity. These stresses are in rows 27, 28 and 29 in the printout samples in PD 5500 Annex G. This factor normally has a value of **2.25** or lower. At the pad edge, this factor is normally **2.0**.

Print Membrane Stress at Nozzle Edge

Select to calculate membrane stress at the attachment junction. You must also enter a value for **Factor for Membrane Stresses**.

Allowable Stress Intensity Factors at Pad Edge

Factor for Memb Stresses

Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane stress intensity. These stresses are in rows 32, 33 and 34 in the printout samples in PD 5500 Annex W. This entry normally has a value of **1.2** or lower at the edge of the reinforcement pad.

Factor for Memb + Bend Stresses

Enter the stress intensity factor. This factor is multiplied by the allowable stress *f* to obtain the allowable stress for the maximum membrane plus bending stress intensity. These stresses are in rows 27, 28 and 29 in the printout samples in PD 5500 Annex G. At the nozzle edge, this factor normally has a value of **2.25** or lower. At the pad edge, this factor is normally **2.0**.

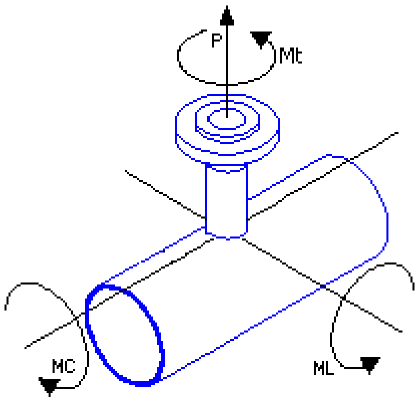
WRC 107/537 Analysis - Local Load Convention

Enter values for the following options when **WRC 107/537** is selected for **Calculation Method** and **Local** is selected for **Load Convention System**. **Local** has the benefit of being independent of the orientation of the vessel. The software calculates stresses for sustained, expansion, and occasional loads and compares stress intensities to allowables.

You can enter values in the following load sets:

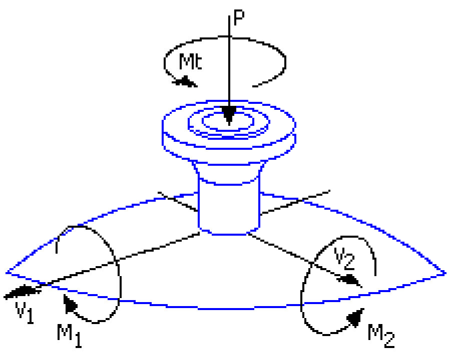
- **Sustained** - (SUS) Primary loads, typically weight + pressure + forces.
- **Expansion** - (EXP) Secondary thermal expansion loads.
- **Occasional** - (OCC) Irregularly occurring loads such as wind loads, seismic loads, and water hammer.

The following WRC 107/537 force/moment convention is used for a **cylindrical** vessel:



P - Radial Force
Vc - Circ. Shear Force
Vl - Long. Shear Force
Mc - Circ. Moment
Ml - Long. Moment
Mt - Torsional Moment

The following WRC 107/537 force/moment convention is used for a **spherical** vessel:



P - Radial Force
V2 - Shear Force, D to C
V1 - Shear Force, B to A
M1 - Moment, B axis
M2 - Moment, C axis
Mt - Torsional moment

Occasional Press Difference

Enter the *difference* between the peak pressure of the system and the system design pressure. The value is always positive or zero. This value is superimposed onto the system design pressure to evaluate the primary membrane stress due to occasional loads. The additional thrust load due to this pressure difference is accounted for in the nozzle radial loading if you also select **Include Pressure Thrust**.

Include Pressure Thrust

Select to include the pressure thrust force (P*A) in the nozzle axial load. For more information on pressure thrust, see the *July 2001 COADE Newsletter http://www.coade.com/Uploads/mechanical-engineering-news/jul01.pdf*.

Use Division 2 Stress Indices

Select to include the pressure stress indices described in ASME Sec. VIII Div. 2 Table AD-560.7. This value should *only* be used to perform a fatigue analysis. Check ASME VIII Div.2 paragraph AD-160 s to see if the fatigue effect needs to be considered. These factors are used for estimating the peak stress intensity due to internal pressure.

NOTES

- Peak stress intensity due to external loads is included in the analysis by selecting **Use Kn and Kb (to find SCF)**. For normal (elastic) analysis, do not select this option or **Use Kn and Kb (to find SCF).**
- The software does *not* perform the complete fatigue analysis of Section VIII Div.2 Appendix 4 and 5 rules. Instead, the value of peak stress intensity is reported for fatigue effect comparison. For more information, see the *June 2000 COADE newsletter http://www.coade.com/Uploads/mechanical-engineering-news/jul01.pdf*.

Use WRC 368

Select to compute pressure stresses in the shell and nozzle according to WRC 368. WRC 368 provides a method for calculating stresses in a cylinder-to-cylinder intersection (such as cylinder-to-nozzle) due to internal pressure and pressure thrust loading.

NOTES

- Using WRC 368 with WRC 107/297 is not accurate for calculating the combined stress from pressure and external loads. So, this option is only available when the attachment type is round and when no external loads are specified.
- For more information on WRC 368 and pressure thrust, see *Modeling of Internal Pressure and Thrust Loads on Nozzles Using WRC-368* in the *July 2001 COADE Newsletter http://www.coade.com/Uploads/mechanical-engineering-news/jul01.pdf*.

Use Kn and Kb (to find SCF)

Select to include the WRC 107/537 Appendix B stress concentration factors (Kn and Kb) in a fatigue analysis. This value should *only* be used to perform a fatigue analysis. Check ASME VIII Div.2 paragraph AD-160 s to see if the fatigue effect needs to be considered. Also enter the needed value for **Fillet Radius Nozzle** or **Fillet Radius Pad**.

NOTES

- Peak stress intensity due to internal pressure is included in the analysis by selecting **Include Pressure Stress Indices per Div. 2?**.
- For normal (elastic) analysis, do not select this option or **Use Division 2 Stress Indices.**
- The software does *not* perform the complete fatigue analysis of Section VIII Div.2 Appendix 4 and 5 rules. Instead, the value of peak stress intensity is reported for fatigue effect comparison. For more information, see *WRC-107 Elastic Analysis v/s Fatigue Analysis* in the *June 2000 COADE newsletter http://www.coade.com/Uploads/mechanical-engineering-news/jul01.pdf*.

Fillet Radius Nozzle

Enter the fillet radius between the nozzle and the vessel shell. The software uses this value to calculate the stress concentration factors Kn and Kb according to Appendix B of the WRC 107 bulletin. A value of **0** sets Kn and Kb to 1.0.

Fillet Radius Pad

Enter the fillet radius between the pad and the vessel shell. The software uses this value to calculate the stress concentration factors Kn and Kb for the vessel/pad intersection, according to Appendix B of the WRC 107 bulletin. A value of **0** sets Kn and Kb to 1.0.

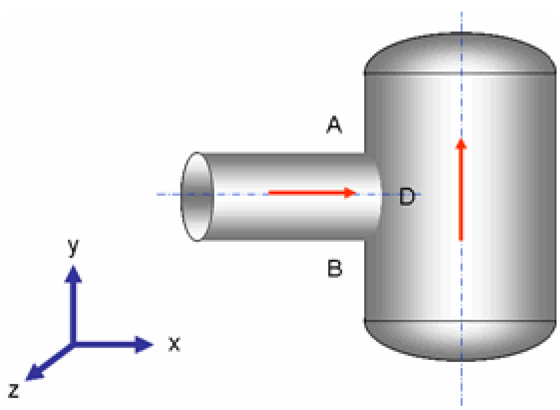
WRC 107/537 Analysis - Global Load Convention

Enter values for the following options when **WRC 107/537** is selected for **Calculation Method** and **Global** is selected for **Load Convention System**. **Global** has the benefit of using the global coordinate system also used by other analyses. When you toggle between the global and local convention systems, the software converts the loads. Options for the global load convention are the same as for **WRC 107/537 Analysis - Local Load Convention (page 88),** except as described below.

Direction Cosines

Displays the direction cosines as described below. You do not usually need to change these values.

The following global convention system is used for a **cylindrical** vessel:



The vessel direction is +Y direction
The nozzle direction is +X direction (towards the vessel)

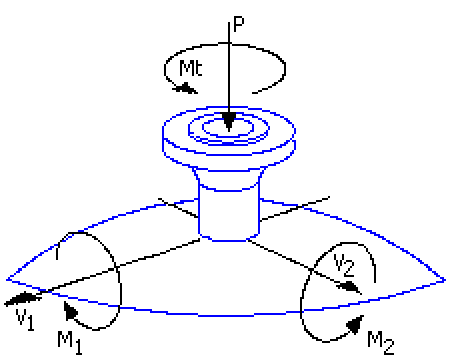
Direction cosines of the vessel are:

- **VX** - 0
- **VY** - 1
- **VZ** - 0

Direction cosines of the nozzle are:

- **NX** - 1
- **NY** - 0
- **NZ** - 0

The following global convention system is used for a **spherical** vessel:



The direction of a spherical vessel is from points **B** to **A**

The software uses these direction vectors to transfer the global forces and moments from the global convention into the traditional WRC107 convention.

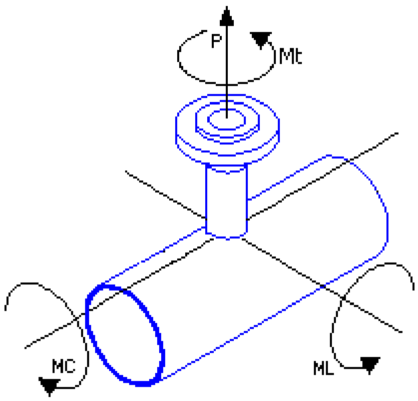
WRC 297 Analysis

Enter values for the following options when **WRC 297** is selected for **Calculation Method**. The software calculates stresses in cylindrical nozzles attached to cylindrical shells.

Loads

Sustained loads (weight+pressure+forces) are used.

The following force/moment convention is used for WRC 297:



P - Radial Force
VC - Shear Force, D to C
VL - Shear Force, B to A
MC - Moment, B axis
ML - Moment, C axis
Mt - Torsional moment

Stress Concentration Factors

Shell Stress Concentration Factor

Enter a value, typically between **1** and **3**, for stress concentration due to weld quality and dimensions in the immediate vicinity of the weld. The stress concentration factor:

- Accounts for *peak stresses* - local stress risers in the immediate vicinity of vessel welds due to factors such as sharp corners and lack of fillet weld radii. Peak stresses are considered in fatigue analysis.
- Applies to the stress calculations in the vessel and the nozzle on both the inside and the outside of the vessel.
- Is used in pressure stress calculations in the vessel on both the inside and outside of the vessel.

Nozzle Stress Concentration Factor

Enter a value, typically between **1** and **3**, for stress concentration due to weld quality and dimensions in the immediate vicinity of the weld. The stress concentration factor:

- Accounts for *peak stresses* - local stress risers in the immediate vicinity of vessel welds due to factors such as sharp corners and lack of fillet weld radii. Peak stresses are considered in fatigue analysis.
- Applies to the stress calculations in the vessel and the nozzle on both the inside and the outside of the vessel.
- Is not used in pressure stress calculations.

Include Pressure Thrust

Select to include the pressure thrust force (P*A) in the nozzle axial load. For more information on pressure thrust, see the *July 2001 COADE Newsletter http://www.coade.com/Uploads/mechanical-engineering-news/jul01.pdf*.

Use Division 2 Stress Indices

Select to include the pressure stress indices described in ASME Sec. VIII Div. 2 Table AD-560.7. This value should *only* be used to perform a fatigue analysis. Check ASME VIII Div.2 paragraph AD-160 s to see if the fatigue effect needs to be considered. These factors are used for estimating the peak stress intensity due to internal pressure.

NOTES

- Peak stress intensity due to external loads is included in the analysis by selecting **Use Kn and Kb (to find SCF)**. For normal (elastic) analysis, do not select this option or **Use Kn and Kb (to find SCF).**
- The software does *not* perform the complete fatigue analysis of Section VIII Div.2 Appendix 4 and 5 rules. Instead, the value of peak stress intensity is reported for fatigue effect comparison. For more information, see the *June 2000 COADE newsletter http://www.coade.com/Uploads/mechanical-engineering-news/jul01.pdf*.

Weight

Home tab: **Details** > **Weight**

Adds piping and miscellaneous weight to the selected cylinder or head element. This is weight that cannot be accounted for in other commands. The weight is a static mass, not an applied force, but affects the natural frequency of the vessel and axial stress calculations. Piping weight is modeled here. The area and mass of the piping are considered in the same manner as a weight.

Previous Weight

If you created more than one weight on the element, click to go back to the previous weight.

Go To Next Weight

If you created more than one weight on the element, click to go to the next weight.

Add New Weight

Click to add a new weight to the shell or head element.

Delete

Deletes all data for the current weight.

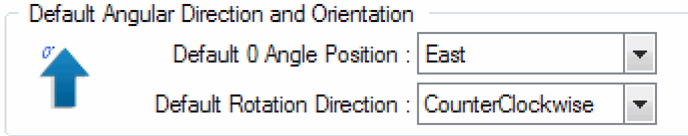
Common Detail Parameters (page 54)

Layout Angle

Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the weight at the point where it is attached to the shell.

For a horizontal vessel, the zero degree line is at 12:00 (looking at a clock); 90 degrees is at 3:00. Entering these layout angles is important if the horizontal vessel has a liquid level and the nozzles are being designed using **Design P + Static Head**, selected for **Nozzle/Clip Design Pressure Options** on the **Load Cases** tab. For a vertical vessel, the angle is more arbitrary. For purposes of rendering the graphics, the assumption is that the zero degree line is at 3:00 and 90 degrees is 12:00.

The position of the zero degree reference direction (North, West or East) and the angular rotation (clockwise or counterclockwise) are set in **Tools** tab, **Set Configuration Parameters**, *Default Value Settings Tab (Configuration Dialog)* (page 233).



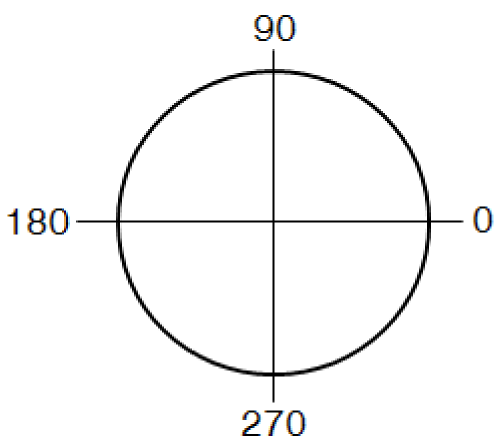
If the new orientation is different from the orientation of the current model, the software asks to update the model with the new settings. If the settings are updated, the weights and areas of platforms must be recalculated:

- Click **List Dialog** (page 218). On the **Detail Listing** dialog box, click the **Platform** tab so that platform data is visible. Click **Platform Wind Area** and **Platform Weight** to recalculate.
- Click **Platform Input** for each platform. Tab through the fields in the **Platform** dialog box, causing the weight and area to be recalculated

NOTE The angular settings apply to nozzles, clips, legs, lifting lugs, support lugs, base rings, platforms, weights and half-pipe jackets.

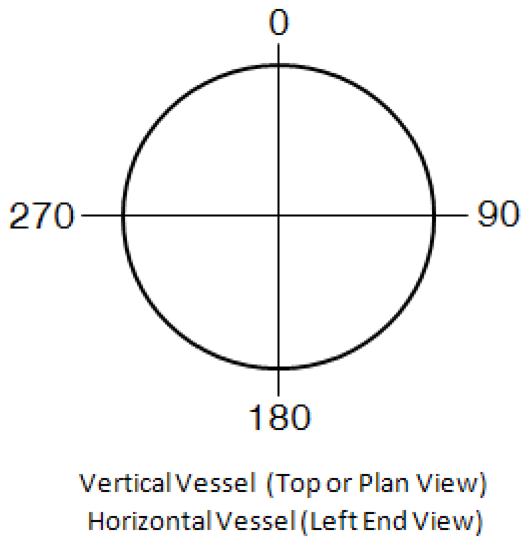
Examples

Default Orientation



Vertical Vessel (Top or Plan View)
Horizontal Vessel (Left End View)

Angle Position: North
Rotation Direction: Clockwise



Offset from Element Centerline

Enter the distance of this weight from the centerline of vessel. The value is multiplied by the weight to obtain a moment that used in stress calculations. For horizontal vessels, the weight is added to the saddle loads and this value is not used.

Miscellaneous Weight

Enter the weight for items such as: an attached motor or other equipment, internal piping, or external structural elements. This value is also used for seismic analysis.

Select the Active Cases for this Weight/Mass

Select the active case for the current weight/mass. You can select any combination of empty, operating, and hydro test cases.

Is this a Welded Internal?

Select if the weight is for an item that is welded to the vessel during shop construction. The weight is then added to the fabricated total weight, other weights that are functions of the fabricated total weight, and the empty total weight.

Area of External Weight/Piping/Equipment

Enter the area to use for the wind load calculation.

Piping Detail

Select **Is this a Piping Detail?** to include the weight and moment of overhead pipelines on vertical pressure vessels. Define the area and weight of the piping with the following options:

- **Pipe Lookup** - Click to open the **Seamless Pipe Selection** dialog box and select values from the piping database for **Pipe Schedule** and **Nominal Pipe Diameter**.
- **Pipe Outer Diameter** - Displays the OD for the pipe selected in **Pipe Lookup**. You can also manually enter a value.
- **Pipe Thickness** - Displays the nominal thickness for the pipe selected in **Pipe Lookup**. You can also manually enter a value.

- **Fluid Specific Gravity** - Enter the specific gravity of the contained fluid. This value is usually **1.0**.
- **Insulation Thickness** - If the pipe is insulated, enter the thickness of the insulation. If there is no insulation, enter **0**.
- **Insulation Density** - Enter the density of the pipe insulation to calculate the weight of the insulation and the moment effect.
- **Compute Weight and Area** - Click to calculate the pipe weight and area based on the entered values.

After weights and offsets are entered, the software calculates the overturning moment due to the eccentricity of the piping. In most designs, the piping is supported by means of braces and clips at specified intervals. The element on which the piping weight is added takes the applied load.

NOTE If piping is specified on a top head, the software attempts to graphically connect the piping to the center-most nozzle.

Force and Moment

Home tab: **Details** > **Force and Moment**

Adds external forces and moments to the selected cylinder or head element. In most cases these are operating loads imposed on the vessel, such as piping loads on nozzles.

Previous For/Mom

If you created more than one force/moment set on the element, click to go back to the previous set.

Go To Next For/Mom

If you created more than one force/moment set on the element, click to go to the next set.

Add New For/Mom

Click to add a new force/moment set to the shell or head element.

Delete

Deletes all data for the current force/moment set.

Common Detail Parameters (page 54)

Applied Forces

Enter the force in each needed direction. For **X**, positive is from left to right. For **Y**, positive is upward. For **Z**, positive is towards you. Forces perpendicular to the vessel are resolved into a single vector and applied with live load to create the worst load combination. Unlike **Weight**, applied forces are not included in seismic analysis because force does not have mass.

NOTE You should generally enter negative Y forces (downward) because these increase the loads on the saddles and other supports.

Applied Moments

Enter the moment about each needed axis. For **X**, positive is from left to right. For **Y**, positive is upward. For **Z**, positive is towards you.

Compute Stresses Due to Applied Loads

Select one or both of the following:

- **Compute Longitudinal Stresses BW normally added to the Wind Case** - Forces and moments act during wind analysis.
- **Compute Longitudinal Stresses BS normally added to the Seismic Case** - Forces and moments act during seismic analysis.

★ IMPORTANT You must select at least one option.

Force/Moment Combination Method

Select the method for resolving forces and moments about the support point:

- **SRSS** - All forces and moments act in the same direction. The software takes the absolute value of the entered loads to determine the overall effect on the structure. This method, while not technically correct, yields a conservative result for bending stresses and support calculations.
- **Algebraic** - Forces and moments act in their positive or negative directions. Loads that oppose each other tend to cancel out. If you have an accurate account of the external forces and moments, due to load conditions such as piping reactions, this option provides more accurate and less conservative results. The software sums the forces and moments in both the X and Z planes for vertical vessels. The greater of the two moments is used in calculations of the stresses and moments at the support.

Platform

Home tab: **Details** > **Platform**

Adds a platform to the selected element.

Previous

If you created more than one platform on the element, click to go back to the previous platform.

Next Platform

If you created more than one platform on the element, click to go to the next platform.

Add Platform

Click to add a new platform to the shell or head element.

Delete

Deletes all data for the current platform.

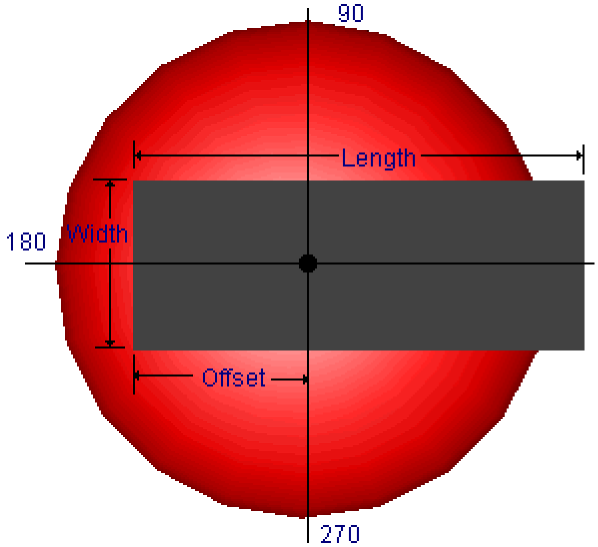
Common Detail Parameters (page 54)

Non-Circular Platform?

Select if the platform is not circular. This option is selected by default for platforms that cannot be circular, such as on a horizontal cylinder.

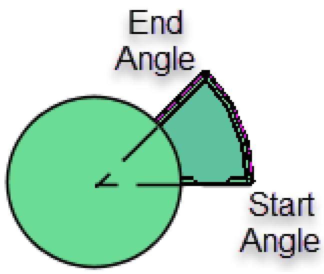
Layout Angle

For a non-circular platform, enter the rotation angle through the centerline of the platform. For example, on a non-circular top head platform:



Platform Start Angle

For a circular platform, enter the angle between the designated zero degree line of the vessel and the start angle of the platform.



Platform End Angle

For a circular platform, enter the angle between the designated zero degree line of the vessel and the ending angle of the platform.

Platform Railing Weight

Enter the weight of the platform railing in units of weight/length. This value is used as part of calculating the weight of the platform.

Platform Grating Weight

Enter the weight of the platform deck grating or plate in units of weight/area. This value is used as part of calculating the weight of the platform. To use standard weights, click ... and select **Open Lattice Grating** or **Checkered Floor Plate**.

Platform Width

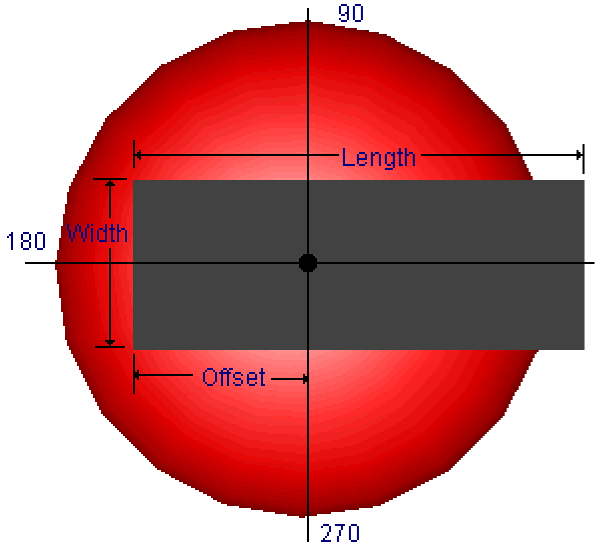
Enter the radial width of the platform. This dimension is used to calculate the weight of the platform in conjunction with **Platform Railing Weight and Platform Grating Weight.**

Platform Height

Enter the height from the bottom of the platform to the top rail. Usually this distance is to be no less than 42 inches. This dimension is used to calculate the wind area of the platform in conjunction with **Platform Width**, **Clearance**, and **Force Coefficient Cf**.

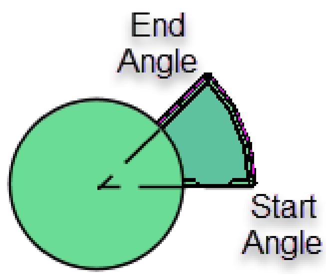
Offset from End

For a non-circular top head platform, enter the offset as shown below. The software uses the offset to calculate the distance to the center of the platform. This is then used as the offset dimension to calculate the eccentric moment for the platform. This value is used in a .DXF drawing of the vessel when **Import/Export** (page 47) is used.



Clearance

For circular platforms, enter the distance between the inside edge of the platform and the outside surface of the cylindrical shell. The platform clearance is used to calculate the wind area of the platform.



For a circular top head platform, enter **0**.

Platform Length (Non-Length)

For a non-circular top head platform, enter the long dimension of the platform.

Force Coefficient Cf

Enter the platform force coefficient *Cf*. This coefficient may be specified in a wind load computation standard, such as Table 6-9 of ASCE7-95. The value is typically between **1.2** and **1.8**. This factor is used to calculate the wind area of the platform in conjunction with the **Platform Height, Platform Width**, and **Clearance**.

Platform Wind Area

Enter the tributary wind area of the platform. This value is typically the greatest span of the platform perpendicular to the vessel multiplied by a nominal platform height of between 12 and 36 inches on the hand rails and other equipment on the platform.

To change the platform wind area calculation method, click **Installation | Misc. Options** on the **Stress Combination Load Cases (Load Cases Tab)** (page 365) of the main window. The methods are:

a. The height times the width times the force coefficient (conservative).

b. One-half of the floor plate area times the force coefficient.

c. The height times the width times the force coefficient divided by three.

d. The projected area of the platform times the force coefficient divided by three. This option yields the same results as option 3 for platforms that have a sweep angle of greater than 180º.

Control Options

Select one or more of these options:

- **Do not include platform in analysis** - Select to remove platform data from analysis.
- **User computes and enters the platform area** - Select to manually enter a value in **Platform Wind Area** and override software platform area calculations.
- **User computes and enters the platform weight** - Select to manually enter a value in **Platform and Ladder Weight** and override software platform weight calculations.

Ladder Layout Angle

Enter the angle between the designated zero degree line of the vessel and the centerline of the ladder.

Ladder Start Elevation

Enter the start elevation of the ladder. This value and **Ladder Stop Elevation** are used to determine the overall length of the ladder. The software assumes that the top of the ladder is attached to the platform.

Ladder Stop Elevation

Enter the stop elevation of the ladder. This value and **Ladder Start Elevation** are used to determine the overall length of the ladder. The software assumes that the top of the ladder is attached to the platform.

Ladder Unit Weight

Enter the unit weight of the ladder in units of weight/length. For example, in English units this is typically the weight of a one-foot ladder section. The software does not differentiate between a caged and a non-caged ladder, so the total weight of a ladder and cage must be included in this value. The total ladder and platform weight at the platform elevation is used in the calculations.

Is this a Caged Ladder?

Select if the ladder has a cage.

Platform and Ladder Weight

Enter the total weight of the platform, ladders, and associated hardware. This overrides weight calculated automatically by the software.

Packing

Home tab: **Details** > **Packing**

Adds packing data to the selected element.

Previous Packing

If you created more than one set of packing data on the element, click to go back to the previous set.

Go To Next Packing

If you created more than one set of packing data on the element, click to go to the next set.

Add New Packing

Click to add a new set of packing data to the shell or head element.

Delete

Deletes all data for the current set of packing data.

Common Detail Parameters (page 54)

Height of Packed Section

Enter the height of the packed section along the element. This value is used to calculate the weight of the packed section. For seismic calculations, the weight center of the packed section is taken at half this height. In the rare case of a packed horizontal vessel, the value is the length of the packed section.

Full

If the element is fully filled with packing material, click to calculate the values of **Distance from "From" Node** and **Height of Packed Section.**

Density of Packing

Enter the density of the packing. The following table list typical densities in lbs/ft^3:

Size (in.)	**Density (lb/ft^3)**	**Size (in.)**	**Density (lb/ft^3)**
Ceramic Raschig Ring		**Carbon Raschig Ring**	
1/4	60.0	1/4	46.0
3/8	61.0	1/2	27.0
1/2	55.0	3/4	34.0
5/8	56.0	1	27.0
3/4	50.0	1 1/4	31.0
1	42.0	1 1/2	34.0

Size (in.)	Density (lb/ft^3)	Size (in.)	Density (lb/ft^3)
1 1/4	46.0	2	27.0
1 1/2	46.0	3	23.0
2	41.0	**Carbon Steel Pall Ring**	
3	37.0	5/8	37.0
4	36.0	1	30.0
Carbon Steel Raschig Ring		1 1/2	26.0
1/4	133.0	2	24.0
3/8	94.0	**Plastic Pall Ring**	
1/2	75.0	5/8	7.25
5/8	62.0	1	5.50
3/4	52.0	1 1/2	4.75
1	39.0	2	4.50
1 1/2	42.0	3	4.50
2	37.0		
3	25.0		

Packing in place during the field hydrotest?

Select this option if the packing will be in place when the field hydrotest is performed.

Percent Volume Hold Up

Enter a percentage value between **0** and **100** for the amount of liquid that the packing retains. Using this value and **Liquid Specific Gravity**, the software calculates the weight of liquid trapped in the packing and adds the weight to the operating weight of the vessel.

Liquid Specific Gravity

Enter the specific gravity of the liquid trapped in the packing. For more information, see **Liquid** (page 124). Using this value and **Percent Volume Hold Up**, the software calculates the weight of liquid trapped in the packing and adds the weight to the operating weight of the vessel.

Saddle

Home tab: **Details** > **Saddle**

Adds a saddle to the selected horizontal cylinder element. The size and location of the saddles are important for Zick calculations of local stresses on horizontal vessels with saddle supports. For proper Zick analysis, only two saddles may be defined; however, they do not have to be symmetrically placed about the center axis of the vessel. If no saddles are defined, dead load and live load calculations are not performed.

Previous Saddle

If you created more than one saddle on the element, click to go back to the previous saddle.

Go To Next Saddle

If you created more than one saddle on the element, click to go to the next saddle.

Add New Saddle

Click to add a new saddle to the shell element.

Delete

Deletes all data for the current saddle.

Add Saddle Ring

Click to open the **Stiffening Ring** dialog box to add a ring to the saddle. For more information, see **Stiffening Ring**.

Select Saddle

Click to display and then select a source for saddle dimension data. Saddle dimension data is maintained in the *SaddleData.xls* Microsoft Excel workbook located in the PV Elite System folder. After selecting the saddle data source, the software uses the value of the inside or outside diameter from the **General Input** tab to populate the **Saddle Dialog** with the saddle dimensions.

Same as First

Click to copy all data from the first saddle to the current saddle.

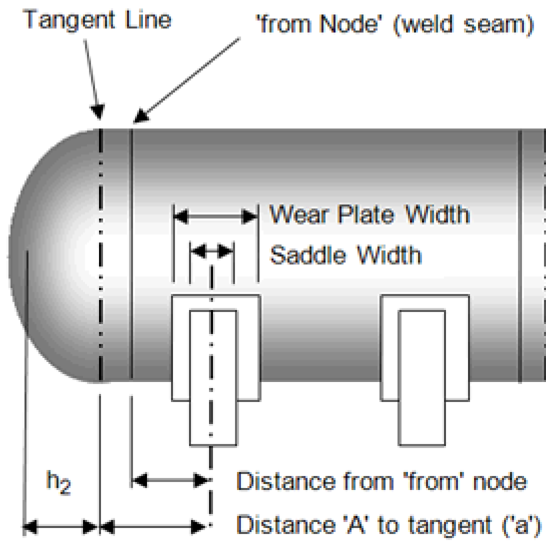
Saddle Data Table

Click to open and edit the *SaddleData.xls* Microsoft Excel workbook. Use this file to modify data for the default saddle dimensions source (Moss) or create a new source. The saddle dimension sources display when you click **Select Saddle.** Once a source is selected, the dimension values automatically populate on the **Saddle Dialog** based upon the vessel's diameter. This saves time and reduces chances of error.

Common Detail Parameters (page 54)

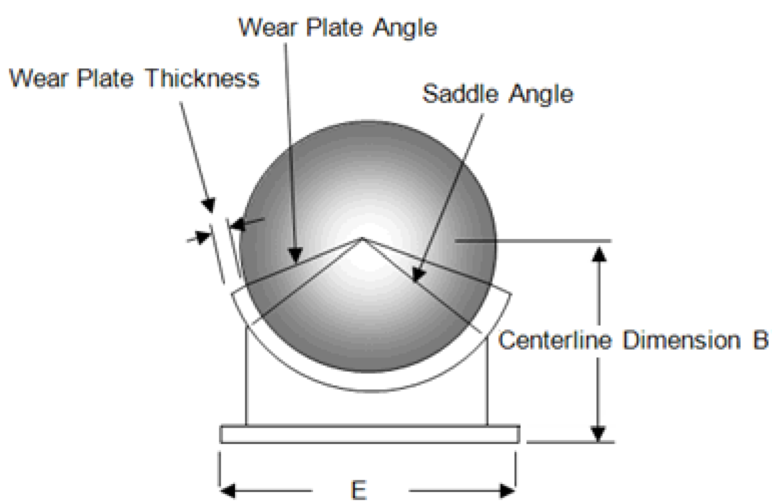
Saddle Width

Enter the width of the saddle support. This value does not include any wear pad on the vessel side, and is used primarily for the Zick analysis of horizontal vessels on saddle supports.



Centerline Dimension B

Enter the distance from the base of the saddle to the centerline of the vessel, referred to as dimension *B* in some pressure vessel texts. This value is used to determine additional saddle loads due to wind or seismic events.



Saddle Contact Angle

Enter the angle between the two contact points ("horns") of the saddle, measured from the axial center of the vessel. This value typically ranges from 120.0º to 150.0º.

Wear Plate Width

Enter the width of the wear plate between the vessel and the saddle support. This value is used primarily for the Zick analysis of horizontal vessels on saddle supports.

Wear Plate Thickness

Enter the thickness of the wear plate between the vessels and the saddle support. This value is used primarily for the Zick analysis of horizontal vessels on saddle supports.

Wear Plate Contact Angle

Enter the angle contained from one edge of the wear plate to the other edge, measured from the axial center of the vessel. Typically this value is approximately 130º.

Height of Section Ring

If a custom fabricated composite (usually T type) stiffener is used over the saddle supports, enter the height from the shell surface to the top of the stiffener. This value will be used to compute the stress at the tip of the stiffener. If a horizontal vessel does not have stiffeners over the saddle supports, enter **0**. If you selected a stiffener from the AISC structural steel database in the **Stiffening Ring** dialog box, enter **0**. In this case, the software gets the ring height from the AISC database.

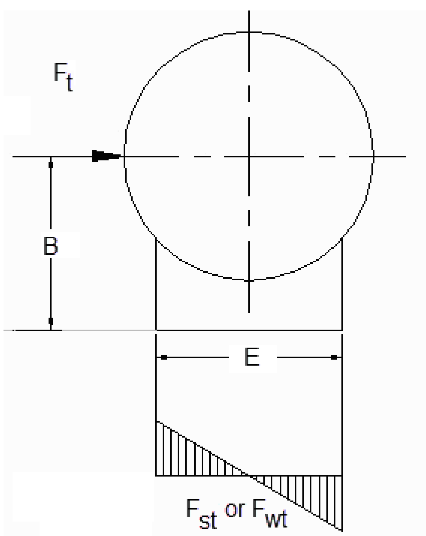
Friction Coefficient Mu

Enter the coefficient of friction m_u between the base of the saddle and the supporting foundation, piers or structure. A frictionless surface has a m_u value of **0**. Other typical values are in the range of **0.3** to **0.5**. The software uses this value to determine the counteracting force caused by thermal expansion and the dead weight of the vessel on the saddle support. This is essentially a resistive force bending the saddle. The generated force is proportional to m_u times the normal force.

Moment Factor, Ftr

Enter the moment factor for calculating the saddle reaction force due to the wind or earthquake transverse load. The recommended value is **3**.

The value of **6** is conservative in that it assumes that the maximum edge load is uniform across the entire base, when realistically it occurs only at the edge. A more accurate method converts this triangular loading into a more realistic uniform load, leading to the value of 3. The following figure shows an end view of a horizontal vessel with a transverse load, simulating wind/seismic loading:

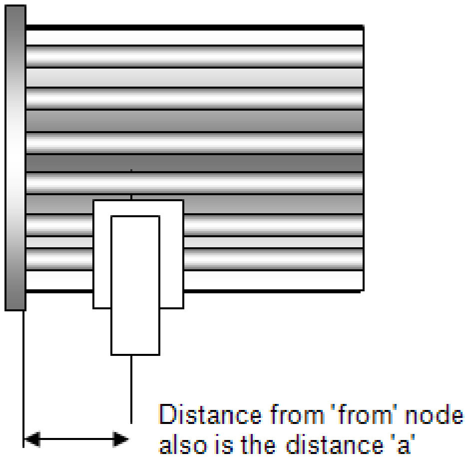


The saddle reaction load Fst (or Fwt for wind) due to the transverse load Ft is:

```
Fst (or Fwt) = (Saddle Moment Factor) * Ft * B / E
```

Saddle Dimension a

Enter the distance between the centerline of the saddle support and the tangent line of the nearest head. This dimension is labeled *A* in most pressure vessel texts.



Dimension E at base

Enter the dimension of the baseplate that is less than the distance in contact with the supporting surface. Dimension E addresses the saddle reaction force due to wind or seismic force when the baseplate distance dimension has a different distance in contact with the supporting surface. This entry is optional.

Tangent to Tangent Distance

Enter the length of the cylindrical shell measured from tangent line to tangent line for a vessel with curved heads or from inner face to inner face for vessels with flat covers or tubesheets. For most horizontal vessels comprised of dished ends and cylinders that are not heat exchangers, PV Elite can determine this value and you can leave the value set to zero. For vessels that have cones, tubesheets, and so forth, you must manually enter this value.

Circ Eff. over Saddle

Enter the circumferential efficiency in the plane of the saddle.

When you create a shell by welding the ends together, there is a longitudinal weld. If that weld is at the saddle, there are bending moment stress.

Circ Eff. over Midspan

Enter circumferential efficiency at the mid-span.

When you create a shell by welding the ends together, there is a longitudinal weld. If that weld is between saddles, there are bending moment stress.

Wear Plate and Shell Matls are the same?

Select if the wear plate and shell materials are the same.

Perform Saddle Check?

Select to add rib, web, and baseplate data, and perform a structural analysis of the saddle.

Saddle Allowable Stress

Enter the saddle allowable stress. Alternatively, click **Matl...** to select a material directly from the **Material Database Dialog Box** (page 547) dialog box.

Material Yield Stress

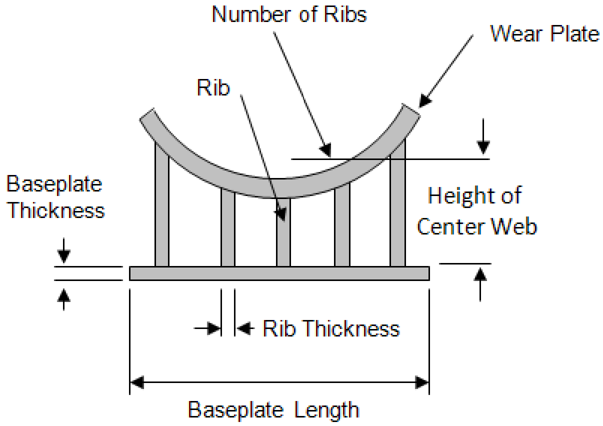
Enter the yield stress for the saddles at their design temperature. Alternatively, click **Matl...** to select a material directly from the **Material Database Dialog Box** (page 547) dialog box.

E for Plates

Enter the modulus of elasticity *E* for the saddle material.

Baseplate Length

Enter the long dimension of the baseplate in the direction of the vessel diameter.

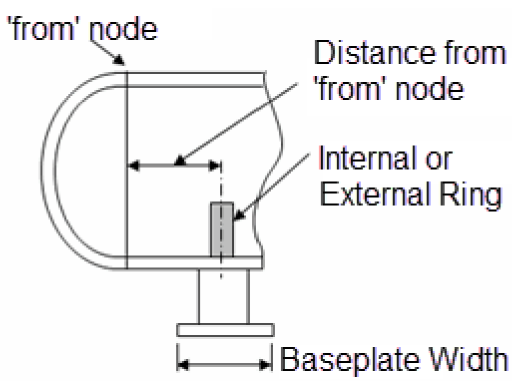


Baseplate Thickness

Enter the thickness of the baseplate.

Baseplate Width

Enter the short dimension (the width) of the baseplate.



Number of Ribs

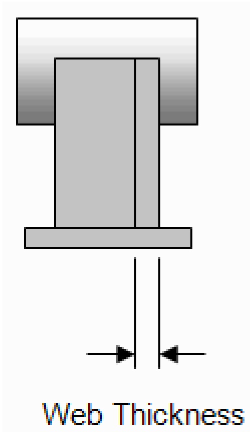
Enter the number of ribs on one saddle, running parallel to the long axis of the vessel.

Rib Thickness

Enter the thickness of the rib supports.

Web Thickness

Enter the thickness of the web. The web is the vertical plate between the baseplate and the wear plate, to which the ribs are attached.

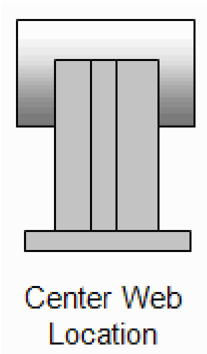


Height of Center Web

Enter the height of the center web as it extends from the baseplate to the shell inside diameter (ID). Think of this value as the column height of all material above the baseplate to the inside of the pressure vessel, including the Wear Plate Thickness and the Shell Thickness. The software uses this value to determine if the centerline of the saddle may fail if the vessel fails at that specific location.

Web Location

Select the web location relative to the saddle baseplate and wear plate. Select **Center**, **Side - Left**, or **Side - Right**.



Perform Anchor Bolts Calculations?

Select to add anchor bolt data, and perform anchor bolt and baseplate calculations. Enter values for the options below.

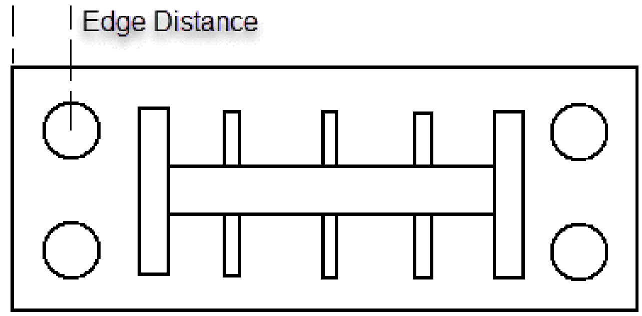
NOTE PV Elite performs anchor bolt calculations for horizontal vessel saddle supports and skirt and leg baseplate supports. This analysis determines the uplift on the saddle supports due to the external loads on the vessel. Loadings include defined forces and moments and wind and seismic loads. If the vessel is in uplift, the required area of the bolts is determined. More information on this analysis is found in the Pressure Vessel Design Manual by Dennis R. Moss, 1997 and later editions. In addition to calculation of the required bolt area, this method also calculates the required thickness of the baseplate due to the applied bolt load. In some cases the bolt load controls the thickness of the saddle baseplate.

Saddle Bolted to Steel Foundation?

Select if the saddles are bolted to a steel substructure.

Number of Bolts

Enter the total number of bolts to be used on the baseplate. The bolts are assumed to be at the edge of the baseplate along the short side.



Num of Bolts in Tension

Enter the number of bolts in tension, generally the total number of bolts divided by two.

Edge Distance

Enter the distance from the edge of the baseplate to the centerline of the bolts.

Bolt Corrosion Allowance

Enter the bolt corrosion allowance (BCA). If the bolt corrosion allowance specified is a "total", then divide it by two; otherwise this calculation will be overly conservative. When dealing with bolt corrosion, the following equation is used to calculate the equivalent bolt OD and corresponding reduced area of the bolt:

$$BOD = (BLTAREA * 4.0 / \Pi)^{1/2} - 2.0 * BCA$$

Bolt Material

Enter the bolt material. Alternatively, click **Matl...** to select a material directly from the **Material Database Dialog Box** (page 547) dialog box.

Bolt Allowable Stress

Enter the allowable stress of the bolt. If you select a material directly from the **Material Database** dialog box for **Bolt Material**, the software provides the allowable stress at ambient temperature. If you need the allowable at an elevated temperature (such as for an insulated vessel), then you must enter the allowable stress at that temperature.

Thread Series

Enter the thread series identifier:

- **TEMA -** 8 thread series, adapted from the TEMA Standard
- **TEMA Metric**
- **UNC** - Unified National Course Threads, adapted from Mark's Handbook
- **BS 3643**
- **SABS 1700**

- **User Defined Root Area** - Also enter a value for the root area of a single bolt in **Bolt Root Area**. This information can be obtained from a standard engineering handbook.

Nominal Bolt Diameter

Select the nominal bolt diameter. In general this value ranges from 1/2 inch to 4 inches.

Bolt Root Area

For nonstandard or metric bolts, enter the root cross-sectional area of the bolt. If you have entered a value for **Bolt Corrosion Allowance**, the software modifies the area of the bolt using the equation described above.

Optional Moments for Saddle Analysis

According to definitions in ASME VIII-2 4.15.6 enter values for the following optional moments:

Moment M1 - Net-section maximum, maximum longitudinal bending moment at the saddle support. The moment is negative when it results in a tensile stress at the top of the shell.

Moment M2 - Net-section maximum, maximum longitudinal bending moment between the saddle support. The moment is negative when it results in a tensile stress at the top of the shell.

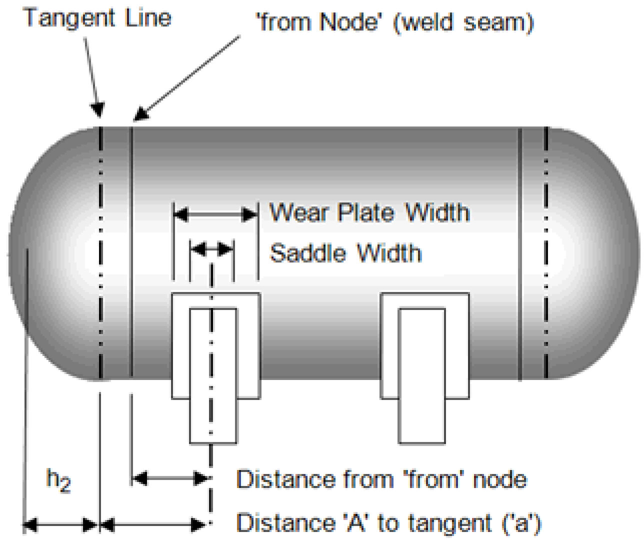
The saddle analysis used in PV Elite can be either the method outlined in ASME VIII-2 paragraph 4.15.3 or PD 5500 Annex G. Both of these analyses are based on the original method outlined in the September 1951 Paper by L.P. Zick, "Stresses in Large Horizontal Cylindrical Pressure Vessels on Two Saddle Supports." This paper first appeared in the T*he Welding Journal Research Supplement*. The basis of the analysis is a typical pressure vessel with ellipsoidal, torispherical or hemispherical heads at either end, and cylindrical sections in the middle. The two saddle supports are assumed to be equidistant from the ends. With these assumptions, the bending moments between the saddles and over the saddles can be calculated.

Unfortunately, the geometry of many vessels and exchangers do not fall in line with these assumptions. As a result, the analysis of these vessels may produce moments that are not correct with regard to the original Zick formulas. The software allows the optional input of **Moment M1** and **Moment M2**, as defined in ASME VIII-2 4.15.3, for both the operating and hydrotest conditions. If **Moment M1** and **Moment M2 are zero**, moments are calculated based on the design information provided in the **Saddle** dialog. If **Moment M1** and **Moment M2** are non-zero, these moment values are used, overriding the calculated moments. **Moment M1** and **Moment M2** can be positive or negative.

According to ASME VIII-2, paragraph 4.15.3.2, the moments are calculated for equidistant saddles with $a \leq 0.25L$. If the relationship between a and L is not maintained, the moments should be calculated using an appropriate engineering beam type analysis method (such as shear and moment diagrams). From ASME VIII-2 4.15.6:

- a - Distance from the axis of the saddle support to the tangent line on the curve for a dished head or to the inner face of a flat cover or tubesheet.

- *L* - Length of the cylindrical shell measured from the tangent line for a vessel with dished ends or from the inner face to inner face for vessels with flat covers or tubesheets.



NOTE These overriding moments are only used in Division 2 saddle analysis and PD 5500 saddle analysis.

Tray

Home tab: **Details** > **Tray**

Adds a set of equally spaced trays with a set liquid height to the selected element on a vertical vessel.

Previous Tray Group

If you created more than one tray set on the element, click to go back to the previous set.

Go To Next Tray Set

If you created more than one tray set on the element, click to go to the next set.

Add New Tray Set

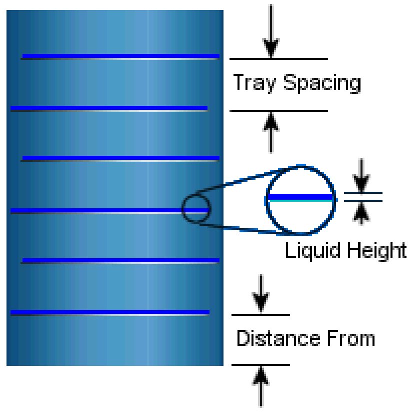
Click to add a new tray set to the element.

Delete

Deletes all data for the current tray set.

Set Default Tray Weight

Sets the value in **Tray Weight Per Unit Area** as the default weight for new trays added to the model.



Common Detail Parameters (page 54)

Number of Trays

Enter the number of trays on the element.

Tray Spacing

Enter the vertical distance between trays.

Tray Weight Per Unit Area

Enter the unit weight of each tray in the set. Do not enter the total weight, because the software multiplies the unit weight by the cross-sectional area of the element.

Support Ring and Bolting Bar Weight

Enter the support ring and bolting bar weight.

Height of Liquid on Tray

Enter the height of the liquid on each tray.

Density of Liquid on Tray

Enter the density of the liquid on each tray. For more information, see **Liquid** (page 124).

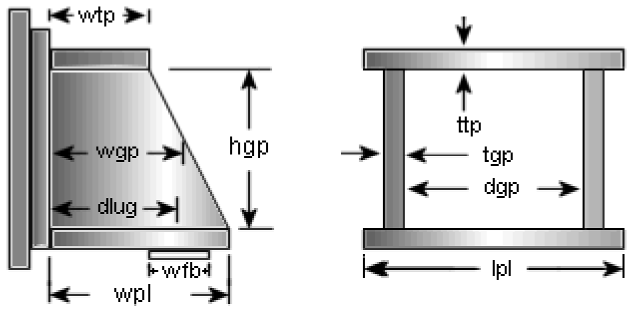
Lug

Home tab: **Details** > **Lug**

Adds support lugs to the selected element on a vertical vessel. If no skirt or legs are defined for a vertical vessel, the lowest set of lugs are used as the vessel support point for dead load and live load calculations.

Delete

Deletes all data for the lug.

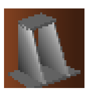


Common Detail Parameters (page 54)

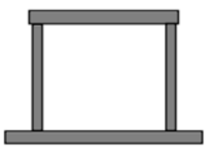
Lug Type

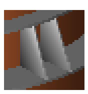
Select the type of geometry for the support lug:

Simple geometry with gussets

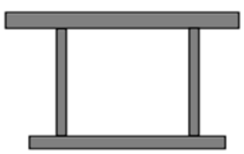


Gusseted geometry with top plate





Gusseted geometry with continuous top encirclement ring



Lug Start Angle

Enter the angle between the designated zero degree line of the vessel and the start angle of the lug.

Lug Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Number of Lugs

Enter the number of support lugs around the periphery of the vessel at this location.

Dist. from OD to Lug MidPt (dlug)

Enter the radial distance from the wall of the vessel to the midpoint where the lug attaches to the structural steel.

Weight of One Lug

Enter the actual weight of one support lug. The software does not gather enough information to do the detailed calculation of the support lug weight.

Force Bearing Width (wfb)

Enter the width of the structure that is in contact with the bottom lug support plate.

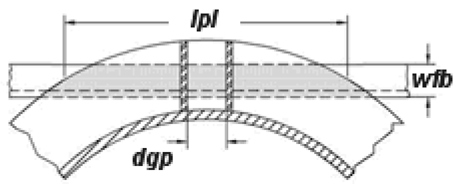
Radial Width of Bottom Ring (wpl)

Enter the distance that the bottom support plate extends from the OD of the vessel. This value must be greater than or equal to **Mean Width of Gussets (wgp)**.

Effective Force Bearing Length (lpl)

For lug types with a bottom plate and no continuous rings (Simple geometry with no gussets and gusseted geometry with top plate in **Lug Type**), enter the distance between gussets plus two times the gusset plate thickness.

For lug types with continuous top and bottom rings (Gusseted geometry with continuous top encirclement ring in **Lug Type**), enter the length of the bottom plate located on a support:



Thickness of Bottom Ring (tpl)

Enter the thickness of the bottom support plate.

Distance Between Gussets (dgp)

Enter the distance between the inside surfaces of the gusset plates.

Mean Width of Gussets (wgp)

Enter the mean gusset width, defined as the gusset width at the top plus the gusset width at the bottom divided by two. The software uses this value to calculate the actual stresses in the gusset plates.

Height of Gussets (tgp)

Enter the height of one gusset.

Thickness of Gussets (tgp)

Enter the thickness of the gusset plate.

Radial Width of Top Ring (wtp)

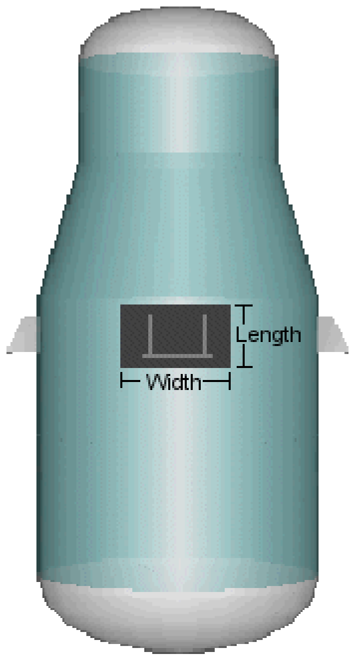
Enter the radial dimension from the OD of the shell to the edge of the plate. This value should be less than or equal to **Mean Width of Gussets (wgp)**.

Thickness of Top Ring (ttp)

Enter the thickness of the top support plate ring that sits above the gussets.

Perform WRC 107/537 Calculation

Select to perform the WRC 107 local stress analysis on a reinforcing pad for the lug. The software calculates the stresses at the edge of the attachment and the edge of the pad. Enter values for the width, length, and thickness options.



Bolt Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Type of Threads

Enter the thread series identifier:

- **TEMA -** 8 thread series, adapted from the TEMA Standard
- **TEMA Metric**
- **UNC** - Unified National Course Threads, adapted from Mark's Handbook
- **BS 3643**
- **SABS 1700**
- **User Root Area** - Also enter a value for the root area of a single bolt in **Bolt Root Area**. This information can be obtained from a standard engineering handbook.

Nominal Bolt Diameter

Select the nominal bolt diameter. In general this value ranges from 1/2 inch to 4 inches.

Bolt Root Area

When **User Root Area** is selected for **Type of Threads**, enter the root cross-sectional area of the bolt.

Legs

Home tab: **Details** > **Legs**

Adds support legs to the selected element. Legs can be created for a vertical vessel without a skirt element.

Leg Input Parameters Tab (Leg Dialog Box) (page 116)

Base Plate Parameters Tab (Leg Dialog Box) (page 121)

Leg Input Parameters Tab (Leg Dialog Box)

Defines parameters for the legs and leg pads.

Common Detail Parameters (page 54)

Delete

Deletes all data for the legs.

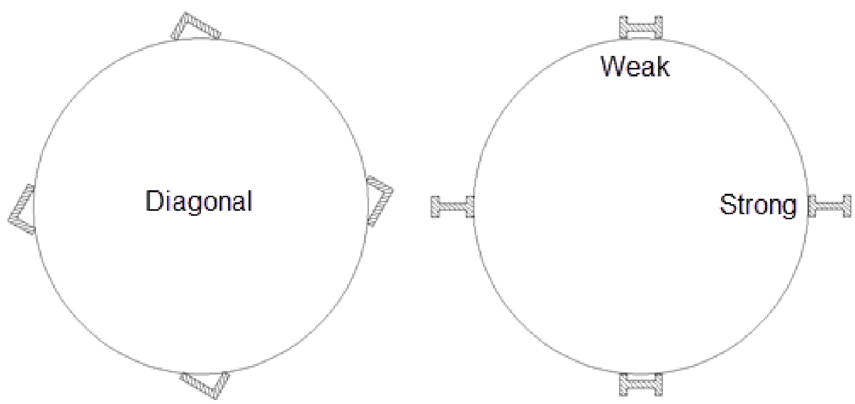
Leg Centerline Diameter

Enter the distance between the centerlines of two legs on opposing sides of the vessel. If there are an odd number of legs (therefore, no two legs are opposing), then enter the diameter of a circle drawn through the centerlines of the legs. Alternatively, click **Compute Centerline Diameter** to calculate the value.

Leg Orientation

Select the orientation of the leg cross-section with respect to the centerline. Select:

- **Strong Axis** - The strong axis is perpendicular to the vessel.
- **Weak Axis** - The weak axis is perpendicular to the vessel.
- **Diagonal** - The strong axis is diagonal to the vessel.



Number of Legs

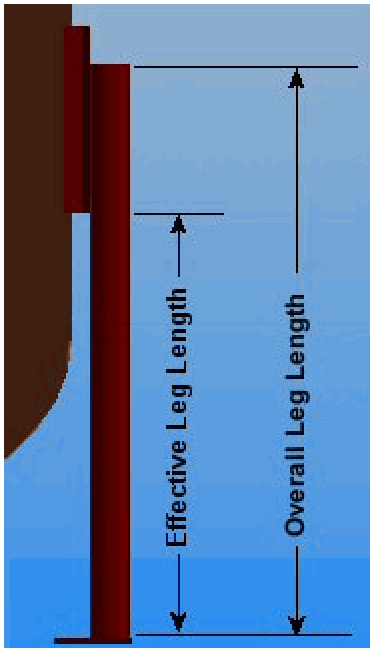
Enter the number of legs. You must create at least three legs.

Overall Length of Legs

Enter the distance from the ground to the attachment point of the leg on the vessel.

Effective Leg Length "L"

Enter the effective leg length *L*, defined as the free length of the leg that is subject to bending. This is the value that is used in the AISC formula kl/r. This value cannot be explicitly calculated because there are numerous configurations of legs. If the value is zero, the software conservatively uses the overall leg length as the effective leg length in the calculation.

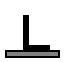


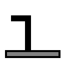
Leg Database

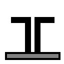
Select the structural specification database to use for leg cross-sections.

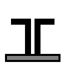
Section Identifier

Enter a section name in the format of the specification selected in **Leg Database**, or click **LookUp** to open the **Select a Leg Shape** dialog box and select a section from the database.

 L - Equal angle

 L - Unequal angle

 B/D - Double angles with large or small sides back to back

 C/HP/M/MC/S - Channels and other miscellaneous channels

W - Wide Flanges

ST - Structural Tees

Leg Yield Stress

Enter the yield stress for the legs. Alternatively, click ... to select a material directly from the **Yield Stress Selection** dialog box.

Effective End Condition "K"

Enter the effective end condition *K*. For pressure vessel legs, a value of **1.0** or **1.5** is commonly used.

End Condition	Theoretical K	Recommended K
Fixed - Fixed	0.5	**0.65**
Fixed - Pinned	0.7	**0.80**
Fixed - Trans	1.0	**1.20**
Pinned - Pinned	1.0	**1.00**
Fixed - Rotates	2.0	**2.10**
Pinned - Rotates	2.0	**2.00**

Leg Start Angle

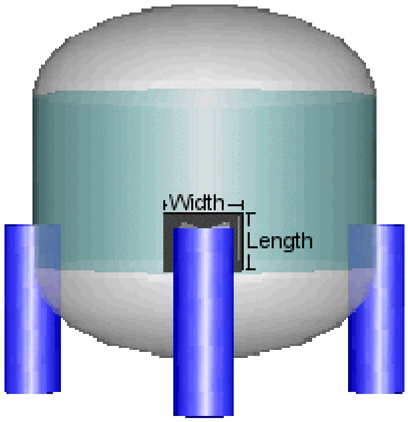
Enter the angle between the designated zero degree line of the vessel and the start angle of the first leg.

Occasional Load Factor

Enter a factor for irregularly occurring loads. The default value is **1.333**.

Perform WRC 107/537 Analysis

Select to perform the WRC 107 local stress analysis on a reinforcing pad between a leg and the vessel. The software calculates the stresses at the edge of the attachment and the edge of the pad.



Pad Width

Enter the pad width as measured along the circumferential direction of the vessel. The pad width must be greater than attachment width.

Pad Length

Enter the length of the pad as measured along the long axis of the vessel.

Pad Thickness

Enter the thickness of the pad.

Compute Centerline Diameter

Click to calculate the value for **Leg Centerline Diameter** based on the element OD, **Number of Legs**, the cross-section selected for **Section Identifier**, and the **Pad Thickness** for **Perform WRC 107 Analysis**.

Are the Legs Cross Braced?

Select if the legs are diagonally braced. Bracing the legs reduces bending and increases the axial load in the legs.

Are these Pipe Legs?

Select to analyze pipe legs. Also enter values for **Pipe Leg Inside Diameter** and **Pipe Leg Outside Diameter**.

Pipe Leg Inside Diameter

Enter the corroded pipe inside diameter. Alternatively, click ... to open the **Seamless Pipe Selection** dialog box, and select a pipe schedule and nominal diameter.

Pipe Leg Outside Diameter

Enter the corroded pipe outside diameter. Alternatively, click ... to open the **Seamless Pipe Selection** dialog box, and select a pipe schedule and nominal diameter.

NOTE This value must be greater than **Pipe Leg Inside Diameter**. The software uses these values to calculate moment of inertia, section modulus and radius of gyration for the legs. These values are used in the AISC unity check and natural frequency calculations.

Vessel Translates during Occasional Load?

Select to calculate a more conservative longitudinal moment when **Perform WRC 107 Analysis** is also selected.

Employ Directional Check for W and C Types

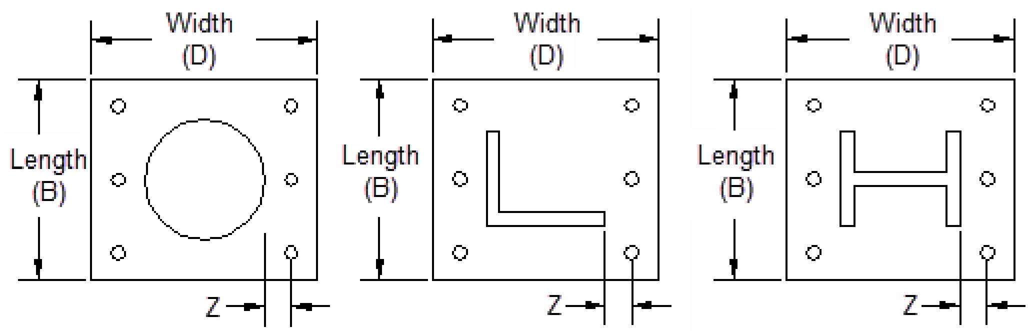
Select to indicate you want the software to perform an AISC unity check for the leg angles with W and C types only. When selected, the software calculates additional results for AISC H1-1 and H1-2 on the Leg Check report.

Base Plate Parameters Tab (Leg Dialog Box)

Defines parameters for the leg base plates.

Perform Baseplate Analysis?

Select to place a baseplate on the bottom of each leg and analyze baseplate, bolt, and foundation loads and enter values for the options below. The software assumes the leg is attached symmetrically on the baseplate.



Length, B

Enter the length along the bolt side.

Width, D

Enter the width.

Thickness

Enter the thickness of the baseplate.

Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Thread Series

Enter the thread series identifier:

- **TEMA -** 8 thread series, adapted from the TEMA Standard
- **TEMA Metric**
- **UNC** - Unified National Course Threads, adapted from Mark's Handbook
- **BS 3643**
- **SABS 1700**
- **User Root Area** - Also enter a value for the root area of a single bolt in **Root Area**. This information can be obtained from a standard engineering handbook.

Nominal Diameter

Click ... and select the nominal bolt diameter from the bolt table selected for **Thread Series**. If you have a bolt that is outside of the bolt table ranges, enter the nominal size and select **User Root Area** in **Thread Series**.

For **TEMA** and **UNC** bolts, the available bolt diameters are:

Bolt Size (inches)	**Bolt Root Area**	
	(cm.^2)	**(in.^2)**
0.500	0.8129	0.126
0.625	1.3032	0.202
0.750	1.9484	0.302
0.875	2.7032	0.419
1.000	3.5548	0.551
1.125	4.6968	0.728
1.250	5.9935	0.929
1.375	7.4516	1.155
1.500	9.0645	1.405
1.625	10.8387	1.680
1.750	12.7741	1.980
1.875	14.8645	2.304
2.000	17.1096	2.652
2.250	22.0838	3.423
2.500	27.6903	4.292
2.750	33.9290	5.259
3.000	40.7999	6.324
3.250	48.3031	7.487
3.500	56.4450	8.749
3.750	65.2130	10.108

Bolt Size (inches)	Bolt Root Area	
4.000	74.6192	11.566

NOTE This information is adapted from Jawad and Farr*, Structural Analysis and Design of Process Equipment*, pg 425.

Corrosion Allowance

Enter the bolt corrosion allowance. The software uses this value to corrode the radius of the root area and calculate a corroded root stress area based on the nominal bolt size and bolt table. This area is then used in the remainder of the bolt load/stress calculations.

The software calculates the required area of the bolt. If the bolt corrosion allowance is greater than zero, the software adds the corrosion allowance and recalculates the diameter based on the new required area:

Corroded Bolt Root Diameter = (4 * New Bolt Area/Pi)$^{1/2}$ - 2 * Bolt Corrosion Allowance

Number per Baseplate

Enter the number of bolts per baseplate. In most cases, this should be an even number, but for angle legs, the value can be **1.**The software assumes that the bolts are located along **Length, B**. The required size of the bolt and the baseplate thickness are looked up from a table in the Pressure Vessel Design Manual and Pressure Vessel Handbook. There are no calculations for one bolt.

Number in Tension

Enter the number of bolts in tension under wind, earthquake, and horizontal loads, defined as the number of bolts along one **Length, B** side (three bolts in the examples above). If these load cases are not needed, no value is required.

Distance from Edge to Bolt

Enter the dimension *z* from the edge of the leg to the centerline of the bolts, measures along the **Width, D** side.

Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Root Area

When **User Root Area** is selected for **Thread Series**, enter the root cross-sectional area of the bolt.

Nominal Compressive Strength

Enter the following for the concrete to which the base is bolted:

- **F'$_c$ -** The nominal ultimate compressive stress of the concrete. This value is F'$_c$ in Jawad and Farr or FPC in Meygesy. A typical entry is 3000 psi.

- **F**$_c$ - The allowable compressive stress of the concrete
- **n** - The steel-to-concrete modulus of elasticity ratio, E_{plates}/E_c.

Average Values of Properties of Concrete Mixes (adapted from Brownell and Young)

Water Content (US Gallons per 94 lb Sack of Cement)	**f'$_c$ 28-day Ultimate Compressive Strength (psi)**	**f$_c$ Allowable Compressive Strength = 0.45*f'c (psi)**	**n Modular Ratio (E_s/E_c)**
7.5	2000	800	15
6.75	2500	1000	12
6	3000	1200	10
5	3750	1400	8

NOTE According to Jawad and Farr, Ec is equal to 57000 multiplied by the square root of f'$_c$ psi. The modulus of elasticity of steel is assumed to be 30 x 10^6.

Liquid

Home tab: **Details** > **Liquid**

Adds liquid data to the element. Normally, **Liquid** is used on the bottom head of the vessel. Each element is then filled with the appropriate amount of liquid. We recommend that you model the entire vessel before adding liquid data.

Delete

Deletes all data for the liquid.

Full

Click to calculate the values of **Distance from "From" Node** and the value of **Height in this Element**, assuming that the element is completely filled with the defined liquid.

Fill Elements Now

Click to fill the vessel to the level specified by **Height from Datum**. If the height is greater than the vessel height, the vessel is completely filled. The software assigns detail ID values using the **Liquid** (page 124). To use this command effectively, all elements in the vessel model should be created first.

Common Detail Parameters (page 54)

Liquid Density

Enter the density or specific gravity of the liquid. Typical specific gravities and densities are shown below. The densities should be converted if you use another units system.

Name	**Specific Gravity**	**Density (lb/ft3)**
Ethane	0.3564	22.23

Propane	0.5077	31.66
N-butane	0.5844	36.44
Iso-butane	0.5631	35.11
N-Pentane	0.6310	39.35
Iso-Pentane	0.6247	38.96
N-hexane	0.6640	41.41
2-methypentane	0.6579	41.03
3-methylpentane	0.6689	41.71
2,2-dimethylbutane	0.6540	40.78
2,3-dimethylbutane	0.6664	41.56
N-heptane	0.6882	42.92
2-methylheptane	0.6830	42.59
3-methylheptane	0.6917	43.13
2,2-dimethylpentane	0.6782	42.29
2,4-dimethylpentane	0.6773	42.24
1,1-dimethylcyclopentane	0.7592	47.34
N-octane	0.7068	44.08
Cyclopentane	0.7504	46.79
Methylcyclopentane	0.7536	46.99
Cyclohexane	0.7834	48.85
Methylcyclohexane	0.7740	48.27
Benzene	0.8844	55.15
Toluene	0.8718	54.37
Alcohol	0.7900	49.26
Ammonia	0.8900	55.50

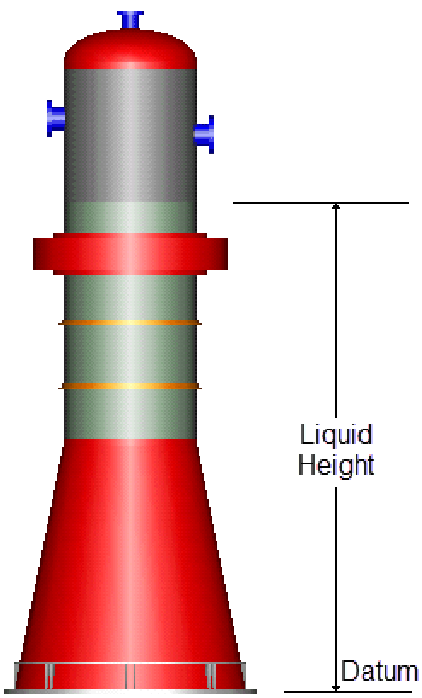
Benzine	0.6900	43.03
Gasoline	0.7000	43.65
Kerosene	0.8000	49.89
Mineral Oil	0.9200	57.37
Petroleum Oil	0.8200	51.14
Water	1.0	62.4

Height in this Element

Enter the height or length of the liquid on this element. This value is used only to calculate the weight of the liquid section. For seismic calculations, the weight center of the liquid section is taken at half this height. The value is also used to calculate the operating pressure at all points below the liquid.

Height from Datum

Enter the height or length of the liquid from the datum line to the operating or design level. Normally, **Liquid**  is used on the bottom head of the vessel. Each element is then filled with the appropriate amount of liquid. It is best to model the entire vessel before adding liquid data. If a value is entered that is greater than the height of the vessel, the software completely fills the vessel.



Insulation

Home tab: **Details** > **Insulation**

Adds insulation to the element.

Delete

Deletes all insulation data.

Full

Click to calculate the values of Distance from "From" Node and the value of Height/Length of Insulation, assuming that the element is fully covered by the defined insulation.

All

Click to cover the entire vessel with insulation. The software assigns detail ID values using the defined insulation values. To use this command effectively, all elements in the vessel model should be created first.

Common Detail Parameters (page 54)

Height/Length of Insulation

Enter the height or length of the insulation on the element. This value is used only to calculate the weight of the insulation. For seismic calculations, the weight center of the insulated section is taken at half this height. If you have insulation on a horizontal vessel, the value is the length of the insulated section.

NOTE The only software distinction between insulation and lining is that insulation is on the OD of the element, while lining is on the ID of the element. Use **Insulation** to add OD fireproofing, and **Lining** to add ID fireproofing.

Thickness of Insulation

Enter the thickness of the insulation or fireproofing.

Density of Insulation

Enter the density of the insulation, such as the following typical densities:

Material Type	**Density (lbs/ft^3)**
Calcium Silicate	22.5
Foam Glass	16.0
Mineral Wool	14.0
Glass Fiber	11.0
Asbestos	30.0
Careytemp	18.0
Kaylo 10	22.0

Perlite / Celo-temp 1500	23.0
Polyurethane	4.0
Styrofoam	3.0

Type of Insulation

Enter a description for the type of insulation.

Lining

Home tab: **Details** > **Lining**

Adds lining to the element.

Delete

Deletes all lining data.

Full

Click to calculate the values of **Distance from "From" Node** and the value of **Height/Length of Lining**, assuming that the element is fully covered by the defined lining.

Common Detail Parameters (page 54)

Height/Length of Lining

Enter the height or length of the lining on this element. This value is used only to calculate the weight of the lined section. For seismic calculations the weight center of the lined section will be taken at half this height. If you have a lining in a horizontal vessel, the value is the length of the lined section.

NOTE The only software distinction between insulation and lining is that insulation is on the OD of the element, while lining is on the ID of the element. Use **Insulation** to add OD fireproofing, and **Lining** to add ID fireproofing.

Thickness of Lining

Enter the thickness of the lining or fireproofing.

Density of Lining

Enter the density of the insulation, lining, or packing, such as the following typical lining densities:

Material Type	**Density (lbs/ft^3)**
Alumina Brick	170.0
Fire Clay	130.0
High Alumina	130.0
Kaolin	135.0

Magnesite	180.0
Silica	110.0
Insulating Fire Brick	40.0
Concrete	140.0
Cement	100.0

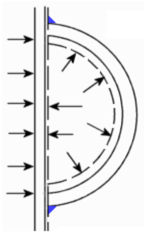
Also see **Insulation** (page 127).

Halfpipe Jacket

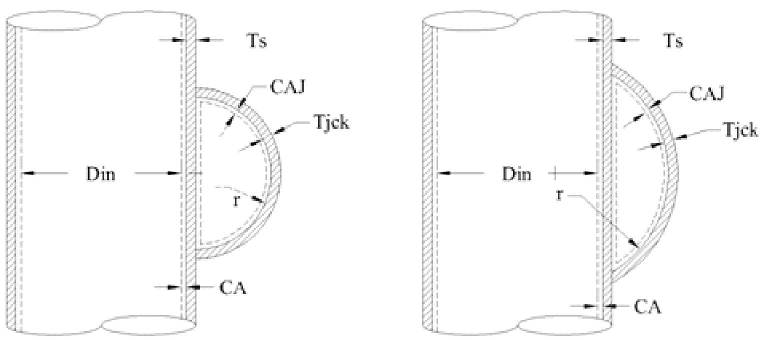
Home tab: **Details** > **Halfpipe Jacket**

Adds half-pipe jackets to the shell on the selected cylinder element, according to ASME Section VIII, Division 1, Appendix EE or ASME Section VIII, Division 2, Part 4.11.6.

Half pipe jackets are commonly made by rolling and forming flat bar in a specific radius and helical curvature that matches that of the parent shell course. These jackets are used to heat or cool the contents inside of the vessel. For vessels that are under cyclic service, it is advised that the jacket be attached by both a fillet and full penetration groove weld.



PV Elite performs required thickness and maximum allowable working pressure (Mawp) calculations for cylindrical shells with half-pipe jackets attached. The analysis is based on ASME Section VIII, Division 1, Paragraph EE-1, Appendix EE or ASME Section VIII, Division 1, Part 4.11.6. The analysis is only valid for the cylindrical geometries shown in Figure EE-4.



Additionally, only nominal pipe sizes from 2 to 4 can be used. Although there are no charts for sizes 2.5 and 3.5, the software accepts these sizes and performs iterations between the given charts. If the half-pipe is a nonstandard pipe size, or has a formed radius, the actual radius is used in the calculations.

The software takes full account of corrosion allowance. Actual thickness values and corrosion allowances are entered, and the software adjusts thicknesses and diameters when making calculations for the corroded condition.

Delete

Deletes all jacket data.

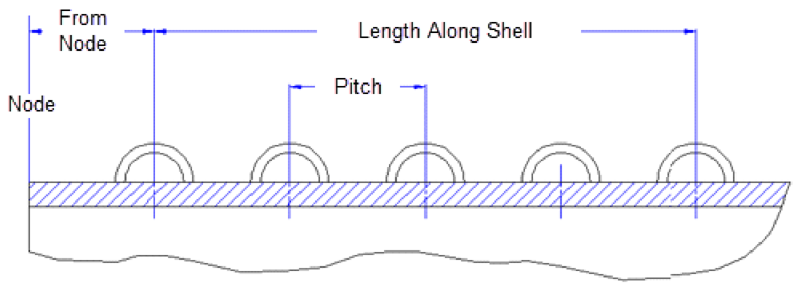
Quick Results

Click to see a quick report of half pipe jacket analysis results.

Common Detail Parameters (page 54)

Length Along Shell of Jacket Section

Enter the distance that the jacket extends along the length of the shell section. This value cannot be greater than the specified length of the shell course.



Pitch Spacing

Enter the distance between centers of adjacent half-pipes.

Shell Corrosion Allowance in Jacket

Enter the internal corrosion allowance of the half-pipe jacket.

Start Angle

Enter the start angle.

Jacket Design Temperature

Enter the design temperature of the jacket.

Jacket Design Pressure

Enter the design pressure of the fluid inside the jacket.

Jacket Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Pipe

Click to open the **Seamless Pipe Selection** dialog box and select values from the piping database for **Jacket Corrosion Allowance** and **Minimum Jacket Thickness**.

Jacket Corrosion Allowance

Displays the corrosion allowance for the pipe that you selected in **Pipe**. You can also manually enter a value.

Minimum Jacket Thickness

Displays the thickness for the pipe that you selected in **Pipe**. You can also manually enter a value.

Inside Radius of Formed Jacket

Enter a value for a jacket formed from bar or plate. This value is only used if **Formed Radius Type** is selected for **Nominal Pipe Size**.

Nominal Pipe Size

Select the nominal pipe size of the jacket. Select **2 inch**, **3 inch**, or **4 inch** to use sizes recognized by Appendix EE. Select **2.5 inch** or **3.5 inch** to interpolate from Appendix EE graphs. Select **Formed Radius Type** for a non-standard jacket, and also enter a value for **Inside Radius of Formed Jacket**.

Contents Specific Gravity

Enter the specific gravity of any fluid contained within the jacket.

Enter the density or specific gravity of the liquid. Typical specific gravities and densities are shown below. The densities should be converted if you use another units system.

Name	Specific Gravity	Density (lb/ft3)
Ethane	0.3564	22.23
Propane	0.5077	31.66
N-butane	0.5844	36.44
Iso-butane	0.5631	35.11
N-Pentane	0.6310	39.35
Iso-Pentane	0.6247	38.96
N-hexane	0.6640	41.41
2-methypentane	0.6579	41.03
3-methylpentane	0.6689	41.71
2,2-dimethylbutane	0.6540	40.78
2,3-dimethylbutane	0.6664	41.56
N-heptane	0.6882	42.92
2-methylheptane	0.6830	42.59

3-methylheptane	0.6917	43.13
2,2-dimethylpentane	0.6782	42.29
2,4-dimethylpentane	0.6773	42.24
1,1-dimethylcyclopentane	0.7592	47.34
N-octane	0.7068	44.08
Cyclopentane	0.7504	46.79
Methylcyclopentane	0.7536	46.99
Cyclohexane	0.7834	48.85
Methylcyclohexane	0.7740	48.27
Benzene	0.8844	55.15
Toluene	0.8718	54.37
Alcohol	0.7900	49.26
Ammonia	0.8900	55.50
Benzine	0.6900	43.03
Gasoline	0.7000	43.65
Kerosene	0.8000	49.89
Mineral Oil	0.9200	57.37
Petroleum Oil	0.8200	51.14
Water	1.0	62.4

Tubesheet

Home tab: **Details** > **Tubesheet**

Adds a heat exchange tubesheet to the element. To build and analyze shell and tube heat exchangers with PV Elite, you must understand the modeling techniques involved:

- First, define and design the pressure envelope or exterior components of the exchanger first, including all covers, flanges and cylinders. Verify that the design pressures and temperatures are correctly specified.
- Tubesheets are frequently paired to cylinders or flanges. Click on the front end flange element just to the left of the tubesheet for proper definition, then click **Tubesheet Analysis** to define the assembly. Enter data on each tab of the **Heat Exchanger Tubesheet Input** dialog box. The software dynamically adds or removes tabs depending on the type of exchanger selected. For example, an exchanger with a floating head requires more input and displays more tabs than a U-tube type exchanger.
- Tubesheets that are either completely integral or integral on the channel side are attached to the channel cylinder. These tubesheets must have a value for **Distance from "From" Node** to define the length of that cylinder because they start at the end of the channel cylinder.
- Verify that the 3D graphic looks like the needed geometry. Use the transparency feature to see inside of the exchanger. The software does not draw all tubes to improve performance.

NOTE Examples are located in the PV Elite installation folder.

Delete

Deletes all tubesheet data.

HTRI In

Select to open a .dbo HTRI output file.

See also

Tubesheet Type and Design Code Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 134)
Tubesheet Properties Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 138)
Tube Data Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 148)
Expansion Joint Data Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 162)
Load Cases Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 168)
Floating TubeSheet Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 173)
Spherical Cover/Backing Ring Tab (Heat Exchanger Tubesheet Input Dialog Box) (page 177)

Tubesheet Type and Design Code Tab (Heat Exchanger Tubesheet Input Dialog Box)

General Exchanger Data

Tubesheet Analysis Method

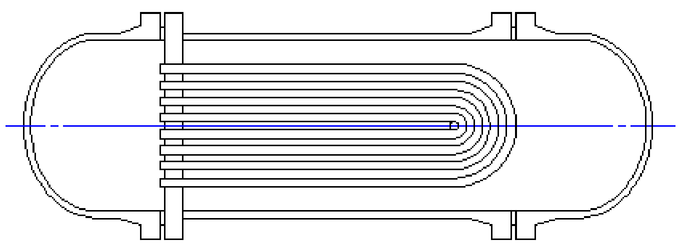
Displays the standard to use for analysis:

- **TEMA** - Tubular Exchanger Manufacturers Association
- **ASME** - ASME Section VIII, Division 1, Section UHX or ASME Section VIII, Division 2, Part 4
- **NOTE** The option selected in **Design Code** determines whether the software uses Division 1 or Division 2.
- **PD 5500 (British Code)** - British Pressure Vessel Code, Section 3.9, Flat Heat Exchanger Tubesheets
- **EN-13445**

Exchanger Type

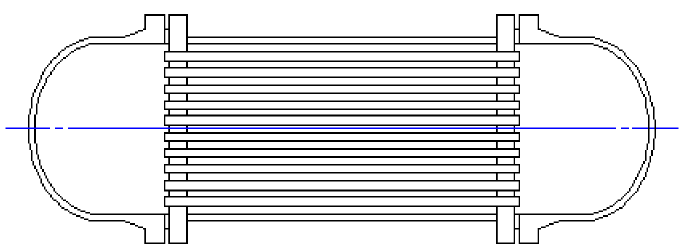
Select the type of heat exchanger.

- **U-Tube** has only one tubesheet. The tubes are bent in the form of a "U." The bundle can removed for maintenance, but the inside of the tube is harder to clean because of the bend.

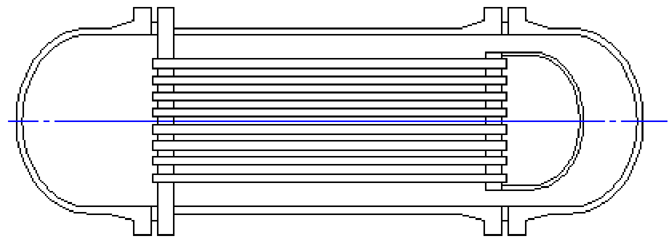


- **Fixed** has two tubesheets that are fixed at each end of the exchanger and are connected to each other by straight tubes. Both the tubesheets are stationary, so differential thermal expansion can develop between the shell and the tubes. An expansion joint is sometimes required to absorb the thermal growth.

  NOTE You must select a fixed type of tubesheet exchanger to perform finite element analysis (FEA) on expansion joints.



- **Floating** has one tubesheet that is fixed (stationary) and one that is free to move. Because one tubesheet *floats*, any differential thermal expansion between the shell and tubes is absorbed. This category of exchangers is the most versatile and also the costliest. Tubes can also be cleaned easily compared to U-tube exchangers.



Expansion Joint Type (if any)

Select the expansion joint type.

- **No Joint** - The exchanger type has no expansion joint type.
- **Thin Bellows Type** - The joint is comprised of a thin bellows that is very flexible and has low stiffness. Analysis is performed according to ASME Appendix 26 for thin bellows expansion joint type.
- **Thick Joint Type (Flanged and Flued)** - The joint is comprised of a number of shell elements that are added together to form the expansion joint. A thick joint is stiffer than the thin joint and its stiffness must be taken into account. Analysis is performed according to Appendix 5 guidelines, TEMA or Kopp & Sayre Method. You can also perform finite element analysis (FEA) on thick expansion joints. For more information, see *Finite Element Analysis (FEA) on ASME or TEMA Expansion Joints (PV Elite)* (page 172).

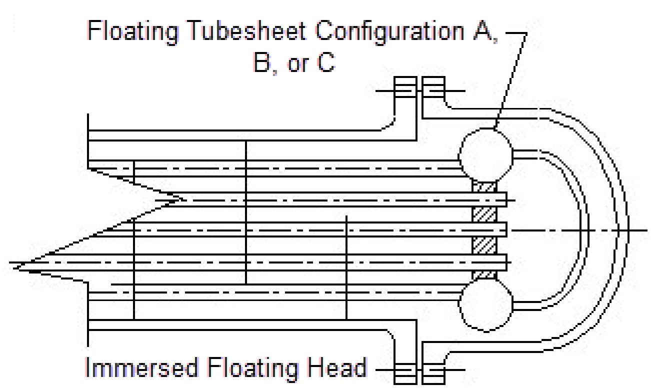
NOTE Click **Vacuum Pressures and Report Options for this Load Case** to open the **Report Print Options** dialog box, then select **Print Intermediate Results for Expansion Joint Calculations**.

ASME or EN-13445

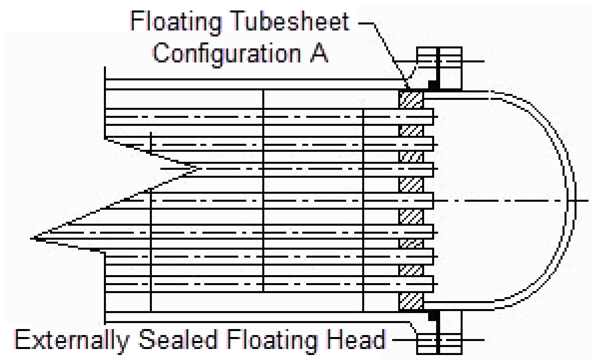
Floating Exchanger Type

Select the type of floating exchanger, as defined in Division 1 and Division 2 of the ASME code:

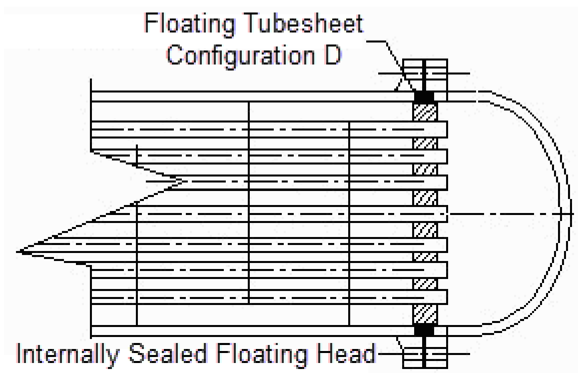
- **Exchanger with an Immersed Floating Head**



- **Exchanger with an Externally Sealed Floating Head**



- **Exchanger with an Internally Sealed Floating Head**



Tubesheet/Shell Junction Stress Reduction Option

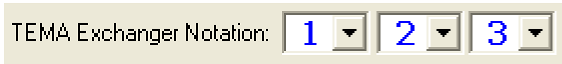
Select the option used to reduce the possible over stress at the junction of the tubesheet and the integral cylinder. If the U-tube tubesheet is welded to the shell then the integral cylinder is the shell cylinder and if the tubesheet is welded to the channel then it is the channel cylinder. The U-tube stress reduction options are displayed below:

- **Increase Tubesheet Thickness**
- **Increase Integral Cylinder Thickness** - Increase for the shell, or channel, or both.
- **Increase Cylinder and Tubesheet Thickness** - Increase for both the tubesheet and the integral cylinder.
- **Perform Elastic-Plastic Calculation** - Performing the elastic-plastic calculation at that junction is recommended if all prerequisites are satisfied. For example, the equipment is not operating in the temperature creep range where time-dependent properties affect material allowables.

TEMA

TEMA Exchanger Notation

For a TEMA analysis, select a value for each of the three options. The order of the options is:



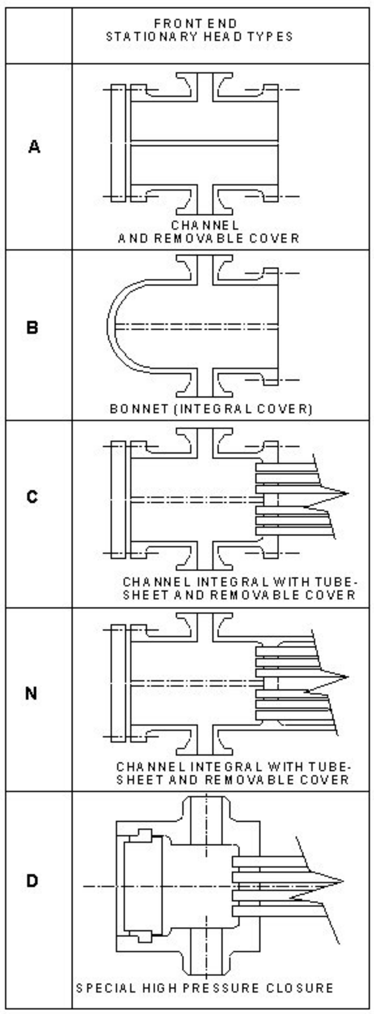
1 - Front End

2 - Shell Type

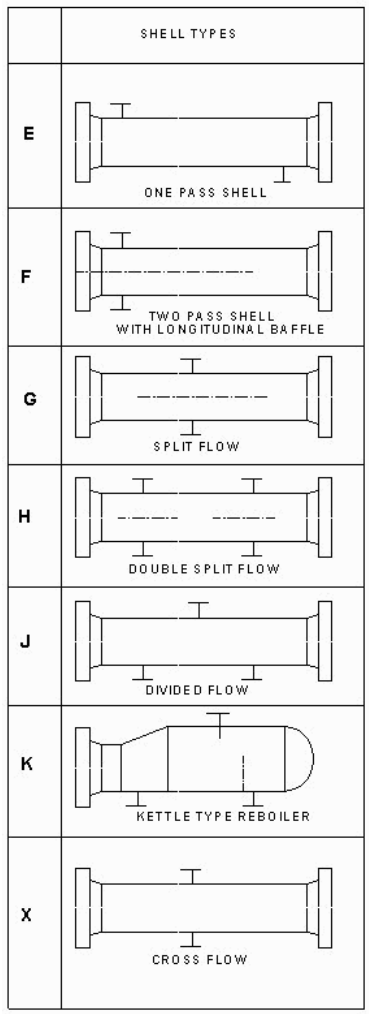
3 - Rear End

The nomenclature for each option is according to the conventions in TEMA page 2, figure N-1.2:

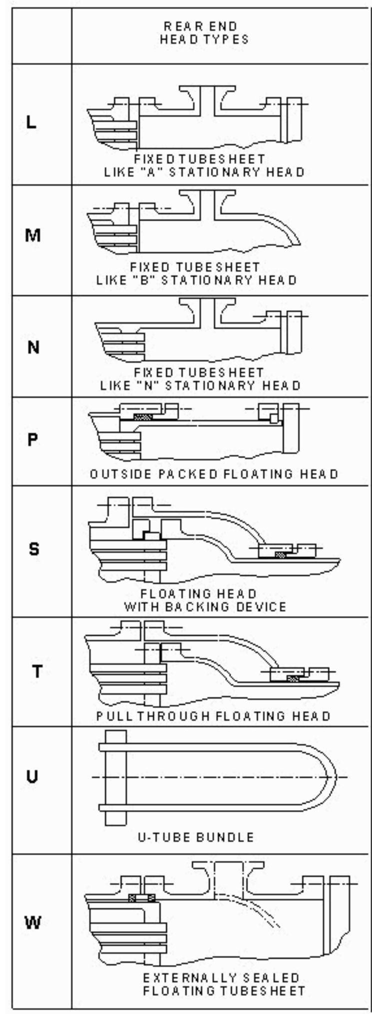
Front End



Shell Type



Rear End



TEMA Exchanger Class

For a TEMA analysis, select one of the classes of exchangers, **R**, **C**, or **B**. The class is determined by the severity of service and is discussed in the TEMA code, paragraph RCB-1.1. The calculation method is the same for all classes, but each class has its own design restrictions. Verify the correct class to use for your application.

Tubesheet Properties Tab (Heat Exchanger Tubesheet Input Dialog Box)

Description

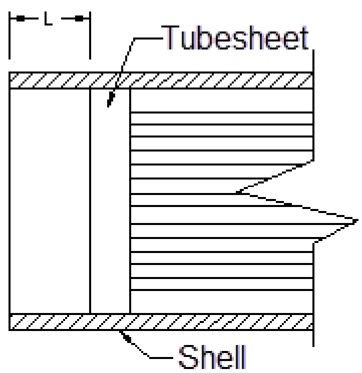
Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

Element From Node

Displays the **From Node** for the selected element.

Dist. from "From" Node

Enter the distance from the shell to the outer face of the nearer tubesheet.



Tubesheet Type

Select the type of tubesheet to analyze, according to ASME classifications. The available selections change depending on the **Exchanger Type** selected on the **Tubesheet Type and Design Code** tab.

- **Fixed** tubesheet exchangers are subject to loads arising from differential thermal expansion between the tubes and the shell. They have stationary tubesheets on both sides. Fixed tubesheet exchangers are classified by ASME:

Configurations	Description
A	Tubesheet integral with both shell and channel
B	Tubesheet integral with shell, gasketed with channel, with tubesheet extended as a flange
C	Tubesheet integral with shell, gasketed with channel, with tubesheet not extended as a flange
D	Tubesheet gasketed with both shell and channel

NOTE You must select a fixed tubesheet type to perform finite element analysis (FEA).

- **U-Tube** exchangers are classified either as integral with the shell, channel, both, or gasketed on both sides, according to ASME classifications:

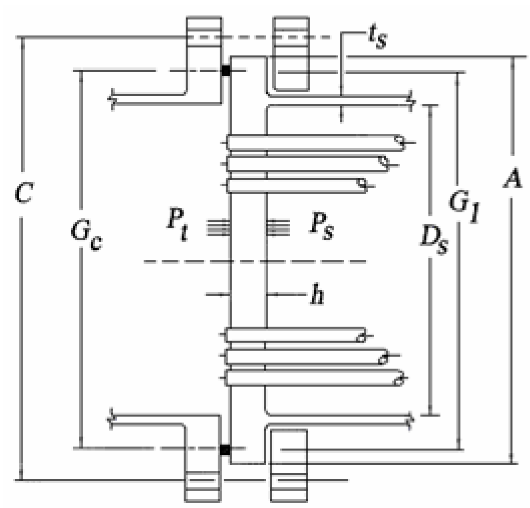
Configurations	Description
A	Tubesheet integral with both shell and channel
B	Tubesheet integral with shell, gasketed with channel, with tubesheet extended as a flange
C	Tubesheet integral with shell, gasketed with channel, with tubesheet not extended as a flange
D	Tubesheet gasketed with both shell and channel
E	Tubesheet integral with channel, gasketed with shell, with tubesheet extended as a flange
F	Tubesheet integral with channel, gasketed with shell, with tubesheet not extended as a flange

- **Floating** tubesheet exchangers have a stationary tubesheet and a floating tubesheet, according to ASME classifications:

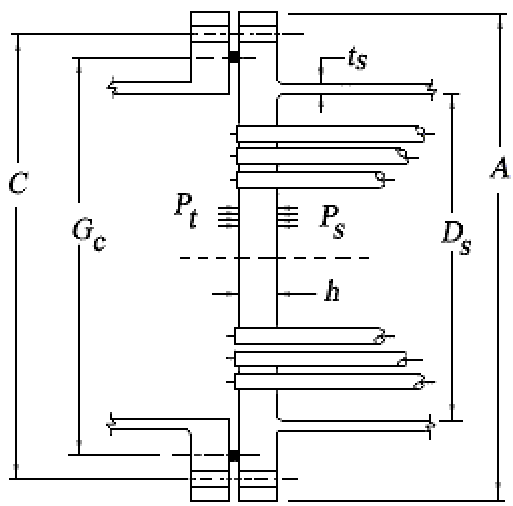
Configurations	Description
A	Tubesheet integral
B	Tubesheet gasketed and extended as a flange
C	Tubesheet gasketed and not extended as a flange
D	Tubesheet internally sealed

Some ASME tubesheet configurations are shown below:

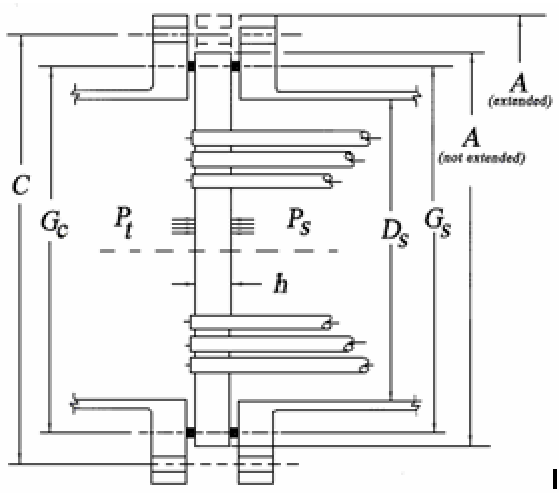
- Tubesheet is integral with the shell and is gasketed on the channel side and is not extending as a flange



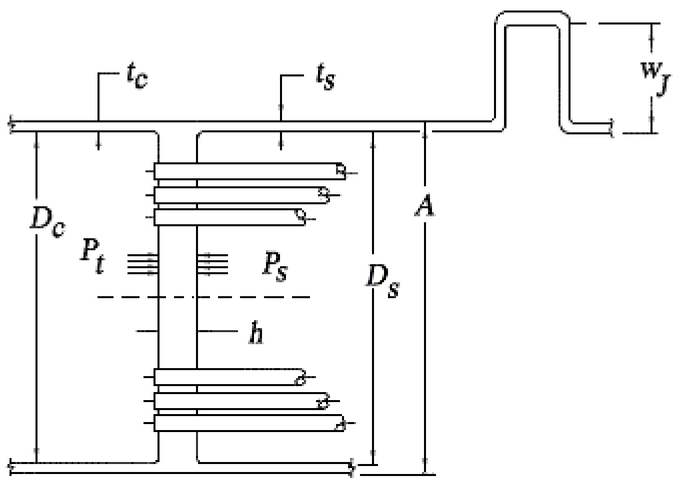
- Tubesheet is integral with the shell and is gasketed on the channel side and is extending as a flange



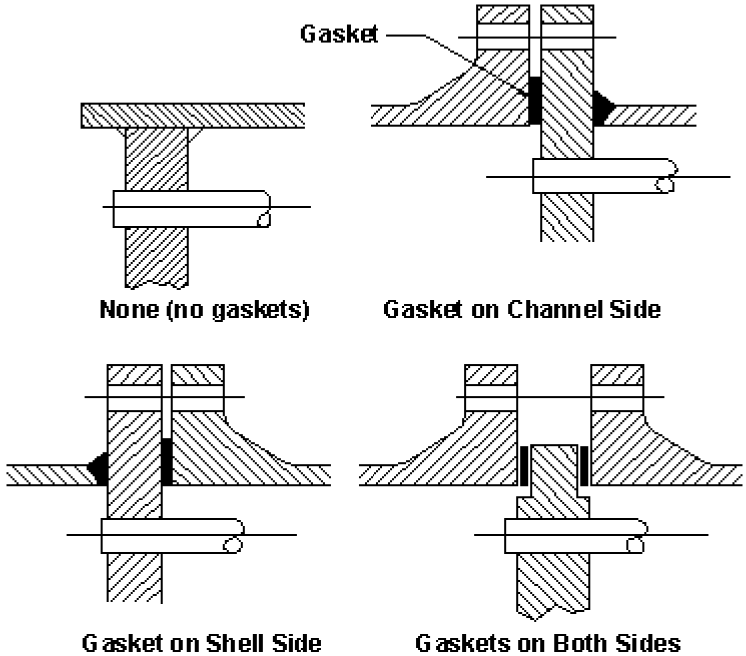
- Tubesheet is gasketed on both the shell and the channel sides and is not extended as a flange. In an alternative arrangement the tubesheet is extending as a flange.



- Tubesheet is integral with both the shell and the channel. This is a fixed tubesheet exchanger, as a flanged and flued expansion joint is used to reduce the differential thermal expansion, between the tubes and the shell.



The following tubesheet attachment types are used:



Outside Diameter

Enter the outside diameter of the tubesheet. If the tubesheet is stepped, click **Stepped Tubesheet** to open the **Stepped Tubesheet Effective Diameter Calculation** dialog box and define the tubesheet dimensions and calculate the tubesheet's effective diameter.

NOTE The **Tubesheet Extended as Flange?** check box must be selected for **Stepped Tubesheet** to be enabled.

Stepped Tubesheet

Click to open the **Stepped Tubesheet Effective Diameter Calculation Dialog Box** (page 147).

Tubesheet Thickness

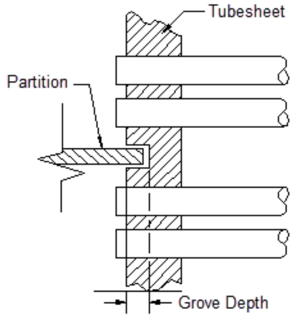
Enter the tubesheet thickness in an uncorroded condition. If it is a re-rate, then the actual measured thickness is typically used.

Corr. Allow. Shell Side / Channel Side

Enter the corrosion allowance on the shell side (the inner face of the tubesheet), and the corrosion allowance on the channel side (the outer face of the tubesheet facing the channel side).

Depth of Groove in Tubesheet (if any)

Enter the depth of a groove in the tubesheet, used to locate the channel partition plate and its gasket. If there is no groove, such as in a single pass exchanger, this value is **0**.



Weld Leg at Back of Tubesheet (if any)

If the tubesheet is welded to shell and/or channel, then enter the fillet weld length at the back of the tubesheet.

Tubesheet Assembly is Down/Left?

Select if a horizontal U-tube exchanger is modeled with the tube bundle facing left and the tubesheet on the right or for a vertical U-tube exchanger.

Tubesheet Extended as Flange?

Select if the tubesheet is extended as a flange, so that it is subject to the bolt load from the mating flange.

NOTE This option must be selected for **Stepped Tubesheet** to be enabled.

Thickness of Extended Portion

When **Tubesheet Extended as Flange?** is selected, enter the thickness of the portion of the tubesheet that is extended for bolting.

Tfr/T Ratio for U-Tubesheets (optional)

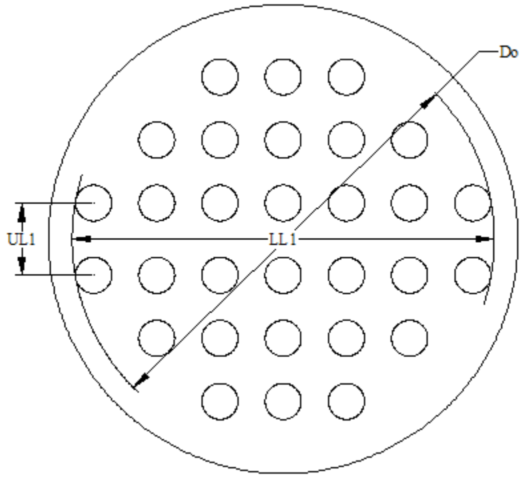
When **Tubesheet Extended as Flange?** is selected, enter the ratio of the required thickness of the tubesheet flanged extension to the tubesheet, if reducing the required thickness of the flanged extension is required. The ratio should be less than **1.0** and more than **0.2**. The default value is **1.0**. This value is used in TEMA RCB 7.1342 for U-tube tubesheet exchangers. This entry is optional.

Bolt Load Transferred to Tubesheet?

When **Tubesheet Extended as Flange?** is selected, also select this option if the bolt load is transferred to the tubesheet, extended as the flange. Do not select this option if the tubesheet is gasketed with both the shell and channel flanges. Otherwise, the tubesheet can still be extended, but the bolt load is not transferred to the tubesheet extension. Carefully consider all possible cases, such as the hydrotest. When this option is not selected, the required thickness of tubesheet extension is not calculated.

UnTubed Lane Area

Enter the total area of all the untubed lanes on the tubesheet. If there is no pass partition lane then the value is **0**. This value is only needed for ASME code analysis. In a single pass exchanger, this area is UL1 * Do:



The area is UL1 * Do

The maximum limiting value of is 4*Do*p.

Do - Equivalent diameter of outer limit circle

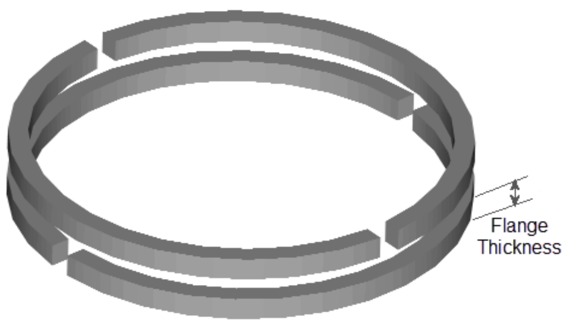
p - Tube pitch

UL1 - Distance between innermost tube hole centers (width of pass partition lane)

Backing Ring

Backing Ring Thickness

Enter the actual thickness of the backing ring. This value is needed when a tubesheet is clamped and gasketed on one side by a backing ring or device. For doubly-split rings, this is the thickness of each piece:

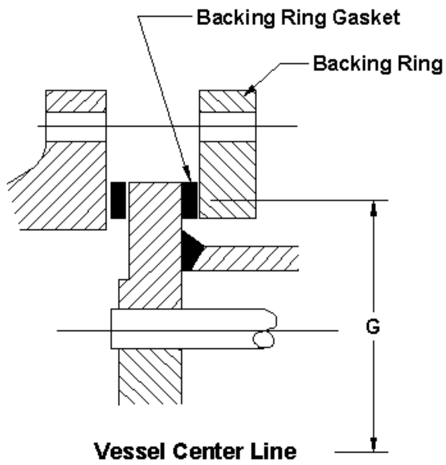


Backing Ring ID / OD

Enter the inside and the outside diameters of the backing ring.

G Dimension for Backing Ring

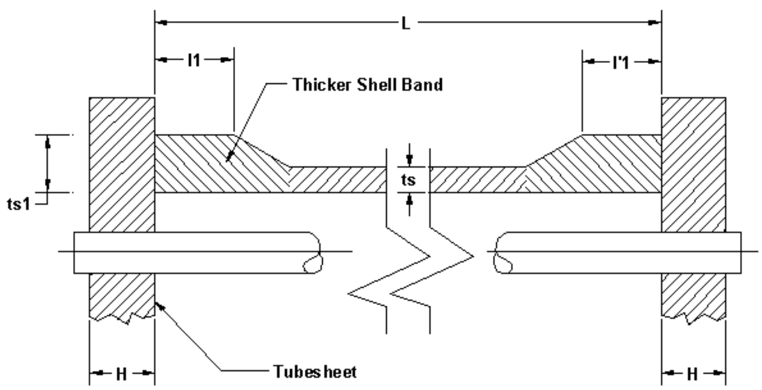
Enter the effective diameter *G*, defined as the mid-point of the contact between the backing flange and the tubesheet. When a tubesheet is clamped and gasketed on one side by a backing ring or device, the effective diameter of the gasket between the backing ring and the tubesheet is needed.



ASME / EN-13445 Shell Band Data

Is There a Shell Band?

Select if there is a shell band. The shell might have thicker courses at either end, called shell bands. Shell bands give added strength to the shell to tubesheet region:



Shell Thickness Adjacent to Tubesheet

Enter the thickness of the shell bands *ts1*.

Shell Band Corrosion Allowance

Enter the corrosion allowance for the shell band.

Shell Band Length Adjacent to Tubesheet, Front End L1

Enter the front end length *l1* for the shell band.

Shell Band Length Adjacent to Tubesheet, Rear End L1'

Enter the rear end length *l1'* for the shell band.

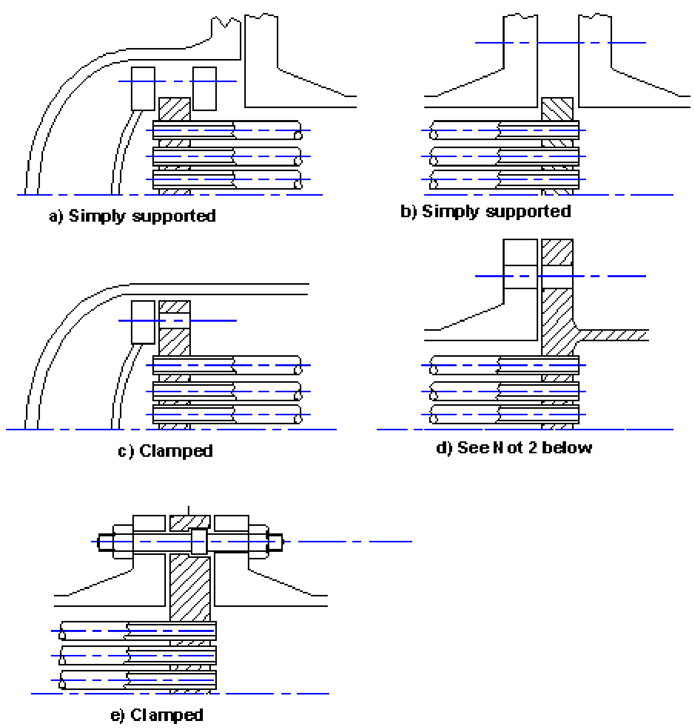
NOTE Also specify the shell band material on the **Load Cases** tab.

PD 5500

How are Tubesheets Clamped

Select the method of clamping for a fixed or a floating tubesheet, as defined by PD 5500, Figure 3.9-6:

Stationary Tubesheet	Floating Tubesheet
Simply Supported	Simply Supported
Simply Supported	Clamped
Clamped	Simply Supported
Clamped	Clamped



If either shell or channel is welded to a U-tubesheet, then:

- The U-tubesheet is simply supported if:

$$\frac{4.3 \cdot D_{star} + 0.65 \cdot D_1 \cdot K_\theta}{1.3 \cdot D_{star} + 0.5 \cdot D_1 \cdot K_\theta} > 2.3$$

- The U-tubesheet is clamped if:

$$\frac{4.3 \cdot D_{star} + 0.65 \cdot D_1 \cdot K_\theta}{1.3 \cdot D_{star} + 0.5 \cdot D_1 \cdot K_\theta} < 2.3$$

Stepped Tubesheet Effective Diameter Calculation Dialog Box

The Stepped Tubesheet Effective Diameter Calculation dialog box allows you to define the dimensions of a stepped tubesheet in accordance with Part UHX-10(b) of the ASME code. As you enter the tubesheet dimensions, the status bar at the bottom of the dialog displays computed results including the effective tubesheet diameter which will be used in the ASME tubesheet calculations.

Is this a stepped tubesheet?

Select to indicate that the tubesheet is stepped.

Flat on tube side

Select to indicate that the tubesheet is flat on the tube side.

Step 1 (Tube Side)

Select to define the dimensions for the first step of the tubesheet on the tube side.

Step 1 Diameter (dt1)

Enter the diameter of the first step on the tube side of the tubesheet.

Step 1 Depth (ht1)

Enter the depth of the first step on the tube side of the tubesheet.

Step 2 (Tube Side)

Select to define the dimensions for the second step of the tubesheet on the shell side.

Step 2 Diameter (dt2)

Enter the diameter of the second step on the tube side of the tubesheet.

Step 2 Depth (ht2)

Enter the depth of the second step on the tube side of the tubesheet.

Flat on shell side

Select to indicate that the tubesheet is flat on the shell side.

Step 1 (Shell Side)

Select to define the dimensions for the first step of the tubesheet on the shell side.

Step 1 Diameter (ds1)

Enter the diameter of the first step on the shell side of the tubesheet.

Step 1 Depth (hs1)

Enter the depth of the first step on the shell side of the tubesheet.

Step 2 (Shell Side)

Select to define the dimensions for the second step of the tubesheet on the shell side.

Step 2 Diameter (ds2)

Enter the diameter of the second step on the shell side of the tubesheet.

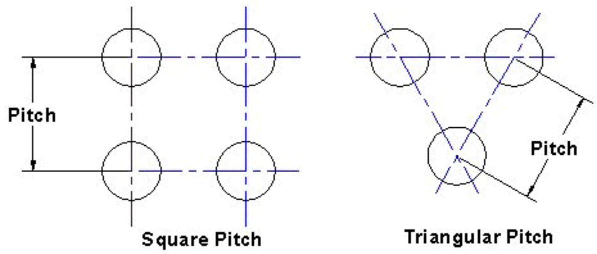
Step 2 Depth (hs2)

Enter the depth of the second step on the shell side of the tubesheet.

Tube Data Tab (Heat Exchanger Tubesheet Input Dialog Box)

Number of Holes / Pattern

Enter the total number of tube holes drilled in one of the tubesheets. Also select the hole pattern: **Square**, or **Triangular**. The code expects the holes to be fairly evenly spaced over the entire area of the tubesheet without large areas that are not drilled.



NOTE For a square rotated pattern, select **Square**. For a triangular rotated pattern, select **Triangular**.

Wall Thickness / Corrosion Allowance

Enter the wall thickness of the exchanger tubes, and the corrosion allowance to which the tubes are subjected. Typical tube thicknesses are below:

Tube O.D. inches (mm)	B.W.G. gage	Thickness inches	Thickness mm
1/4 (6.35)	22	0.028	0.711
	24	0.022	0.559
	26	0.018	0.457
	27	0.016	0.406
3/8 (9.53)	18	0.049	1.245
	20	0.035	0.889
	22	0.028	0.711
	24	0.022	0.559
1/2 (12.7)	16	0.065	1.651
	18	0.049	1.245
	20	0.035	0.889
	22	0.028	0.711

Tube O.D.	B.W.G.	Thickness	Thickness
5/8 (15.88)	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	17	0.058	1.473
	18	0.049	1.245
	19	0.042	1.067
	20	0.035	0.889
3/4 (19.05)	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	17	0.058	1.473
	18	0.049	1.245
	20	0.035	0.889
7/8 (22.23)	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651

Tube O.D.	B.W.G.	Thickness	Thickness
	17	0.058	1.473
	18	0.049	1.245
	20	0.035	0.889
1 (25.4)	8	0.165	4.191
	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	15	0.072	1.829
	16	0.065	1.651
	18	0.049	1.245
	20	0.035	0.889
1-1/4 (31.75)	7	0.180	4.572
	8	0.165	4.191
	10	0.134	3.404
	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
	16	0.065	1.651
	18	0.049	1.245
	20	0.035	0.889
1-1/2 (38.10)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108

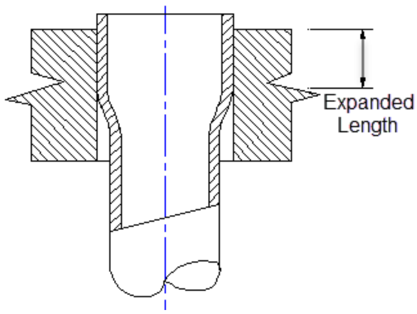
Tube O.D.	B.W.G.	Thickness	Thickness
	16	0.065	1.651
2 (50.8)	11	0.120	3.048
	12	0.109	2.769
	13	0.095	2.413
	14	0.083	2.108
2-1/2 (63.5)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108
3 (76.2)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108
2-1/2 (63.5)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108
3 (76.2)	10	0.134	3.404
	12	0.109	2.769
	14	0.083	2.108

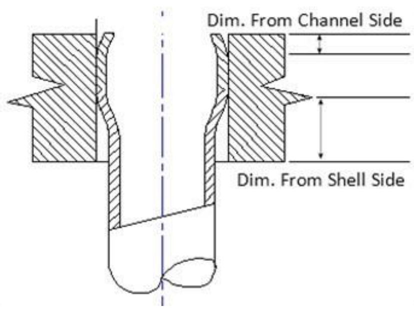
Outside Diameter / Pitch

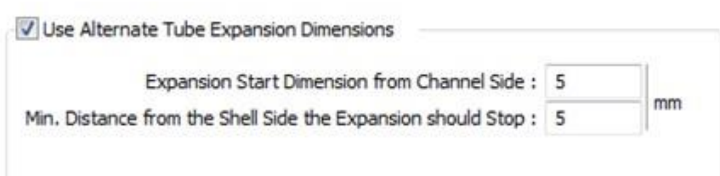
Enter the outside diameter and the pitch of the tubes. The tube pitch is the distance between the centers of the adjacent tubes.

Length of Expanded Portion of Tube

Enter the length of tube that is expanded into the tubesheet hole. This value may not exceed the full thickness of the tubesheet to avoid failure of the tube at the inner tubesheet face and is usually 80% to 90% of the tubesheet thickness.







In most cases, tubes are not fully expanded through the thickness of the tubesheet. There is a typical distance of a few millimeters that is not expanded.

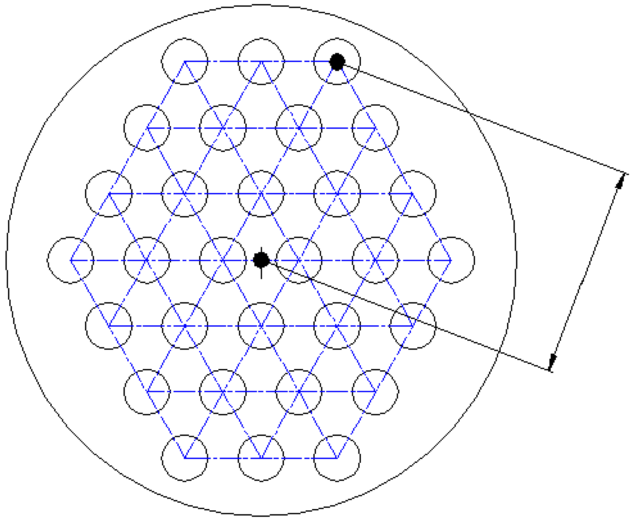
Select the **Use Alternate Tube Expansion Dimensions** check box, to enable the **Expansion Start Dimension from Channel Side** and **Min. Distance from the Shell Side the Expansion should Stop** fields in which to enter the dimensions.

When given the dimensions, the expanded length of the tube in the tubesheet can be computed in cases involving both the corroded and uncorroded conditions.

The dimensions are measured from the face of the tubesheet in the new condition.

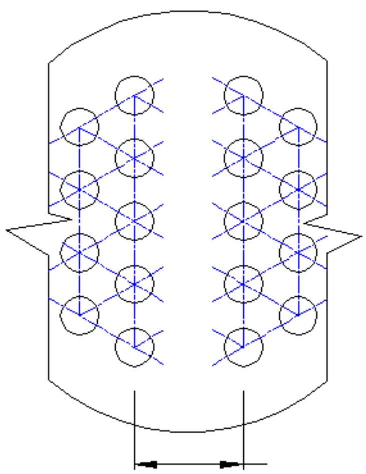
Radius to Outermost Tube Hole Center

Enter the distance from the center of the tubesheet to the centerline of the tube furthest away.



Distance Between Innermost Tube Centers

Enter the maximum distance between the tube innermost centers when a partition plate is installed, because the innermost lanes of tubes may be further apart than the general tube pitch in the remainder of the tubesheet. This is the maximum distance between the tube innermost centers. If there is no partition plate, this value is **0**.

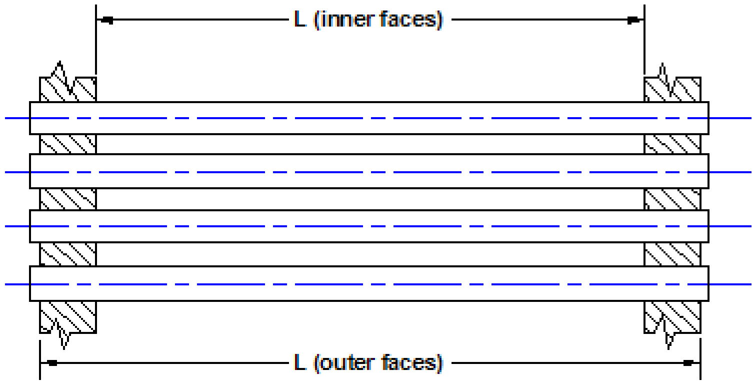


Straight Tube Length

Enter the straight length of the tubes, based on the selection for **Straight Tube Length Measured Between**.

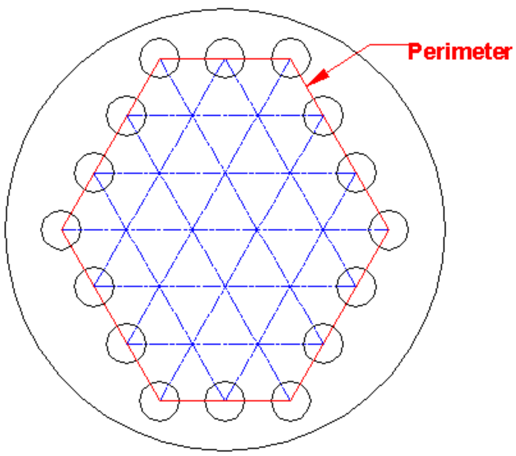
Straight Tube Length Measured Between

Select the method of tube length measurement. Select **Inner Faces** or **Outer Faces**.



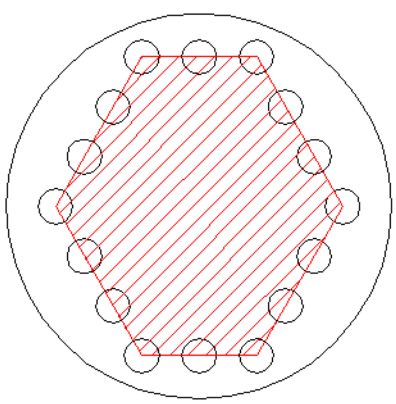
Perimeter of Tube Layout (if needed)

Enter the total linear length of the outermost tubes. This value is only required if the software is calculating the punching shear stress. Otherwise, the value is **0**.



Area of Tube Layout (if needed)

Enter the area defined by the tube layout, including all the tubes in the tube bundle.



Tube Layout Assistant

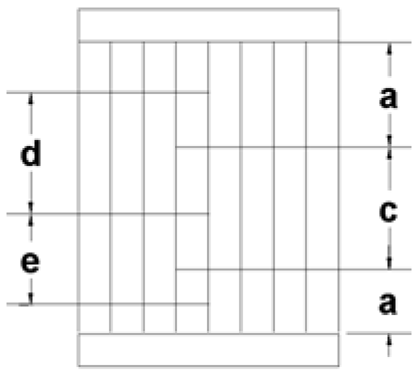
Opens the **Tube Layout Assistant** (page 160) utility. This utility helps you determine the number of tubes that will fit in a layout configuration for a heat exchanger tubesheet.

Import Layout Results

Imports results saved in the **Tube Layout Assistant**, and fills in values for basic tube data from the results.

Max. Dist. from Tubesheet to 1st Tube Support

Enter the maximum distance between two tube supports. Supports, often called baffles, are along the heat exchanger, and are in addition to the tube support provided by each of the tubesheets. The software uses the maximum unsupported length to determine the buckling stress in the tubes. Carefully examine the design of the exchanger, and enter the maximum possible unsupported length. For the example below, *a* is the distance between a baffle and a tubesheet, while *c*, *d*, and *e* are the distances between baffles.



Max. Dist. bet. 2 Tube Supports

Enter the maximum distance from the tubesheet to the first tube support (baffle). This is the maximum of the *a* distances shown above.

End Condition k

Enter the tube end condition value *k*, as defined in the TEMA, ASME, and PD 5500 codes. Each code uses different values, so verify that you use a value for the code defined for your model.

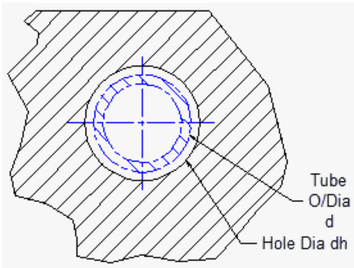
Condition	'k' Value	
	TEMA / ASME	PD 5500
Between two tubesheets	0.60	0.50
Between tubesheet and baffle	0.80	0.707
Between two baffles	1.00	1.00

Max Unsupported Len SL

Enter *l* x *k* as the effective buckling length of the tubes between supports.

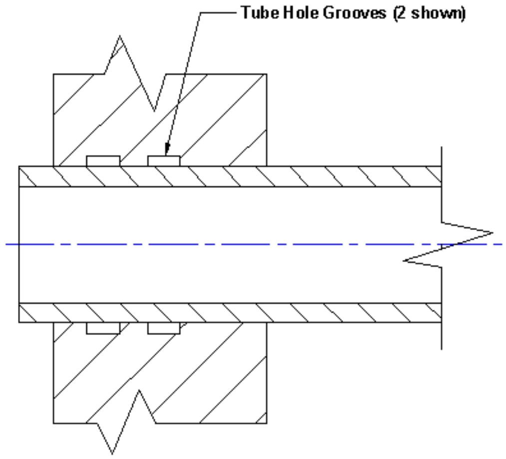
Tube Hole Diameter, dh

Enter the diameter of the holes drilled in the tubesheet. The tube hole diameter should be slightly larger than the outside diameter of the tube. This provides a clearance that is closed as the tube is expanded in the hole.



Number of Grooves in Hole

Enter the number of grooves machined into the tube hole.

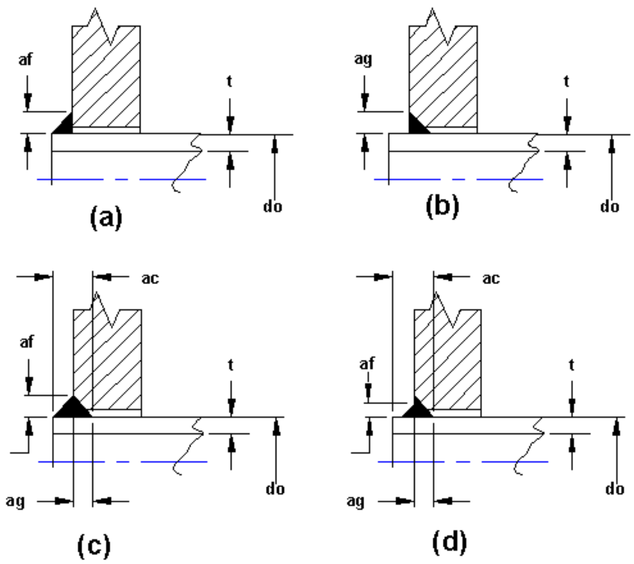


Fillet Weld Leg Size (if any)

Enter the fillet weld size, if used when the tube is welded to the tubesheet, as defined in ASME VIII Div 1, UW-20. The weld may be fillet-only, groove-only, or both, as shown below.

Groove Weld Leg Size (if any)

Enter the groove weld size, if used when the tube is welded to the tubesheet, as defined in ASME VIII Div 1, UW-20. The weld may be fillet-only, groove-only, or both, as shown below.



Design Strength (not for fixed TS types)

Enter the design strength axial load *Fd*, according to ASME Code paragraph UW-20. This value is used to determine the minimum acceptable fillet/groove weld size that connects the tube to the tubesheet. The design strength should not be greater than the tube strength (*Ft*), defined as $\Pi t(do - t)Sa$.

NOTES

- Because U-tube tubesheet exchangers do not experience differential thermal expansion between the tubes and the shell, the axial load on the tubes cannot be easily calculated. You can specify your own value for the actual load (required design strength) for U-tube exchangers. This value is optional for fixed and floating tubesheet exchangers.
- For partial strength tube-to-tubesheet welds on fixed/floating tubesheet exchangers, the higher of the actual tube-to-tubesheet load and the entered design strength is used to size the welds.
- For full strength tube-to-tubesheet welds on fixed/floating tubesheet exchangers, the tube strength (*Ft*) is used to size the welds.

Tube Weld Joint Type

Select the type of tube/tubesheet weld, as defined in ASME UW-20:

- **Full Strength** - The design strength is equal to or greater than the maximum allowable axial tube strength.
- **Partial Strength** - Design strength is based on the actual tube-tubesheet axial load.
- **Seal/No Weld** - The weld is used to seal and has no strength value, so no calculations are performed.

Tube Joint Type

Select the weld joint type, as defined by TEMA and ASME using ASME Section VIII, Division 1 Table A-2, Efficiencies and Joint Types:

Joint Type	Description	Fr.(test)	Fr.(no test)
a	Welded only, a >= 1.4t	1.00	.80
b	Welded only, t <= a < 1.4 t	.70	.55
b-1	Welded only, a < t	.70	...
c	Brazed, examined	1.00	.80
d	Brazed, not fully examined	0.50	.40
e	Welded, a>=1.4t, expanded	1.00	.80
f	Welded, a<1.4t, expanded, enhanced with 2 or more grooves	.95	.75
g	Welded, a<1.4t, expanded, enhanced with 1 groove	.85	.65
h	welded a 1.4t, expanded, not enhanced (no grooves)	.70	.50
i	Expanded, enhanced with 2 or more grooves	.90	.70
j	Expanded, enhanced with single groove	.80	.65
k	Expanded, not enhanced (no grooves)	.60	.50

For PD 5500, select the weld joint type as defined by Table 3.9-3, Efficiencies and Joint Types:

Joint Type	Description	Fr.(1)
a	Welded with min throat thk. >= tube thk.	.80
b	Welded with min throat thk. < tube thk.	.55
c	Expanded and welded with min throat thk. >= tube thk.	.80
d	Expanded and welded with min throat thk. < tube thk.	.55
e	Expanded only	.50
f	Explosion expanded/welded	.80

Allowable Joint Load Method

Select the joint allowable load method:

- **ASME APP. A** – Applicable for fixed and floating tubesheet heat exchangers, covering many types of tube-tubesheet joints, such as welded, brazed and expanded. The British code PD 5500 method for determining the tube-tubesheet joint allowable is similar to this method.
- **ASME UW-20** – Applicable for full strength and partial strength tube-tubesheet welds.
- **None**

Is Tube-Tubesheet Jt. Tested?

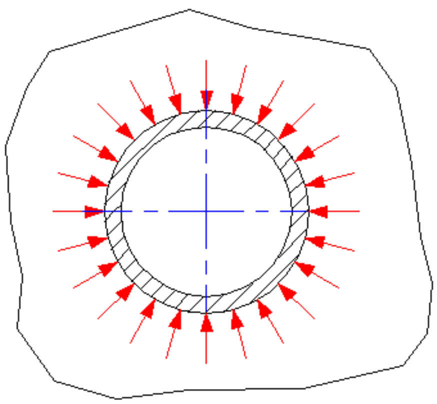
Select if the tube/tubesheet joint is tested and qualified for ASME. Tested joints get an increase in their strength value.

ASME Tube Jt. Reliability Factor

Displays the reliability factor value after a value for **Tube Joint Type** is selected.

Tube Expansion, Po

Enter the pressure (*Po*) exerted on the outside of the tube after it has expanded. This value is only required for a **Tube Joint Type** of **i**, **j**, and **k**.



Differential Thermal Expansion, Pt

Enter the interface pressure (*Pt*) between the tube and the tubesheet due to differential thermal growth. This value is only required for a **Tube Joint Type** of **i**, **j**, and **k**.

★ IMPORTANT The ASME code provides neither formulae to calculate *Po* and *Pt* pressures, nor indicates analytical or experimental methods to establish them. If you do not have good method to establish values of interface pressures then enter **0**. This tells the software to ignore the effects of interface pressures on the joint allowable. If you enter **1** as an approximation for both of these pressures, then the joint allowable doubles.

Is This a Welded Tube (Not Seamless)?

Select if the tubes are of welded construction.

Specific Gravity of the Operating Liquid in the Tubes

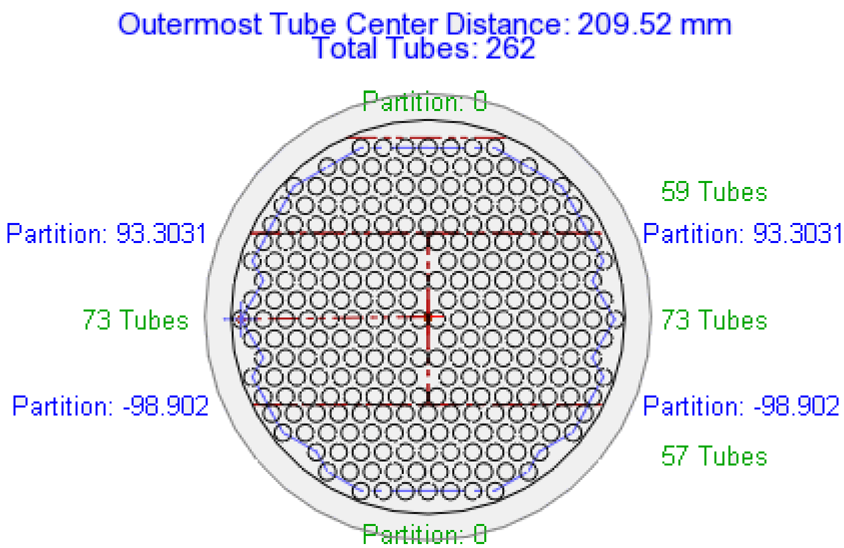
Enter the specific gravity of the fluid in the tube channel side.

Tube Layout Assistant

The **Tube Layout Assistant** utility helps you determine the number of tubes that will fit in a layout configuration for a heat exchanger tubesheet. The utility is available from:

- The **Tube Data** tab of the **Heat Exchanger Tubesheet Input** dialog box, opened from **Tubesheet Analysis** in PV Elite.
- **TSLayout.exe**, found in the *[Program Folder]*\Intergraph CAS\PV Elite*[Version Number]* folder.

The distance from the center of the tubesheet to the outermost tube center and the total number of tubes is calculated. As you change options on the **Layout Pattern** tab, the layout graphics dynamically update in the right panel. For example:



NOTES

- The outermost tube center distance is used in tubesheet stress analysis.
- Tie rods that are located in tube positions are not considered.
- Layout files are saved with the .tsd extension. You can import a layout file into PV Elite.

Layout Pattern Tab

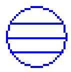
Units

Select **Imperial (in)** or **Metric (mm)**. You can switch units at any time.

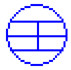
Partitions

Partition Layout

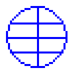
Select the partition plate layout within the shell:

 - Horizontal partitions only. With this layout, **Number of Horizontal Partitions** can have values from **0** to **8**.

NOTE For a tubesheet with no partitions, select this option and set **Number of Horizontal Partitions** to **0**.



- Horizontal partitions with an interior vertical partition. With this layout, **Number of Horizontal Partitions** can have values from **2** to **8**.



- Horizontal partitions with a full vertical partition. With this layout, **Number of Horizontal Partitions** can have values from **2** to **8**.

Number of Horizontal Partitions

Type or select the number of horizontal partitions.

Partition Thickness

Type the thickness of the partition plates.

Partition Clearance

Type the clearance distance between tubes and the partition plates.

Symmetrical (Mirror) Layout

Select for a symmetrical tube layout.

Tubes

Tube Pattern

Select the needed tube pattern. Select **Square**, **Square Rotated**, **Triangular**, or **Triangular Rotated**.

Tube Boundary (OTL)

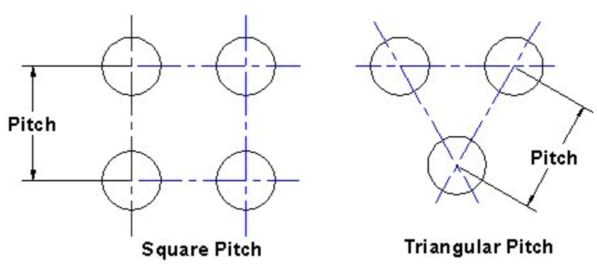
Enter the outer tube limit (OTL) diameter of the shell. Tubes cannot be placed beyond the OTL.

Tube Outer Diameter

Enter the un-corroded outside diameter of each tube.

Tube Pitch

Enter the distance between tube centers.



Nozzle

Nozzle Clearance (Top, Bottom, Left, Right)

Enter the dimension that a nozzle projects inside the surface of the exchanger shell. Note that all dimensions can be specified. This allows for side and vertical entry nozzles.

Nozzle Clearance (Top)

Enter the distance that a nozzle projects vertically and from the top of the surface of the exchanger shell.

Nozzle Clearance (Bottom)

Enter the distance that a nozzle projects vertically and from the bottom of the surface of the exchanger shell.

Nozzle Clearance (Left)

Enter the distance that a nozzle projects horizontally and from the left of the surface of the exchanger shell.

Nozzle Clearance (Right)

Enter the distance that a nozzle projects horizontally and from the right of the surface of the exchanger shell.

Shell

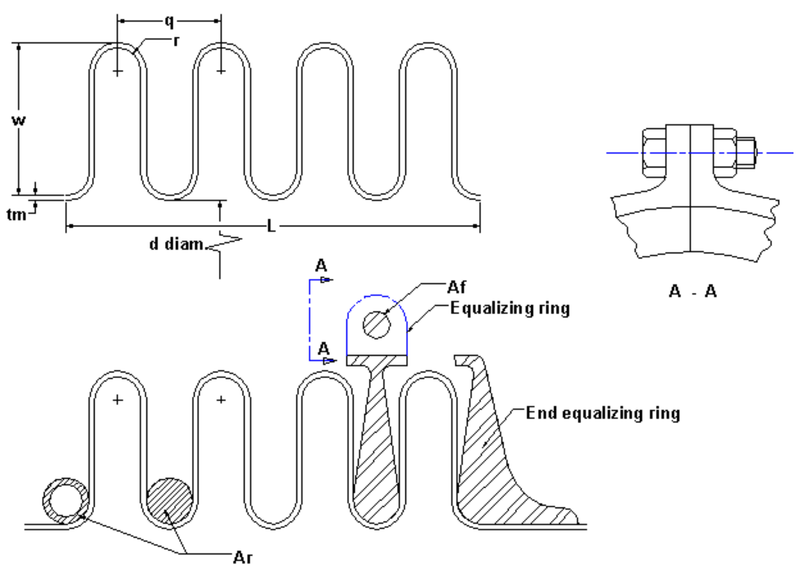
Shell Inside Diameter

Enter the inside diameter of the shell.

Expansion Joint Data Tab (Heat Exchanger Tubesheet Input Dialog Box)

Expansion joints are selected in **Expansion Joint Type (if any)** on the **Tubesheet Type and Design Code** tab. Two types are available:

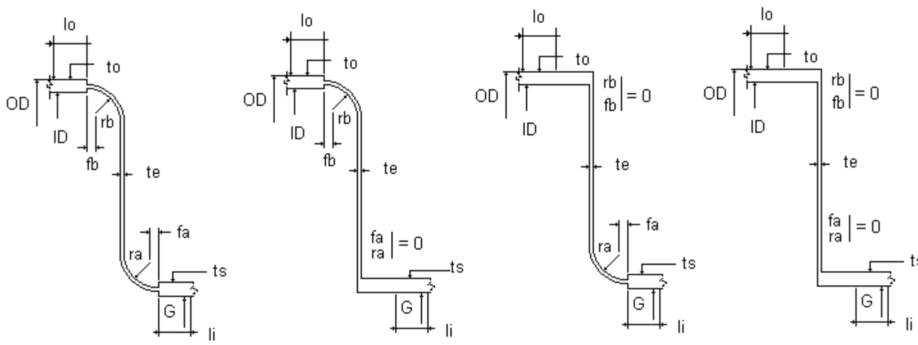
Thin Joint is comprised of a thin bellows that is very flexible and has a low stiffness. Below are examples designed according to ASME Appendix 26:



Thick Joint or *Flanged/Flued Expansion Joint* is comprised of a number of shell elements that are added together to form the joint. A thick joint is stiffer than the thin joint, and its stiffness must be taken into account.

NOTE If you have also installed the Paulin Research Group third party software, NozzlePRO™, you can also perform finite element analysis (FEA) on some thick expansion joint properties. For more information, see **Finite Element Analysis (FEA) on ASME or TEMA Expansion Joints (PV Elite) (page 172).**

Below are typical combinations of flexible shell elements for a thick expansion joint, showing one-half of a convolution and using TEMA nomenclature:

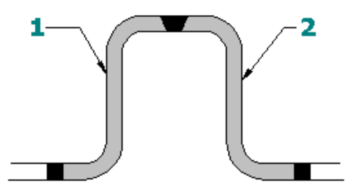


Perform App. 26 Stress and Life Cycle Calculations for the Bellows?

Select if **Thin Bellows Type** is selected for **Expansion Joint Type (if any)** on the **Tubesheet Type and Design Code** tab. Then click **>>** to define the thin joint according to ASME Appendix 26 analysis. For more information, see **Thin Joint Options** (page 166).

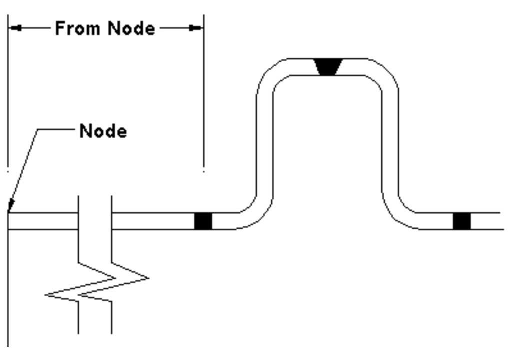
Number of Flexible Shell Elements

Enter the number of flexible shell elements for a thick joint. One convolution has two flexible shell elements:



Dist. from "From" Node

Enter the axial or longitudinal distance from the **From Node** to the start of the expansion joint:



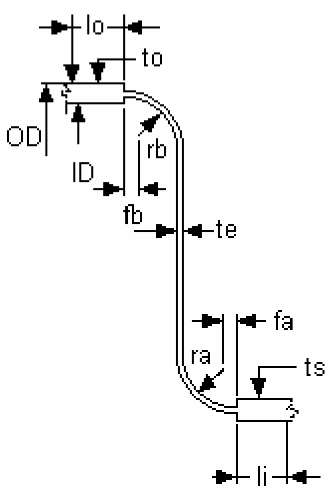
Design Option

Select **Existing** to enter the stiffness characteristics of the expansion joint. Use this option if you do not want to analyze the expansion joint itself but are just specifying its spring rate to be used in the tubesheet calculations. For example, you are purchasing the expansion joint from the manufacturer, who has already analyzed the joint. Select **Analyze** to calculate the stiffness characteristics (spring rate) and the stresses in the joint. You must also select **Analyze** to perform finite element analysis on an expansion joint.

Set Defaults

Click to set the dimension starting points for the expansion joint design.

Enter values for the following options to define thick joint characteristics:



Expansion Joint ID

Expansion Joint OD

Wall thickness (te)

Corrosion Allowance

Knuckle Offset Dimension Inside (fa)

Knuckle Offset Dimension Outside (fb)

Knuckle Radius Inside (ra)

Knuckle Radius Outside (rb)

Shell Cylinder Length (Li)

Desired Cycle Life

Enter the number of cycles needed for the life of the expansion joint.

User Input Spring Rate Corroded

Enter a value for thin joints and for thick joints when **Existing** is selected for **Design Option**.

User Input Spring Rate Uncorroded

Enter a value for thin joints and thick joints when **Existing** is selected for **Design Option**.

Thick Expansion Joint Calculation Method

Select **TEMA, Kopp and Sayre** or **FEA**.

Select **FEA** to perform finite element analysis using the third party NozzlePRO software on your expansion joint. When you select **FEA**, the software activates several inputs for the analysis.

NOTES

- You must have installed the third-party NozzlePRO software (from Paulin Research Group) before running PV Elite.
- The software displays the **FEA** option in the **Thick Expansion Joint Calculation Method** box when you have selected a thick expansion joint using a fixed tubesheet

type. For more information on the process for FEA, see **Finite Element Analysis (FEA) on ASME or TEMA Expansion Joints (PV Elite) (page 172).**

Elastic Modulus optional (computed if zero)

Type a value that is in the range of 1.5e7 to 3.2e7 psi (or 1.034e8 to 2.2e8 kPa). If you specify a value outside of this range, the software changes the value to zero.

When you select to perform finite element analysis on an expansion joint, the software activates this input and the values you specify here determine what stress calculations the software uses and displays on the **ASME TS Calc** or **TEMA TS Calc** output reports.

Poisson's Ratio

Type the Poisson's ratio value for the expansion joint material. For most steels, this value is close to 0.3. You can find Poisson's ratio values in material reference standards. When you select to perform a finite element analysis on an expansion joint, the software activates this input. The values you specify here determine what stress calculations the software uses and displays on the **ASME TS Calc** or **TEMA TS Calc** output reports.

FEA Runtime Options

Select an option that indicates the type of finite element analysis calculation performed for the expansion joint. The software activates this input and the values you specify here determine what stress calculations the software uses and displays on the **ASME TS Calc** or **TEMA TS Calc** output reports.

Perform ASME Combinations - Indicates that the software performs all seven of the Unfired Heat Exchanger (ASME) combination calculations, including:

- Displacements due to tubeside pressure
- Shellside pressure + shellside displacements
- Shellside pressure + shellside displacements + tubeside displacements (both pressures)
- Thermal displacements only
- Displacements due to tubeside pressure plus thermal displacements
- Shellside pressure + shellside displacements plus thermal displacements
- Shellside pressure + shellside displacements + tubeside displacements (both pressures) plus thermal displacements

Calculate Stiffnesses Only - Includes only stiffnesses.

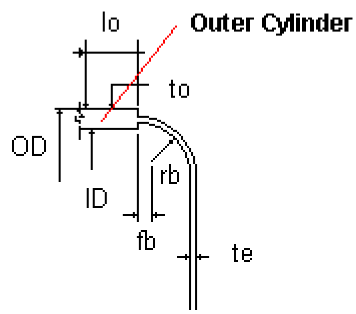
Calculate Thermal Displacements Only - Includes displacements.

Calculate Shellside Pressures Only - Includes displacements due to shellside pressure.

Calculate Tubeside Pressures Only - Includes displacements due to tubeside pressure.

Outer Cylinder Specification

Enter values for the following options to define outer cylinder characteristics:



Is there an Outer Cylinder?

Outer Cylinder Material

Outer Cylinder Thickness (to)

Outer Cylinder Corrosion Allowance

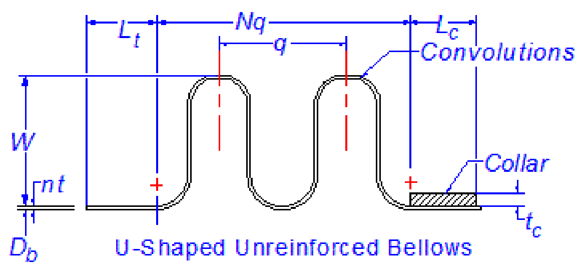
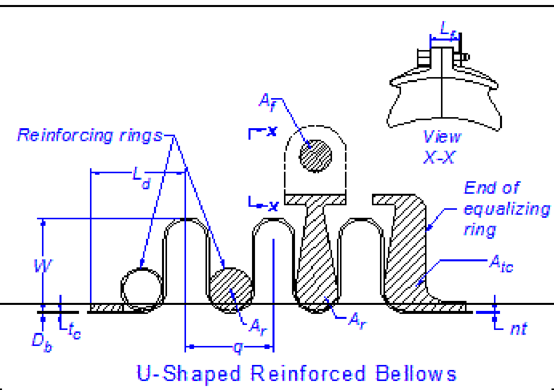
Outer Cylinder Length (lo)

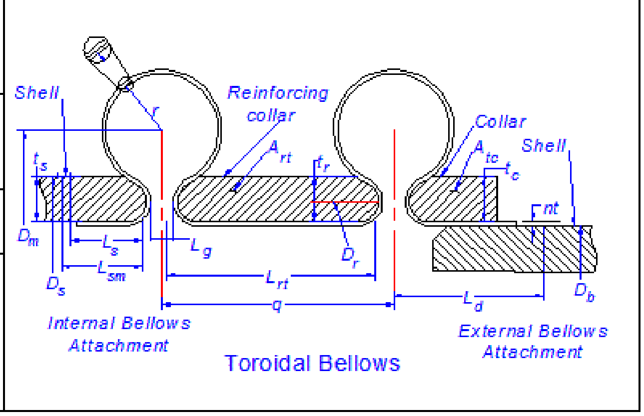
Outer Cylinder Design Temperature

Thin Joint Options

Thin expansion joints are analyzed according to ASME Section VIII, Division 1, Appendix 26 of the Boiler and Pressure Vessel Code. Life cycle and stress analysis is performed.

Enter values for the dimensions corresponding to your joint shape:

Dimension	Description	Sketch
A_f	Cross sectional metal area of one reinforcing fastener	
D_b	Inside diameter of bellows convolution	
D_m	Mean diameter of bellows convolution	
D_r	Cross sectional diameter of the reinforcing ring	
K_f	Forming method factor. Use 1.0 for expanding mandrel or roll forming. Use 0.6 for hydraulic, elastomeric, or pneumatic tube forming.	
L_c	Bellows collar length. For the toroidal bellows, *Lc* is determined by dividing the collar cross section area with the collar thickness.	
L_d	Length from attachment weld to the center of the first convolution for	

	externally attached bellows	
L_f	Effective length of one reinforcing fastener	
L_g	Maximum distance across the inside opening of a toroidal convolution, considering all movements	
L_r	Effective reinforcing collar length, calculated by $\frac{1}{3}\sqrt{D_r t_r}$	
L_{rt}	Overall length of reinforcing collar	
L_s	Effective shell strength, calculated by $\frac{1}{3}\sqrt{(D_s + t_s)t_s}$	
L_{sm}	Minimum required shell length having thickness, *t*	
L_t	End tangent length	
Lw	Distance between toroidal bellows attachment welds	
q	Convolution pitch	
r	Mean radius of toroidal bellows convolution	
t	Bellow nominal thickness of one ply	
tc	Collar thickness	
w	Convolution depth	

delta Q

Enter the differential thermal expansion between the tubes and shell divided by the number of convolutions. If this value is **0**, it is calculated during the analysis. If you are evaluating this joint by clicking **Quick Results**, you need to calculate delta Q and enter that value. In this case, resetting the value to **0** before exiting the dialog box is recommended. A non-zero positive value will be used by the software regardless of what was actually computed for **delta Q**. This entry is optional.

Load Cases Tab (Heat Exchanger Tubesheet Input Dialog Box)

Specifying load cases in addition to the design case enables PV Elite to perform calculations for different combinations of pressures and temperatures, ensuring that all the possible conditions are considered. Examples of additional cases are: shut-down, start-up, and upset. For each load case, values can be entered as needed for:

- Shell - The shell between the tubesheets.
- Channel - The channel at the ends of the exchanger.
- Tubes - The exchanger tubes.
- Tubesheet - The tubesheet. For fixed and floating tubesheets, it is one of the pair.
- Shell Band - The thicker shell courses at either end of the main shell next to tubesheet.

NOTE Enter **0** for any options that are not needed.

Number of cases to process

Select the number of load cases to run. You can specify up to eight different load cases.

Active Load Case

Click the up and down arrows to cycle through the load cases.

For each active load case, enter values as needed for the following:

Case Description

Enter a description.

Minimum Pressures and Report Options for this Load Case

Click to open the **Report Print Options Dialog Box** (page 170).

Maximum Design Pressure

Enter the maximum design pressures for the shell side and for the channel side.

Maximum Operating Pressure for ASME

For an ASME exchanger, enter the maximum operating pressures for the shell side and channel side.

NOTE These values are used to determine Ps and Pt.

Psox, max = max. (0, maximum shell side operating pressure for operating condition *x)*

Ptox, max = max. (0, maximum tube side operating pressure for operating condition *x*)

Minimum Operating Pressure for ASME

For an ASME exchanger, enter the minimum operating pressures for the shell side and channel side.

NOTE These values are used to determine Ps and Pt.

Psox, min = min. (0, minimum shell side operating pressure for operating condition *x*)

Ptox, min = min. (0, minimum tube side operating pressure for operating condition *x*)

Design Temperature

Enter the design temperatures for the shell, channel, tubes, and tubesheet.

Use Operating Metal Temperatures (ASME)

For ASME, select, and then enter the operating temperatures for the shell, channel, tubes, and tubesheet. UHX-13.4(b) Elastic moduli, yield strengths, and allowable stresses are taken at design temperatures. For cases involving thermal loading (loading cases 4, 5, 6, and 7), you can use the operating temperatures instead of the design temperatures (see UG-20).

Material

Enter the name of the material for tubes, tubesheet and, shell bands. This software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. Alternatively, use **Database Lookup and Properties.**

Mean Metal Temperature Along Length

Enter the actual metal temperature for the shell and tubes.

Metal Temperature at Tubesheet Rim

Enter the actual temperature of metal at the outer rim of the tubesheet, where the tubes are in the outer rows, for the shell, channel, and tubesheet.

Database Lookup and Properties

Click **Tubes**, **Tubesheet**, and **Shell Band** to select a material for each directly from the **Material Database Dialog Box** (page 547). The selection is then entered into **Material**. To modify the material properties, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties for this analysis. It does not modify the database.

Modulus of Elasticity

User-Defined Values

Select to enter your own modulus of elasticity values, overriding the material properties from the design code selected for analyzing the exchanger.

Modulus at Temperature

Enter the modulus of elasticity at the design temperature.

Modulus at Mean Metal Temp Along Length

Enter the modulus of elasticity at the actual metal temperature.

Modulus at Mean Metal Temperature

Enter the modulus of elasticity for the actual metal temperature at the channel.

Modulus at Ambient Temperature

Enter the modulus of elasticity at ambient temperature.

Coefficient of Thermal Expansion (alpha values)

User-Defined Values

Select to enter your own coefficient of thermal expansion values, overriding the material properties from the design code selected for analyzing the exchanger.

Alpha at Mean Metal Temp Along Length

Enter the actual metal temperature for the shell, tubes, and shell band.

Alpha at Metal Temp at Tubesheet Rim

Enter the actual temperature of metal at the outer rim of the tubesheet where the tubes are in the outer rows.

Differential Pressure Design?

Select to perform the analysis only for the difference between the shell side pressure and the tube side pressure.

Differential Design Pressure

Enter the differential pressure when **Differential Pressure Design?** is selected. If you do not enter a value, the software uses the effective pressures dictated by the code. This entry is optional.

Is the Exchanger Operating in the Creep Range (skip EP, use 3S for Sps)?

Select if the exchanger is operating in high-temperature creep range. This option only applies if the method used to analyze the heat exchanger is ASME.

Exchanger subject to cyclic or dynamic reactions due to pressure or thermal variations? (see UHX-13.8)

Select to calculate for the effect of radial thermal expansion adjacent to the tubesheet, according to UHX 13.8 (fixed tubesheet heat exchangers) or UHX 14.6 (floating tubesheet heat exchangers).

Expansion Joint Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Report Print Options Dialog Box

Sets options for analysis result reports of each load case.

Select the following as needed:

- **For ASME fixed/stat. tubesheets, print intermediate results in tabular form**
- **Print intermediate results for expansion joint calculations**
- **For TEMA and PD 5500 run multiple load cases** - Runs the following load cases for fixed tubesheets:

Load Case #	Load Case Description	
	Corroded	**Uncorroded**
1	Fvs + Pt - Th + Ca	Fvs + Pt - Th - Ca
2	Ps + Fvt - Th + Ca	Ps + Fvt - Th - Ca
3	Ps + Pt - Th + Ca	Ps + Pt - Th - Ca
4	Fvs + Fvt + Th + Ca	Fvs + Fvt + Th - Ca
5	Fvs + Pt + Th + Ca	Fvs + Pt + Th - Ca
6	Ps + Fvt + Th + Ca	Ps + Fvt + Th - Ca
7	Ps + Pt + Th + Ca	Ps + Pt + Th - Ca
8	Fvs + Fvt - Th + Ca	Fvs + Fvt - Th - Ca

Shell-side / Channel-side Vacuum Pressures

Enter positive values. For example, for full atmospheric vacuum condition enter a value of 15.0 psig. If no value is entered, then 0 psi is used.

Set detailed printout options for the currently selected load case

When you want to see detailed equations and intermediate calculations in the results, select the corroded and uncorroded load cases. For all other load cases, the software generates summarized, tabular results.

For ASME tubesheets, the following load cases are run for fixed and floating tubesheet exchangers:

Load Case #	Load Case Description	
	Corroded	**Uncorroded**
1	Fvs + Pt - Th + Ca	Fvs + Pt - Th - Ca
2	Ps + Fvt - Th + Ca	Ps + Fvt - Th - Ca
3	Ps + Pt - Th + Ca	Ps + Pt - Th - Ca
4	Fvs + Fvt + Th + Ca	Fvs + Fvt + Th - Ca
5	Fvs + Pt + Th + Ca	Fvs + Pt + Th - Ca
6	Ps + Fvt + Th + Ca	Ps + Fvt + Th - Ca
7	Ps + Pt + Th + Ca	Ps + Pt + Th - Ca

For ASME stationary tubesheet configuration "d" and ASME floating tubesheet configurations "B", "C" and "D", the design is based only on load cases 1, 2 and 3.

The following load cases are performed for ASME U-tube tubesheet exchangers:

Load Case #	Load Case Description	
	Corroded	**Uncorroded**
1	Fvs + Pt - Th + Ca	Fvs + Pt - Th - Ca
2	Ps + Fvt - Th + Ca	Ps + Fvt - Th - Ca
3	Ps + Pt - Th + Ca	Ps + Pt - Th - Ca

For all ASME exchangers, if vacuum pressures are specified, then an additional load case is run:

Load Case #	Load Case Description	
	Corroded	**Uncorroded**
8	Fvs + Fvt - Th + Ca	Fvs + Fvt - Th - Ca

Additionally, if **Differential Pressure Design?** is selected on the **Load Cases** tab, then only certain load cases are run.

NOTES

- Fvt, Fvs - User-defined Shell-side and Tube-side vacuum pressures or 0.0.
- Ps, Pt - Shell-side and Tube-side Design Pressures.
- Th - With or Without Thermal Expansion.
- Ca - With or Without Corrosion Allowance.

Finite Element Analysis (FEA) on ASME or TEMA Expansion Joints (PV Elite)

Using the PV Elite interface with Paulin Research Group's NozzlePRO software, you can perform FEA while designing thick expansion joints from within PV Elite. FEA lets you calculate more precise date for certain properties of expansion joints.

Complete the following steps to perform finite element analysis (FEA) on thick expansion joints in PV Elite. You can perform FEA on expansion joints for ASME, TEMA, EN-13445, and PD 5500 tubesheets.

NOTE You must have installed NozzlePRO from Paulin Research Group to perform FEA on expansion joints in PV Elite.

1. Open PV Elite .
2. Open an existing or create a new heat exchanger job in PV Elite.

   NOTE From the **Home** tab, you can select **New** and specify a file type for your new file (such as **ASME Section VII-Division 1** or **British Standard PD 5500**). The software opens a new file with many of the inputs already pre-set for the type of file you selected.

3. Click **Tubesheet Analysis** from the **Details** panel on the **Home** tab.

   *The **Heat Exchanger Tubesheet Input** dialog box opens.*

4. Specify the **Tubesheet Analysis Method** and **Exchanger Type**, and then set the **Expansion Joint Type** to **Thick Type (Flanged and Flued)**.

5. On the **TubeSheet Properties** tab, select a fixed **Tubesheet Type** (such as **Fixed Tubesheets, gasketed both sides**).

6. On the **Expansion Joint Data** tab, specify the **Design Option** as **Analyze**.

7. Specify the **Thick Expansion Joint Calculation Method** as **FEA**.

   The software enables several inputs applicable for finite element analysis and disables inputs that are unnecessary to the expansion joint calculations.

8. Specify the **Elastic Modulus**, **Poisson's Ratio (for FEA)**, and **FEA Runtime Options**. The values you specify in these fields determine what stress calculations the software uses and displays on the **ASME TS Calc** or **TEMA TS Calc** output reports. The ASME and TEMA tubesheet modules use TEMA 9th edition standard for thick expansion joint FEA calculations.

9. Click **Analyze**. After the analysis completes, refer to the **Expansion Joint Stress Summary (PRG)** section on the output report.

   *The **Expansion Joint Stress Summary (PRG)** appears at the bottom of the **ASME TS Calc** or **TEMA TS Calc** reports. The **FEA Runtime Option** you select determines which results appear in the stress summary of the reports.*

Floating TubeSheet Tab (Heat Exchanger Tubesheet Input Dialog Box)

Description

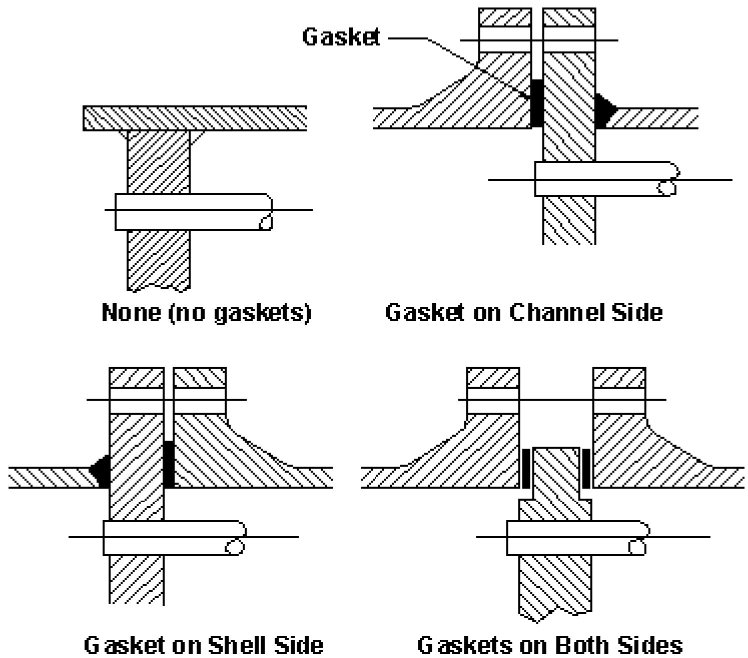
Enter the description for the floating tubesheet. The description appears on the final report.

Floating Tubesheet Type

Select the type of floating tubesheet to analyze, according to ASME:

- **(a) Floating tubesheet, integral**
- **(b) Floating tubesheet, gasketed, extended as flange**
- **(c) Floating tubesheet, gasketed, no extended, w/backing device**

- **(d) Floating tubesheet, internal sealed**



Outside Diameter

Enter the outside diameter of the tubesheet.

Tubesheet Thickness

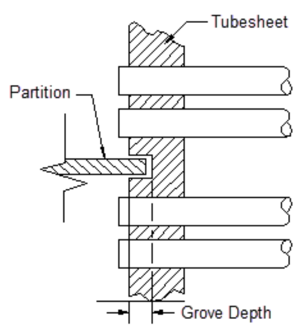
Enter the tubesheet thickness in uncorroded condition. If it is a re-rate, then the actual measured thickness is typically used.

Corr. Allow. Shell Side / Channel Side

Enter the corrosion allowance on the shell side (the inner face of the tubesheet), and the corrosion allowance on the channel side (the outer face of the tubesheet facing the channel side).

Depth of Groove in Tubesheet (if any)

Enter the depth of a groove in the tubesheet, used to locate the channel partition plate and its gasket. If there is no groove for example in a single pass exchanger, this value is **0**.

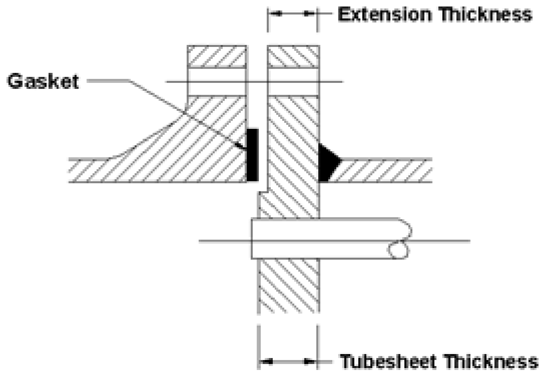


Tubesheet Extended as Flange?

Select if the tubesheet is extended as a flange, so that it is subject to the bolt load from the mating flange.

Thickness of Extended Portion

When **Tubesheet Extended as Flange?** is selected, enter the thickness of the portion of the tubesheet that is extended for bolting.



Bolt Load Transferred to Tubesheet?

When **Tubesheet Extended as Flange?** is selected, also select this option if the bolt load is transferred to the tubesheet, extended as the flange. Do not select if the tubesheet is gasketed with both the shell and channel flanges. Otherwise, the tubesheet can still be extended, but the bolt load is not transferred to the tubesheet extension. Carefully consider all possible cases, such the hydrotest. When this option is not selected, the required thickness of tubesheet extension is not calculated.

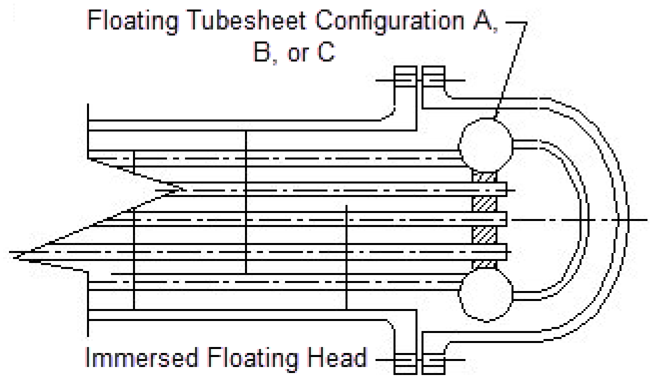
Integral Channel Properties for ASME Floating Configuration A

These properties are needed for floating heat exchangers with the floating tubesheet welded to head.

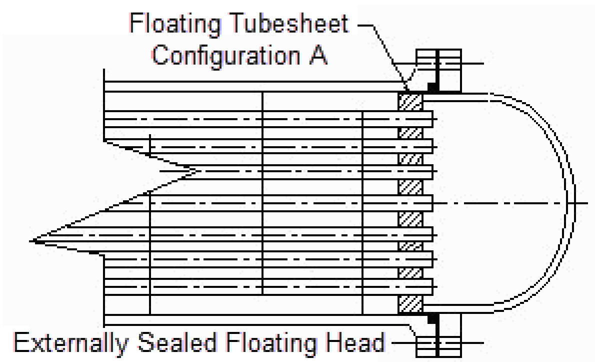
- **Channel Thickness tc**
- **Channel Design Temperature**
- **Channel Material** - Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

In ASME nomenclature this is Configuration A:

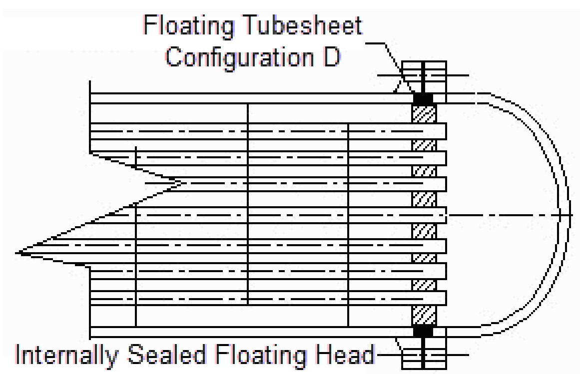
Exchanger with an Immersed Floating Head:



Exchanger with an Externally Sealed Floating Head:



Exchanger with an Internally Sealed Floating Head:



Spherical Cover/Backing Ring Tab (Heat Exchanger Tubesheet Input Dialog Box)

Description

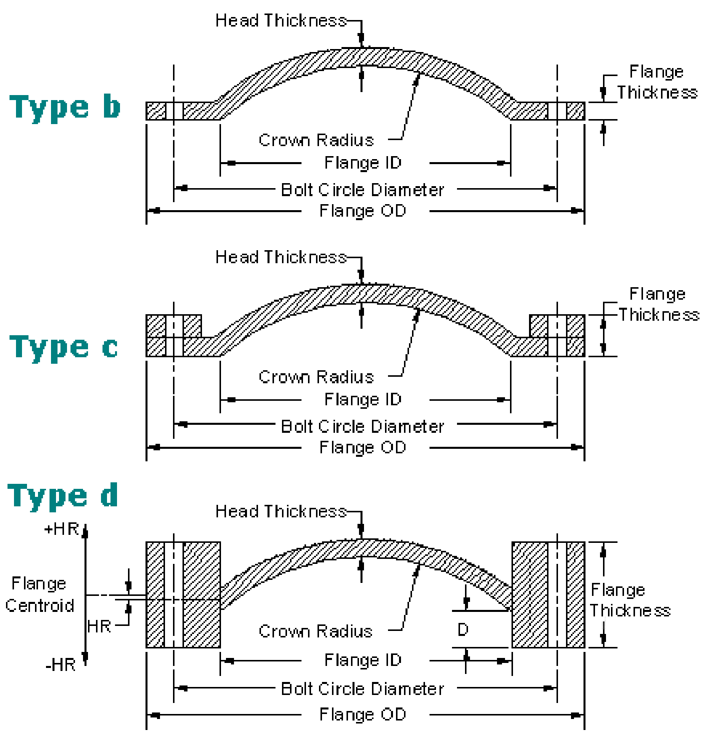
Enter the description for the spherical cover. The description appears on your final report.

Design Temperature

Enter the design temperature for each head. This temperature is used to interpolate the material allowable tables and external pressure curves.

Type of Floating Head

Select the type of floating head or spherically dished cover, corresponding to Figure 1-6 of ASME, Section VIII, Division 1, Appendix 1:



b - Solid thick head, spherically dished.

c - Thin dished head, continuous across flange face.

d - Spherical cap welded to flange ID.

None

NOTE **d** is the most common type of head used for heat exchanger floating heads.

NOTE Click ... to open the **Flange and Gasket Information** dialog box. Enter flange, gasket, and bolt data for the select floating head type.

Flange Thickness (tf)

Enter the through thickness of the flange. For type **c** spherical caps, this includes the thickness of the head.

Inside Crown Radius (L)

Enter the inside crown radius, usually roughly equal to the flange ID. This value may be any dimension greater than the inside radius of the flange.

Head Thickness (t)

Enter the minimum thickness of the actual plate used to build the floating head or spherical cap, or the minimum thickness measured for an existing floating head or spherical cap.

Head Internal Corrosion Allowance

Enter the corrosion allowance on the concave side of the head. The software adjusts the thickness and the diameter for the evaluation of allowable pressure. The allowance is also added to the required thicknesses. Some common corrosion allowances are:

- 0.0625 - 1/16"
- 0.1250 - 1/8"
- 0.2500 - 1/4"

Head External Corrosion Allowance

The software adjusts the thickness and the diameter for the evaluation of allowable pressure. The allowance is also added to the required thicknesses.

Slotted Flange?

Select if the flange has slotted bolt holes for quick opening. A slotted flange has bolt holes extending radially to the outer edge of the flange. The software automatically adjusts for this condition; you do not have to change the flange outside diameter.

Full Face Gasket Option

Select **Program Selects**, **Full Face Gasket**, or **Not a Full Face**. A full face gasket extends from the ID to the OD of the flange, enclosing the bolt holes. These gaskets are usually soft materials, such as rubber or an elastomer, so that the bolt stresses do not go too high during gasket seating. The software adjusts the flange analysis and the design formulas to account for the full face gasket.

Head Material Name

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Flange Material Name

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

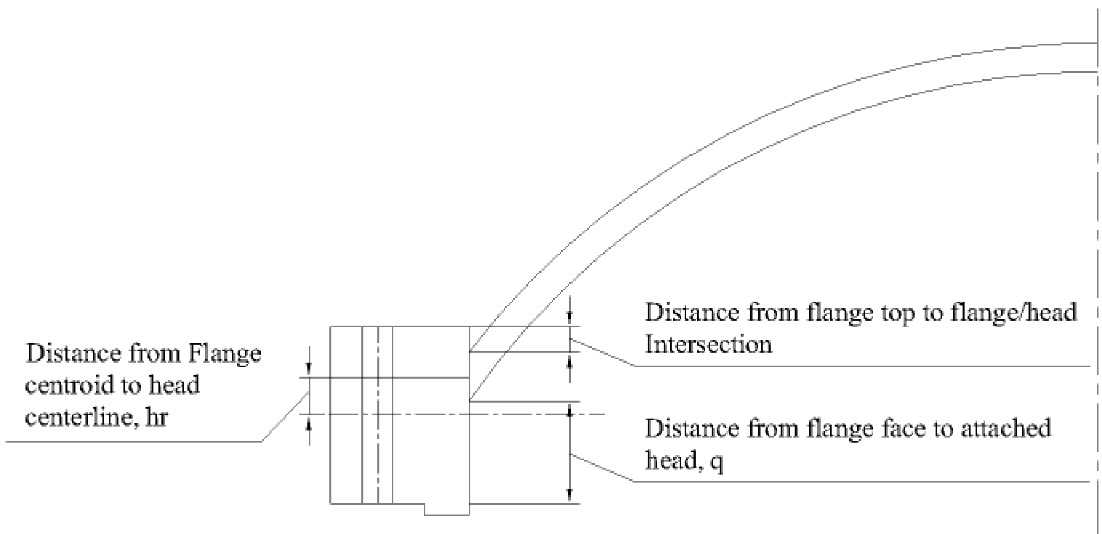
Floating Head Factors (Spherical Cover/Backing Ring Tab)

Compute "F" even if the pressure is 0

Select to calculate the factor *F* in the floating head even when the internal pressure is zero. *F* is a direct function of the internal pressure. If the internal pressure is 0, then *F* is equal to 0. However, a conservative interpretation of the code always calculates *F* regardless of the case being analyzed. Typically, the flange-bolt-up case is in question because there is no internal pressure when bolting up the unit, so this option is not selected. Select this option only for a conservative calculation.

Distance from Flange Centroid to Head Centerline (hr)

Enter the distance *hr* from the flange centroid to the intersection of the head centerline and the flange. Enter the value in the corroded condition. The value is positive if it is above the flange centroid, and negative if it is below the flange centroid. This distance is used in the code calculation but not when **Perform Soehren's Calc** is selected (where *Q* is used). See the illustration below.



Distance from Flange Top to Flange/Head Intersection

Enter the distance from the top of the floating head flange to the intersection of the dished head and the flange in the uncorroded condition, and then click **Compute**. The software considers the corrosion allowance, calculates *hr* in the corroded condition, and places its value in **Distance from Flange Centroid to Head Centerline (hr)**. Other than for this purpose, this value is not used by the software.

Compute

If **Distance from Flange Top to Flange/Head Intersection** is known, you can enter it (in the uncorroded condition) and click **Compute**. The software considers the corrosion allowance, calculates *hr* in the corroded condition, and places its value in **Distance from Flange Centroid to Head Centerline (hr)**.

Perform Soehren's Calc

Select to perform Soehren's calculation, a more detailed analysis of the interaction between the spherical cap and the flange. The stresses calculated are frequently acceptable for heads or flanges that are slightly less thick than required by the normal code rules. This analysis can only be done for type **d** floating heads. Par. 1-6(h) of the code allows this type of analysis.

Dim Q

Enter the distance *Q* from the flange bolting face to the intersection of the attached head inside diameter and the flange. *Q* is used in the Soehren's calculation, but not in the code calculation (where *hr* is used). See the illustration above.

Backing Ring Data (Spherical Cover/Backing Ring Tab)

Is There a Backing Ring?

Select if a backing ring is used. A backing ring is a second flange used to sandwich the tubesheet of a floating head heat exchanger.

Backing Ring Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Backing Ring Inside Diameter

Enter the inside diameter of the backing ring. This value is usually slightly smaller than the outside diameter (OD) of the flange.

Backing Ring Outside Diameter

Enter the outside diameter of the backing ring. This value is usually slightly larger than the inside diameter (ID) of the flange.

Backing Ring Thickness

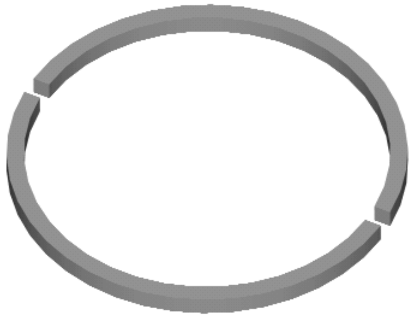
Enter the actual through thickness of the backing ring. For doubly split rings (when **2** is selected for **Number of Splits in Backing Ring)**, this is the thickness of each piece.

Number of Splits in Backing Ring

Select the number of splits in the ring, if any, for loose-type flanges. Select **0**, **1**, or **2**. Split flanges are typically ring-type flanges. A split is used when the flange must be completely removable from the vessel.

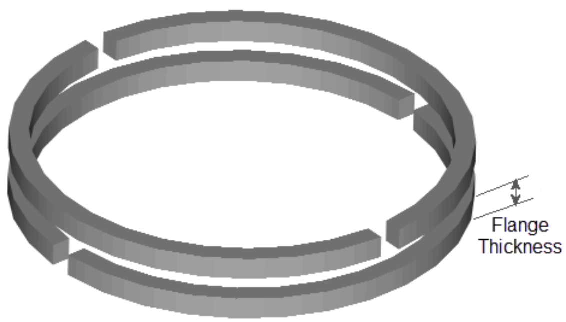
- Ring with a single split:

When the flange is split into two pieces by a single split, the split is along the diameter, and the design bending moment for the flange is multiplied by 2.0.



- Ring with a double split:

  A ring with two splits has two stacked rings, with each half split along the diameter. The software analyzes each ring as if it were a solid flange (without splits) using 0.75 times the design bending moment. **Backing Ring Thickness** is the thickness of each piece. The thickness of the total ring is twice this value. The pair of rings is assembled so that the splits in one ring are at 90º from the splits in the other ring.



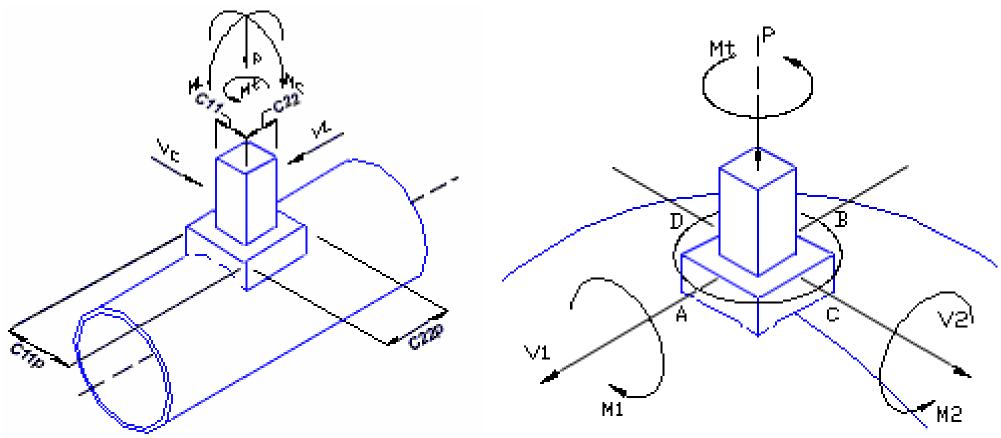
NOTE For TEMA vessels, backing ring equations are found in Section RCB 5.141 of the TEMA standard.

Add a Clip

Home tab: **Details** > **Add a Clip**

Adds a clip to the vessel. In the dialog, enter clip support information on cylinders as well as elliptical, torispherical and spherical heads. Clips are used to carry load, such as from piping, ladders, and platforms. These loads, along with pressures, cause local stress at the clip support location. WRC 107/537 is used to calculate the local stresses and compare them to the allowables.

Sustained, expansion, and occasional loads on the clip must be determined from a separate analysis and entered. Clips supporting piping generally have loads in all categories, while platform clips generally have only sustained and occasional loads.



From Node

Displays the **From Node** for the selected element.

Description of Clip

Enter an alpha-numeric string to identify the detail. This must be a unique value compared to all other detail descriptions on the vessel. A consistent naming convention is recommended. For example, use the **From Node** number with an alphabetical extension showing the detail type and the number of details, such as the following descriptions when 10 is the **From Node**: 10 NOZ A, 10 INS, 10 RIN 1 of 12, or 10 Saddle A.

Distance from "From" Node

Enter the distance from the **From Node** to the center of the clip.

Layout Angle

Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the clip at the point where it is attached to the shell, following the same conventions used for a nozzle. For more information, see **Layout Angle**.

Is the Clip Circular?

Select if the clip has a circular cross-section.

Clip Parameters

Circumferential Length (C11)

Enter the length that the clip extends around the circumference of the cylinder element. For spherical head elements that can be analyzed using this method, the clip must be square in cross-section.

Longitudinal Length (C22)

Enter the length that the clip extends along the length of the cylinder element. For spherical head elements that can be analyzed using this method, the clip must be square in cross-section.

Thickness

Enter the distance that the pad extends radially from the shell surface or reinforcing pad. The thickness is not used to calculate stresses, but is used to calculate the weight and draw the 3D image.

NOTE When **Is the Clip Circular?** is selected, the clip parameters are **Clip Outside Diameter**, **Clip Outside Projection**, and **Wall Thickness**.

Pad Parameters

Reinforcing pad used?

Select if there is a pad under the clip.

Circumferential Length (C11p)

Enter the length that the pad extends around the circumference of the cylinder element. For spherical head elements that can be analyzed using this method, the pad must be square in cross-section.

Longitudinal Length (C22p)

Enter the length that the pad extends along the length of the cylinder element. For spherical head elements that can be analyzed using this method, the pad must be square in cross-section.

Thickness

Enter the distance that the pad extends radially from the shell surface. The thickness is not used to calculate stresses, but is used to calculate the weight and draw the 3D image.

NOTE When **Is the Clip Circular?** is selected, the pad parameters are **Pad Diameter** and **Thickness**.

Fatigue Parameters

Use Kn and Kb

Select to use the Kn and Kb parameters, based on the fillet radius for the clip or pad and used for fatigue analysis.

Clip Fillet Radius

Enter the clip fillet weld radius. See Appendix B in the WRC 107/537 bulletin for illustrations and more information.

Pad Fillet radius

Enter the pad fillet weld radius. See Appendix B in the WRC 107/537 bulletin for illustrations and more information.

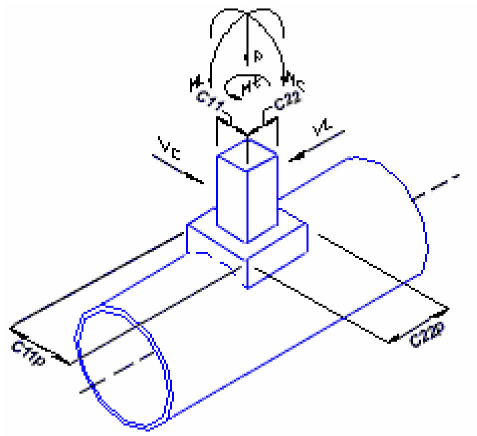
Local Attachment Loads at the Shell Surface

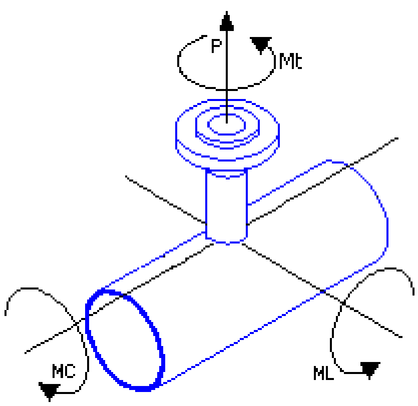
You can enter values in the following load sets:

- **Sustained** - (SUS) Primary loads, typically weight + pressure + forces.
- **Expansion** - (EXP) Secondary, self-limiting thermal expansion loads.
- **Occasional** - (OCC) Irregularly occurring, short-term loads, such as wind loads, seismic loads, and water hammer.

These local loads are used to analyze the stresses at the base of the clip in the shell. They are not used by the software to produce a global bending moment over the entire vessel cross section (Use **Force and Moment** to do this). The positive orientation of the loads is shown below. These loads are generally calculated, given as defaults or calculated by a stress analysis program such as CAESAR II.

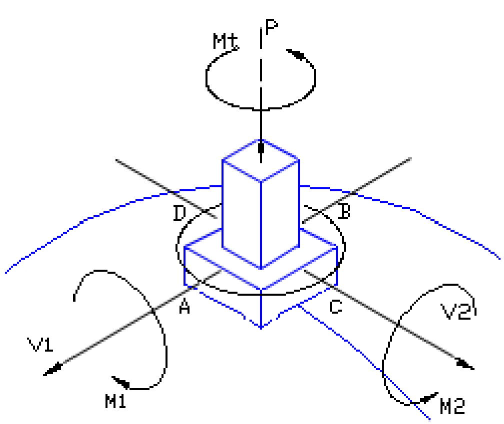
The following force/moment convention is used for a square or circular clip on a **cylindrical** element:



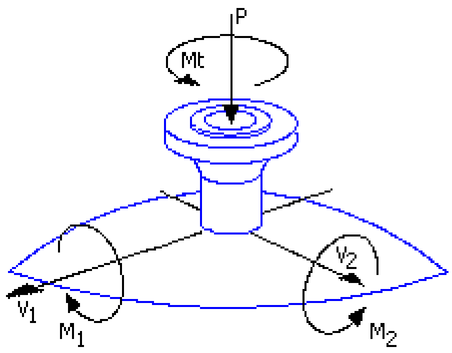


P - Radial Force
Vc - Circ. Shear Force
Vl - Long. Shear Force
Mc - Circ. Moment
Ml - Long. Moment
Mt - Torsional Moment

The following force/moment convention is used for a square or circular clip on a **spherical** vessel:



P - Radial Force
V2 - Shear Force, D to C
V1 - Shear Force, B to A
M1 - Moment, B axis
M2 - Moment, C axis
Mt - Torsional moment



Previous

If you created more than one clip on the element, click to go back to the previous clip.

Go To Next Clip

If you created more than one clip on the element, click to go to the next clip.

Add New Clip

Click to add a new clip to the element.

Delete

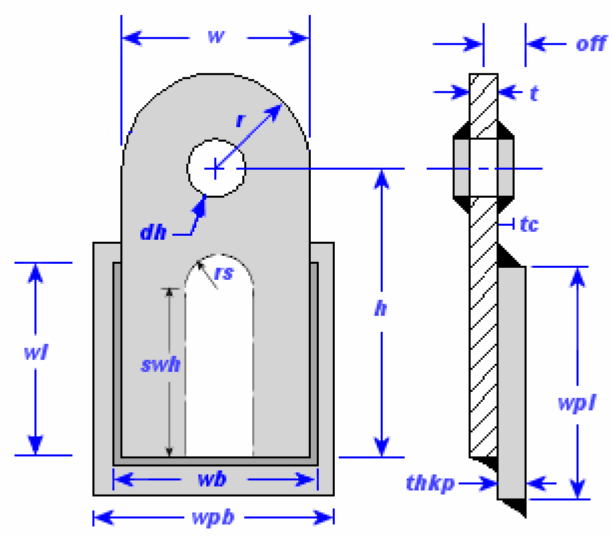
Deletes all data for the current clip.

Lifting Lug Data

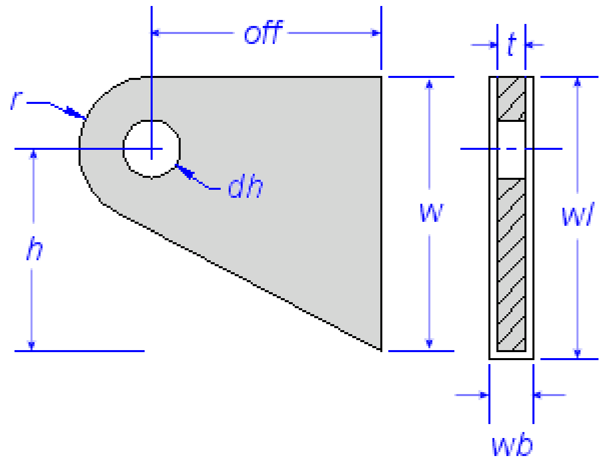
Home tab: **Details** > **Lifting Lug Data**

Adds lugs to the selected element. You can create:

- Flat-type lugs on vertical vessels, generally place near the top head.



- Perpendicular, ear-type lugs for horizontal vessels.



The software calculates the reactions on each lug, the stresses in each lug, and the stress in the welds.

Delete

Deletes all data for the lug.

Common Detail Parameters (page 54)

Layout Angle

Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the lug at the point where it is attached to the shell, following the same conventions used for a nozzle. For more information, see **Layout Angle** on the **Nozzle Orientation (Nozzle Main Tab)** (page 64).

Lug Contact Width (w)

For a flat lug, enter the width of the lug at the base. For a perpendicular lug, enter the longer length of the lug along the surface.

Diameter of Hole in Lug (dh)

Enter the diameter of the hole cut or drilled into most lifting lugs.

Radius of Semi-circular Arc (r)

Enter the radius of the semi-circular part of the lifting lug where the hole is located. This is typically circular on flat lugs and semi-circular on perpendicular lugs.

Height from Bottom to Center of Hole (h)

Enter the distance along the axis of the vessel from the center of hole to the bottom of the lug.

Offset from Vessel OD to Center of Hole (off)

Enter the distance from the center of the hole to base of the lifting lug. For perpendicular lugs this is to the vessel OD. If the orientation is flat this is 1/2 the thickness of the lug.

Lug Fillet Weld Size (tf)

Enter the fillet weld leg size. For stress analysis of the welds, the leg dimension is converted into the throat dimension.

Length of Weld Along Side of Lifting Lug (wl)

Enter the length of the long welds on the side of the lifting lug. The software multiplies this value by two when determining the weld area.

Length of Weld Along Bottom of Lifting Lug (wb)

Enter the length of the short weld. This is usually the bottom weld.

Collar Thickness (tc)

If the lug has a collar, enter the thickness. The thickness is measured from the outside surface of the lug to the edge of the collar.

Collar Diameter (dc)

Enter the diameter of the collar. This value is mainly used for documentation.

Lug Thickness (t)

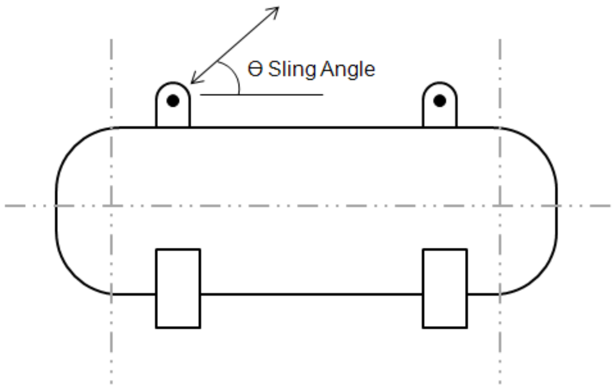
Enter thickness of the lifting lug plate.

Lug Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Sling Angle from Horizontal

Enter the sling angle in degrees:

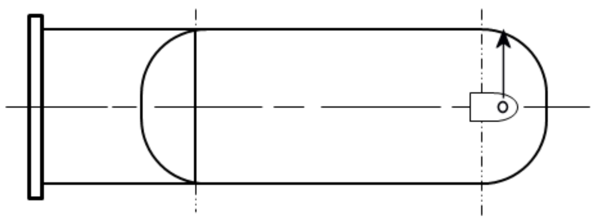


Occasional Load Factor

Enter an occasional load factor, used in many construction codes to increase the allowable stress for an event that is considered occasional in nature. Such occasional loads are wind, seismic, and the lifting of a vessel. The occasional load factor is multiplied by other terms in the allowable stress equation to get the overall allowable. If you do not want to use this value, enter **1**.

Impact Factor

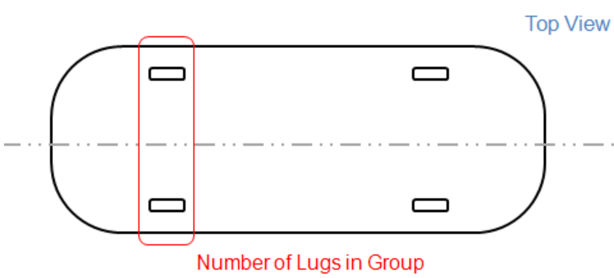
Enter the impact factor to account for lifting a vessel when it may be pulled quickly with sudden force. This value typically ranges from **1.5** to **2.0**, although values as high as 3.0 may be entered. The software multiplies the lifting loads by the impact factor. The theoretical maximum value for this value is 2.0.



Number of Lugs in Group

Enter the number of lugs in this group:

- For a horizontal vessel, there is typically one lifting lug on each side of the vessel. The number of lugs in a group is **1**. If there are two lugs on each end (such as for a large vessels), the value is **2**.

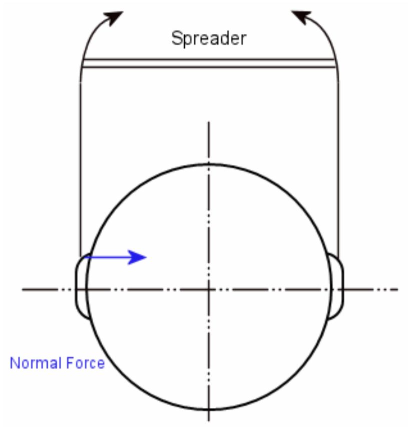


- For a vertical vessel, there are typically two flat lugs located near the top of the vessel. The number of lugs in a group is 2. A value of **1** for a vertical vessel should never be used.

Additional Normal Force (Fn) or **Tangential Force on Lug**

Enter an additional normal force acting on the lifting lug. This force is typically acting when there is no spreader bar used in the lifting procedure. For a horizontal vessel, enter the tangential force acting on the lug.

The normal or tangential force causes weak axis bending on the lug. Even a moderate force can cause a high bending stress. This is to be avoided. If there is an additional force, enter the value for each lug. The software applies the impact factor to this value.



Is there a pad?

Select if there is a pad under the lug.

Circumferential Width (wpb)

Enter the width of the pad along the circumference of the vessel.

Longitudinal Length (wpl)

Enter the length of the pad along the long axis of the vessel.

Pad Thickness (thkp)

Enter the thickness of the pad plate.

Pad Fillet Weld Size (tfp)

Enter the weld leg length for the pad fillet weld.

Is the lug slotted?

Select if the lifting lug is slotted.

Slot Radius (rs)

Enter the slot radius.

Slot Weld Height (swh)

Enter the slot weld height along the axis of the vessel from the center of slot hole to the bottom of the lug.

Jacket or Vapor/Distribution Belt

Home tab: **Details** > **Jacket or Vapor/Distribution Belt**

Adds ASME cylindrical jackets to the shell on the selected cylinder element, according to ASME Section VIII, Division 1, Appendix 9 or ASME Section VIII, Division 2, Part 4.11.

NOTE This command is only available when **Cylinder** is selected as the **Element Type.**

A Type 1 jacket is specified as either a jacket or a vapor belt. Vapor belts cover perforated areas of some vessels but are more typically found on shell and tube heat exchangers. Jackets can have separate design conditions from the parent cylindrical element. Vapor belts have the same design conditions as the parent cylindrical element. The software calculates the required thickness and MAWP for the jacket and closure bars. The MAWP of vapor belts can influence nozzle design as well as the overall MAWP of the vessel. The software calculates the stiffened length of the inner vessel to account for the closure bars.

Both jackets and vapor belts can have nozzles attached. If a nozzle has an internal projection that cuts into the parent shell, the nozzle reinforcement calculations are for the parent shell. If the nozzle does not have an internal projection, the calculation is for the jacket. According to Appendix 9/Part 4.11, nozzle calculations are not required for both the jacket and inner vessel.

The internal pressure in a vapor belt adds to the vacuum (external) pressure in the inner shell. This extra pressure is reflected in the external pressure report generated during analysis.

IMPORTANT

- Appendix 9/Part 4.11 jackets are joined by inter-connecting pipework, affecting static pressure head considerations.
- Enter the specific gravity of the fluid in the jacket or vapor belt. The software uses this value to calculate the weight of the fluid in the annular area.

Delete

Deletes all jacket data.

From Node

Displays the **From Node** for the selected element. The **From Node** is the software-generated node number describing the starting location of the element. The value cannot be modified.

Jacket Length L

Enter the length of the jacket along the cylinder element.

Dist. from 'From' Node to Jacket Bottom

Enter the distance from the From node to the bottom of the jacket.

Jacket Description

Enter a description for the jacket.

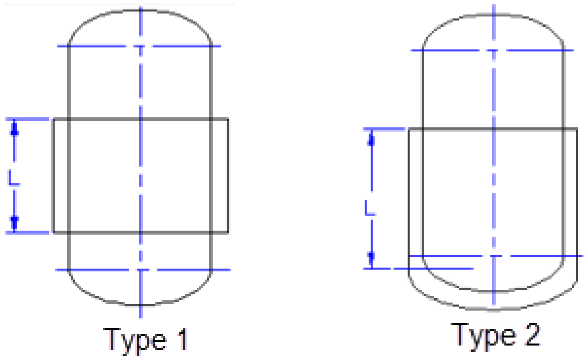
Is this a Vapor / Distribution Belt (cut outs in shell)?

Select if the jacket is a vapor belt or distribution belt.

Select Jacket Figure (Fig. 9-2)

Select either the **Type 1** or **Type 2** jacket configuration. Other jacket configurations in ASME

Section VIII Division 1, Appendix 9, Fig 9-2 or ASME Section VIII Division 2, Part 4, Fig 4.11.1 are not supported.



Jacket Longitudinal Efficiency

Enter the welded joint efficiency, as defined in ASME Section VIII, Division 1, Table UW-12. For the weld on a **Type 1** jacket (welded from both sides or with a removable backing strip), use the following efficiencies:

- Full radiography: 1.00
- Spot x-ray: 0.85
- No radiography: 0.70

Jacket Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Design Temperature, Internal

Enter the design temperature for internal pressure. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Design Temperature, External

Enter the design temperature for external pressure. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Jacket Pressure, Internal

Enter the design pressure for internal pressure analysis. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Jacket Pressure, External

Enter the design pressure for external pressure analysis. Available when **Is this a Vapor / Distribution Belt (cut outs in shell)?** is not selected.

Jacket Thickness

Enter the jacket thickness.

Jacket Corrosion Allowance

Enter the thickness of the corrosion allowance for the jacket.

Jacket Length for External Pressure

Enter the length of the jacket to use for external pressure calculations.

Jacket Inside Diameter

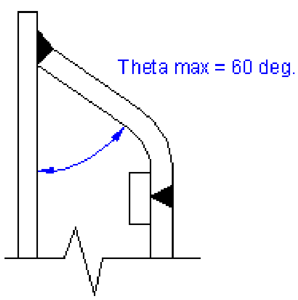
Enter the inside diameter of the jacket.

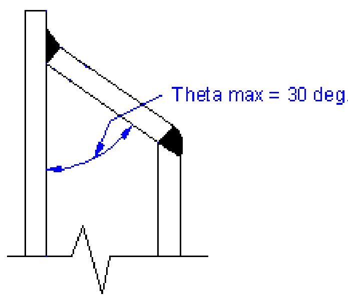
Specific Gravity of Contents

Enter the specific gravity of any fluid contained within the jacket. This value is usually **1.0**.

Half Apex Angle

When **Type (b-2), Detail 2(b)**, **Type (c), Detail 3**, **Type (k),** or **Detail 8** is selected for **Select Closure Figure Type**, enter the half apex angle θ_{max} (or theta max). For example:





For more information, see ASME Section VIII, Division 1, paragraph UG-33, Figure UG-33.1 for half apex angles on typical geometries.

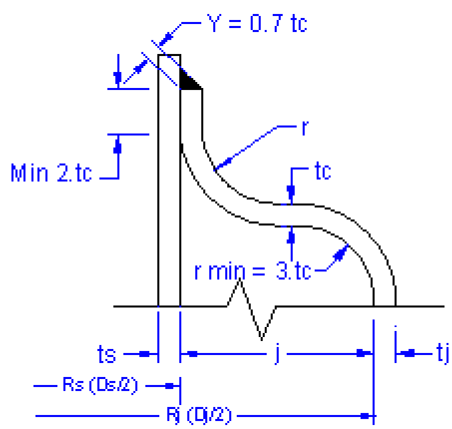
NOTE For most internal pressure calculations, the half apex angle should not be greater than 30º, though the software calculates results for up to 60º. For external pressure calculations, the angle must not be greater than 60º.

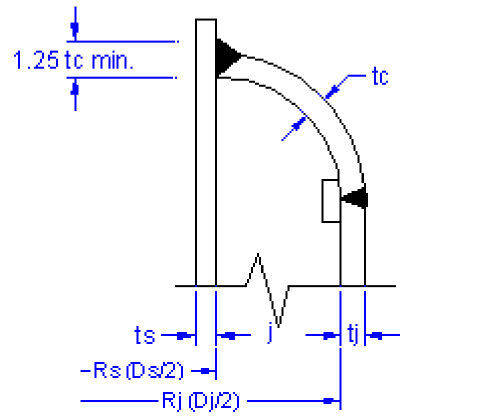
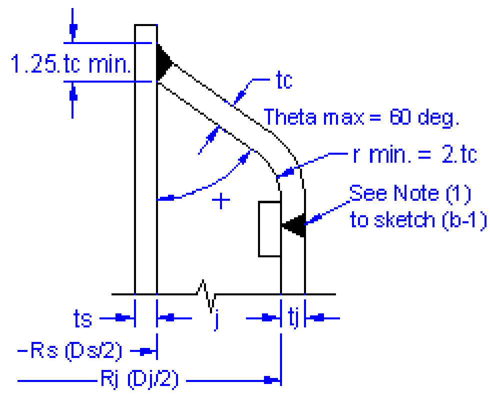
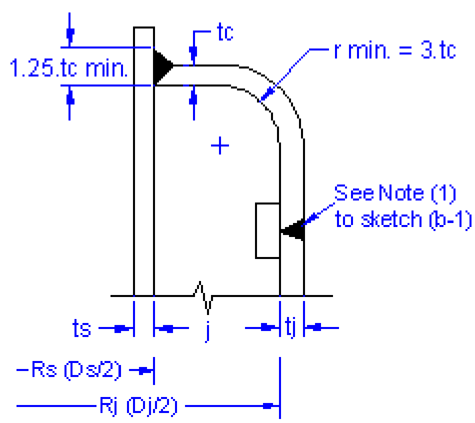
Select Closure Type (Fig. 9-5)

Select one of the following closure types, according to ASME Section VIII, Division 1, Appendix 9, Fig. 9-5 or ASME Section VIII, Division 2, Part 4, Table 4.11.1:

Div. 1 = Type (a)

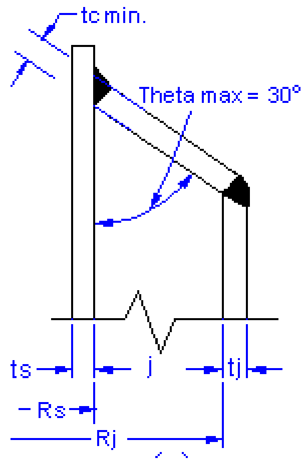
Div. 2 = Detail 1



Div. 1 = Type (b-1) Div. 2 = Detail 2 (a)	
Div. 1 = Type (b-2) Div. 2 = Detail 2 (b)	
Div. 1 = Type (b-3) Div. 2 = Detail 2 (c)	

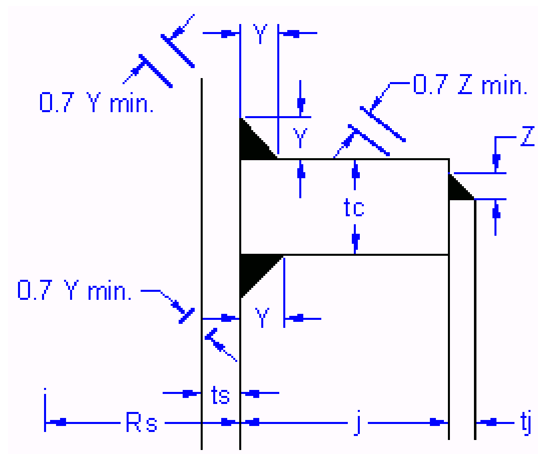
Div. 1 = Type (c)

Div. 2 = Detail 3



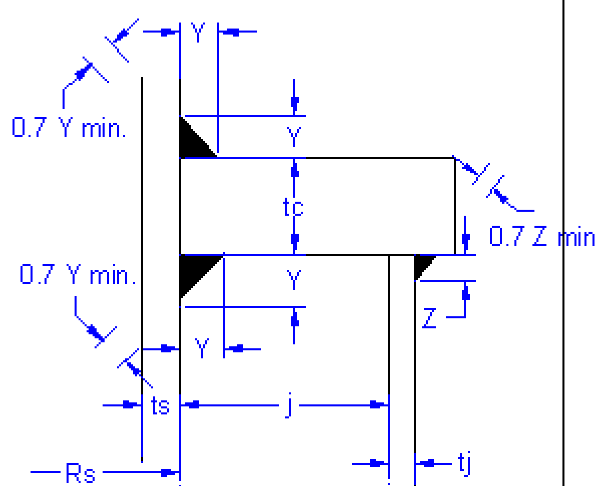
Div. 1 = Type (d-1)

Div. 2 = Detail 4 (a)



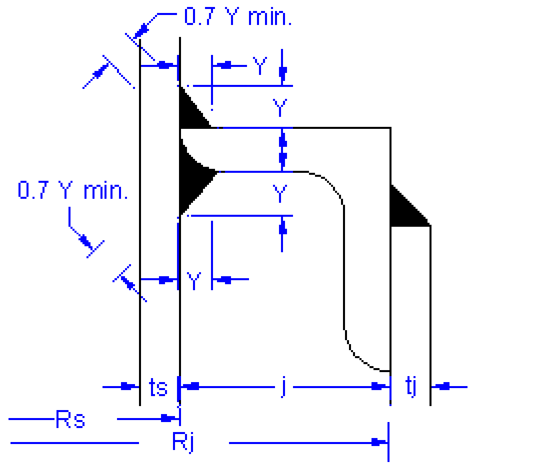
Div. 1 = Type (d-2)

Div. 2 = Detail 4 (b)



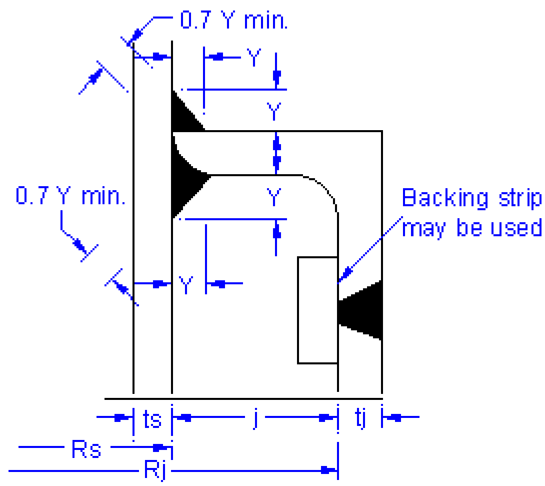
Div. 1 = Type (e-1)

Div. 2 = Detail 4 (c)



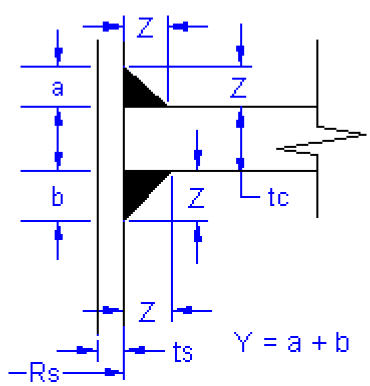
Div. 1 = Type (e-2)

Div. 2 = Detail 4 (d)



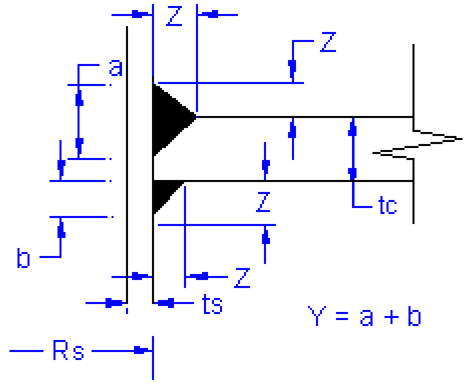
Div. 1 = Type (f-1)

Div. 2 = Detail 5 (a)



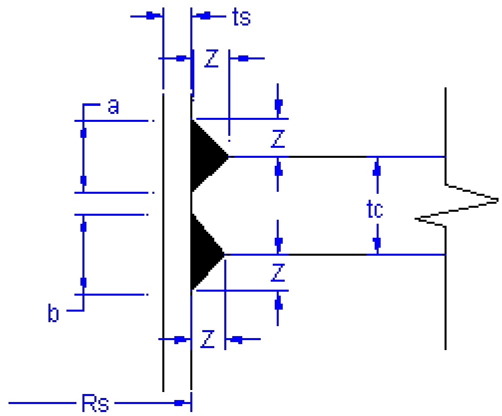
Div. 1 = Type (f-2)

Div. 2 = Detail 5 (b)



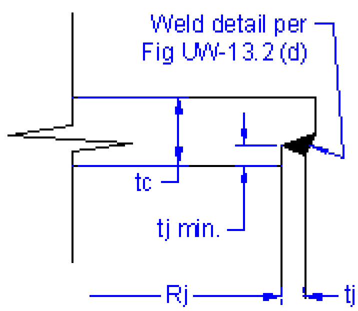
Div. 1 = Type (f-3)

Div. 2 = Detail 5 (c)



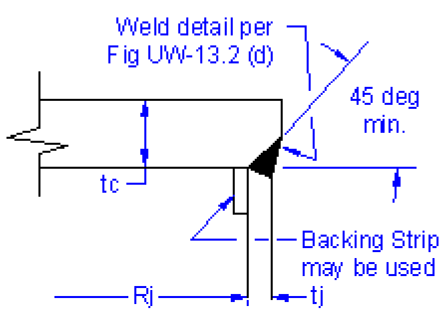
Div. 1 = Type (g-1)

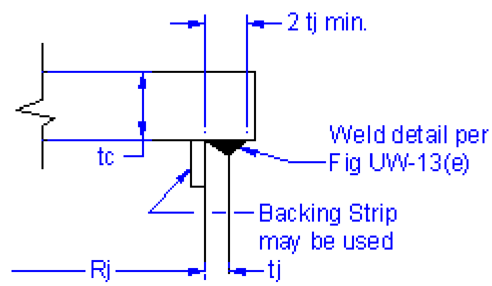
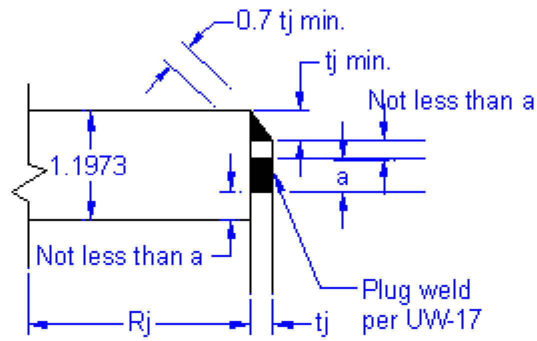
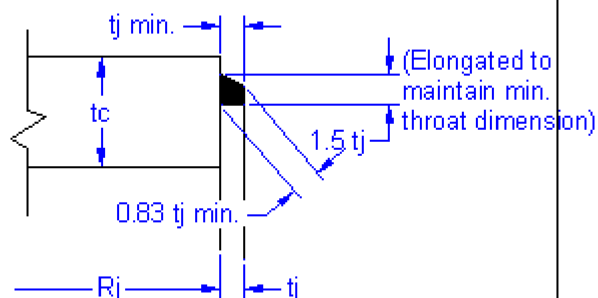
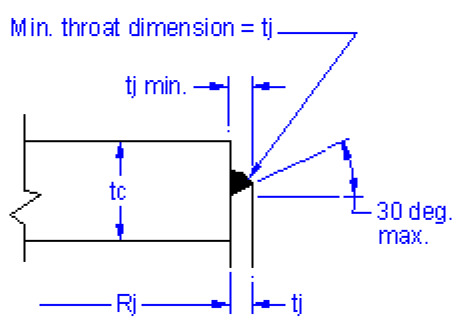
Div. 2 = Detail 6 (a)



Div. 1 = Type (g-2)

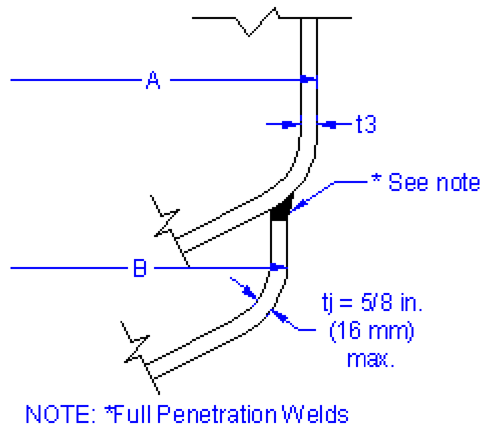
Div. 2 = Detail 6 (b)



Div. 1 = Type (g-3) Div. 2 = Detail 6 (c)	
Div. 1 = Type (g-4) Div. 2 = Detail 6 (d)	
Div. 1 = Type (g-5) Div. 2 = Detail 6 (e)	
Div. 1 = Type (g-6) Div. 2 = Detail 6 (f)	

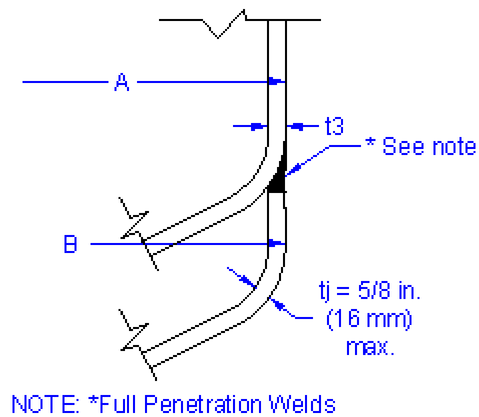
Div. 1 = Type (h, i-1(a))

Div. 2 = Detail 7 (a)



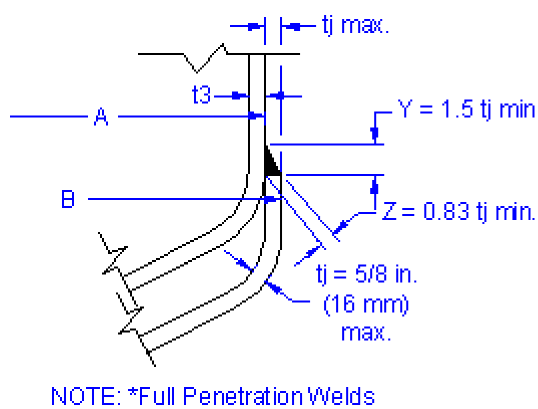
Div. 1 = Type (h, i-1(b))

Div. 2 = Detail 7 (b)



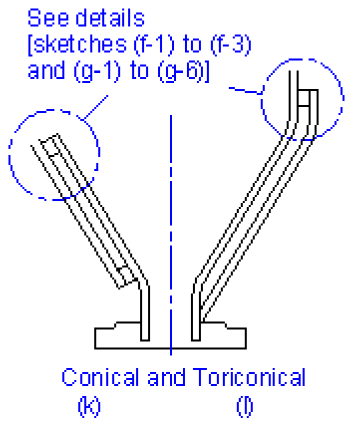
Div. 1 = Type (h, i-2)

Div. 2 = Detail 7 (c)



Div. 1 = Type (k)

Div. 2 = Detail 8



Closure Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Closure Thickness

Thickness of the plate used for the closure.

Total Corrosion Allowance

Enter the thickness of the corrosion allowance for the closure.

API-579 Flaw/Damage Input/Analysis

Home tab: **Details** > **API-579 Flaw/Damage Input/Analysis**

Adds API-579 Flaw/Damage Input/Analysis information on the selected element.

Previous

Select to view the previous flawed defined on the selected element.

Add New Flaw

Select to define another flaw on the selected element.

Delete

Select to delete the current flaw from the element.

From Node

Displays the **From Node** for the selected element.

Distance from 'From' Node

Enter the distance between the **From Node** and the flaw that you are defining.

New Damage Description

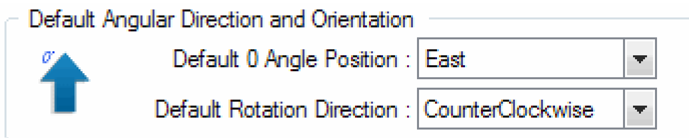
Type a description for the damage.

Layout Angle

Enter the angle between the designated zero degree reference direction on the vessel drawing and the centerline of the weight at the point where it is attached to the shell.

For a horizontal vessel, the zero degree line is at 12:00 (looking at a clock); 90 degrees is at 3:00. Entering these layout angles is important if the horizontal vessel has a liquid level and the nozzles are being designed using **Design P + Static Head**, selected for **Nozzle/Clip Design Pressure Options** on the **Load Cases** tab. For a vertical vessel, the angle is more arbitrary. For purposes of rendering the graphics, the assumption is that the zero degree line is at 3:00 and 90 degrees is 12:00.

The position of the zero degree reference direction (North, West or East) and the angular rotation (clockwise or counterclockwise) are set in **Tools** tab, **Set Configuration Parameters** , *Default Value Settings Tab (Configuration Dialog)* (page 233).



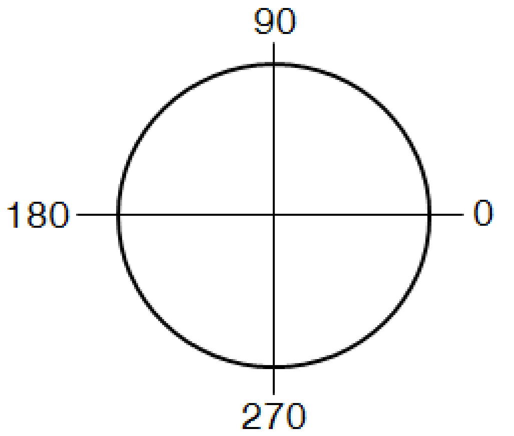
If the new orientation is different from the orientation of the current model, the software asks to update the model with the new settings. If the settings are updated, the weights and areas of platforms must be recalculated:

- Click **List Dialog** (page 218) . On the **Detail Listing** dialog box, click the **Platform** tab so that platform data is visible. Click **Platform Wind Area** and **Platform Weight** to recalculate.
- Click **Platform Input** for each platform. Tab through the fields in the **Platform** dialog box, causing the weight and area to be recalculated

NOTE The angular settings apply to nozzles, clips, legs, lifting lugs, support lugs, base rings, platforms, weights and half-pipe jackets.

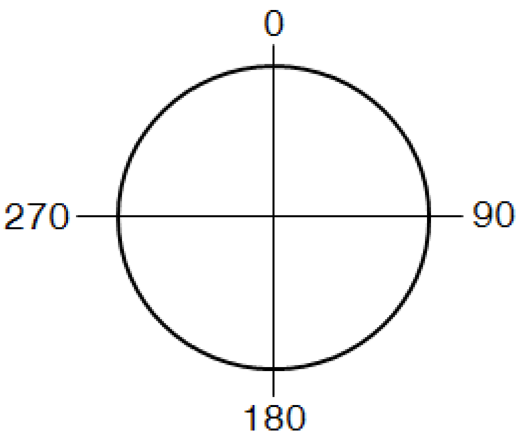
Examples

Default Orientation



Vertical Vessel (Top or Plan View)
Horizontal Vessel (Left End View)

Angle Position: North
Rotation Direction: Clockwise



Vertical Vessel (Top or Plan View)
Horizontal Vessel (Left End View)

Flaw Type

Select the type of damage observed from the following flaw types.

- **Part 4 (General Metal Loss)** - Select this option when the general metal loss (uniform or local, inside or outside) exceeds, or is predicted to exceed, the corrosion allowance before the next scheduled inspection. Assessments are based on point thickness readings and thickness profiles (see paragraph 4.3.3), whether the metal loss is uniform or local, and the degree of conservatism acceptable for the assessment. You can use the methodology shown in Figure 4.2 to determine the assessment procedure for the evaluation.

- **Part 5 (Local Thinning Area)** - Select this option when the metal loss on the surface of the component is localized, and the length of metal loss is about the same as the width.
- **Part 5 (Groove like Flaw)** - Select this option when the flaw is either:
  - **Groove** - A local, elongated, thin-spot caused by directional erosion or corrosion. The length of the metal loss is significantly greater than the width.

    NOTE A sharp radius might be present at the base of a groove-like flaw.
  - **Gouge** - A local, elongated, mechanical removal or relocation of material from the component surface resulting in a reduction in wall thickness at the flaw. The length of the gouge is much greater than the width. The material might have been cold worked in the formation of the flaw. Gouges are typically caused by mechanical damage, for example, denting and gouging of a section of pipe by mechanical equipment during the excavation of a pipeline. Gouges are frequently associated with dents due to the nature of mechanical damage. If a gouge is present, the assessment procedures of Part 12 shall be used.
- **Part 6 (Pitting)** - Select this option when one of the four types of pitting is present:
  - widely scattered pitting over a significant region of the component,
  - a local thin area (LTA) located within a region of widely scattered pitting,
  - localized regions of pitting, and
  - pitting confined within a region of an LTA.

  The flowchart in Figure 6.2 provides details of the required assessment procedures. Depending on the type of pitting damage, you must use for evaluation either the assessment methods in Part 6 or a combination of assessment methods in Part 5 and Part 6.

Assessment Level

Select the assessment level. See the description below that corresponds to your selection for Flaw Type.

Part 4 (General Metal Loss)

Select **Level 1 and Level 2** if the following four points are true. Otherwise select **Level 1 only** or **Level 2 only**.

- The original design criteria was in accordance with a recognized code or standard (see Part 1, paragraphs 1.2.2 or 1.2.3).
- The metal loss region has relatively smooth contours without notches (that is, negligible local stress concentrations).
- The component is not in cyclic service.

  TIP A component is not in cyclic service if the component is subjected to less than 150 cycles throughout its previous operating history and future planned operation, or the component satisfies the cyclic service screening procedure in Annex B1, paragraph B1.5.2. A cycle is defined as pressure or temperature variations including operational changes and start-ups and shut-downs.
- The following limitations on component types and applied loads are satisfied:
  - Level 1 Assessment - Type A Components subject to internal pressure or external pressure (that is, supplemental loads are assumed to be negligible).

    - Level 2 Assessment - Type A or B Components (see Part 4, paragraph 4.2.5) subject to internal pressure, external pressure, supplemental loads (see Annex A, paragraph A.2.7), or any combination thereof.

Part 5 (Local Metal Loss)

Select **Level 1 and Level 2** if the following five points are true. Otherwise select **Level 1 only** or **Level 2 only**.

- The original design criteria were in accordance with a recognized code or standard (see Part 1, paragraphs 1.2.2 or 1.2.3).
- The material is considered to have sufficient material toughness. If there is uncertainty regarding the material toughness, then a Part 3 assessment should be performed. If the component is subject to embrittlement during operation due to temperature or the process environment, a Level 3 assessment should be performed. Temperature or process conditions that result in material embrittlement are discussed in Annex G.
- The component is not in cyclic service.

  TIP A component is not in cyclic service if the component is subjected to less than 150 cycles throughout its previous operating history and future planned operation, or the component satisfies the cyclic service screening procedure in Annex B1, paragraph B1.5.2. A cycle is defined as pressure or temperature variations including operational changes and start-ups and shut-downs.
- The following limitations on component types and applied loads are satisfied:
    - Level 1 Assessment - Type A Components subject to internal pressure or external pressure (that is, supplemental loads are assumed to be negligible).
    - Level 2 Assessment - Type A or B Components (see Part 4, paragraph 4.2.5) subject to internal pressure, external pressure, supplemental loads (see Annex 2C, paragraph 2C.2.7), or any combination thereof.
- A flaw characterized as a groove in accordance with paragraph 5.2.1.b has a groove radius that satisfies the requirements in paragraph 5.4.2.2.f.

Part 6 (Pitting)

Select **Level 1 and Level 2** if these six points are true. Otherwise select **Level 1 only** or **Level 2 only**.

- The original design criteria were in accordance with a recognized code or standard (see Part 1, paragraphs 1.2.2 or 1.2.3).
- The material is considered to have sufficient material toughness. If there is uncertainty regarding the material toughness, then a Part 3 assessment should be performed. If the component is subject to embrittlement during operation due to temperature and/or the process environment, a Level 3 assessment should be performed. Temperature and/or process conditions that result in material embrittlement are discussed in Annex G.
- The component is not in cyclic service.

  TIP A component is not in cyclic service if the component is subjected to less than 150 cycles throughout its previous operating history and future planned operation, or the component satisfies the cyclic service screening procedure in Annex B1, paragraph B1.5.2. A cycle is defined as pressure or temperature variations including operational changes and start-ups and shut-downs.
- The following limitations on component types and applied loads are satisfied:

  - Level 1 Assessment - Type A Components subject to internal pressure or external pressure (that is, supplemental loads are assumed to be negligible).
  - Level 2 Assessment - Type A or B Components (see Part 4, paragraph 4.2.5) subject to internal pressure, external pressure, supplemental loads (see see Annex 2C, paragraph 2C.2.7), or any combination thereof.
- Additional requirements for Level 1 assessments are:
  - The pitting damage is arrested.
  - The pitting damage is located on only one surface (either inside or outside) of the component.
  - The pitting damage is composed of many pits; individual pits or isolated pairs of pits should be evaluated using the assessment procedures in Part 5.
- Additional requirements for Level 2 assessments are:
  - The pitting damage is characterized by localized regions of pitting, an LTA located in a region of widely scattered pitting, or pitting that is confined within an LTA.
  - The pitting damage is located on either one surface or both surfaces of the component and the pitting damage is not overlapping (see Figure 6.3)
  - The pitting damage is composed of many pits; individual pits or isolated pairs of pits should be evaluated as LTAs using the assessment procedures in Part 5.
- A **Level 2 only** assessment should be performed if:
  - An appropriate pit comparison chart cannot be found (see paragraph 6.3.3.1).
  - A more detailed assessment of widespread pitting (inclusion of the pit-couple orientation) is required.

Future Corrosion Allowance, local [FCAml]

Enter the future corrosion allowance (the projected future metal loss) applied to the region of local metal loss.

Future Corrosion Allowance [FCA]

Enter the future corrosion allowance (the projected future metal loss) away from the region of local metal loss. The future corrosion allowance should be based on past inspection information or corrosion rate data relative to the component material in a similar environment. Corrosion rate data can be obtained from API Publication 581 or other sources (see paragraph 2C.2.7). The FCA is calculated by multiplying the anticipated corrosion rate by the future service period considering inspection interval requirements of the applicable inspection code.

Note for Part 6 – Per paragraph 6.2.7, the future corrosion allowance (FCA) shall be based on the projected future metal loss in the pitting region. The FCA is not applied to the depth or diameter of the pits.

Uniform Metal Loss [LOSS]

Specify the amount of uniform metal loss away from the local metal loss location at the time of the assessment. The software uses this value to determine a suitable wall thickness to use in the assessment.

Note for Part 4 - This is not applicable to Part 4 as the uniform metal loss is general loss.

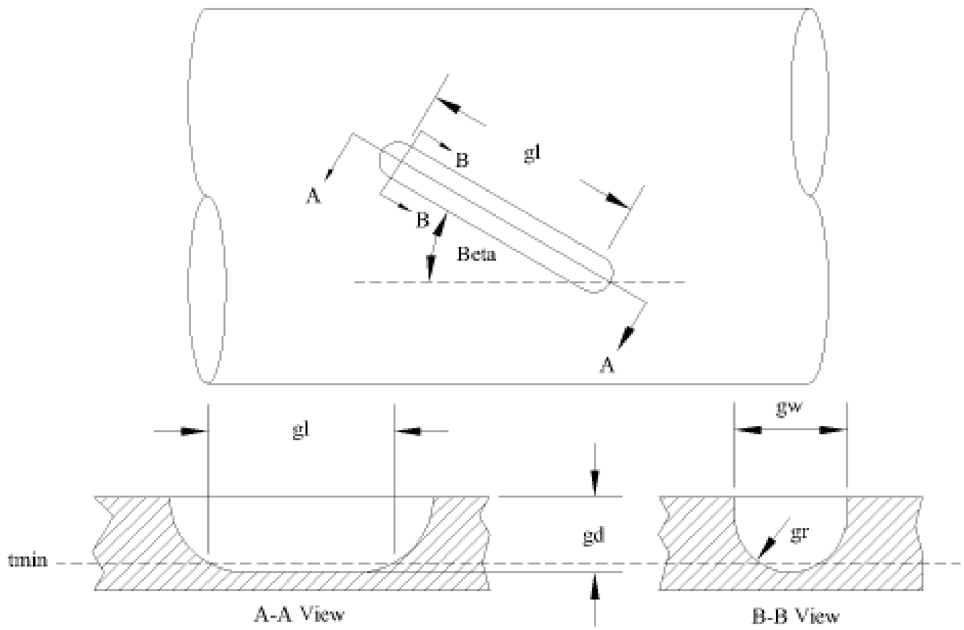
Remaining Strength Factor [RSFa]

Enter the allowable remaining strength factor [RSFa]. The recommended RSFa for all major design codes per the 2016 edition of API 579 is 0.90.

Local Metal Loss Data

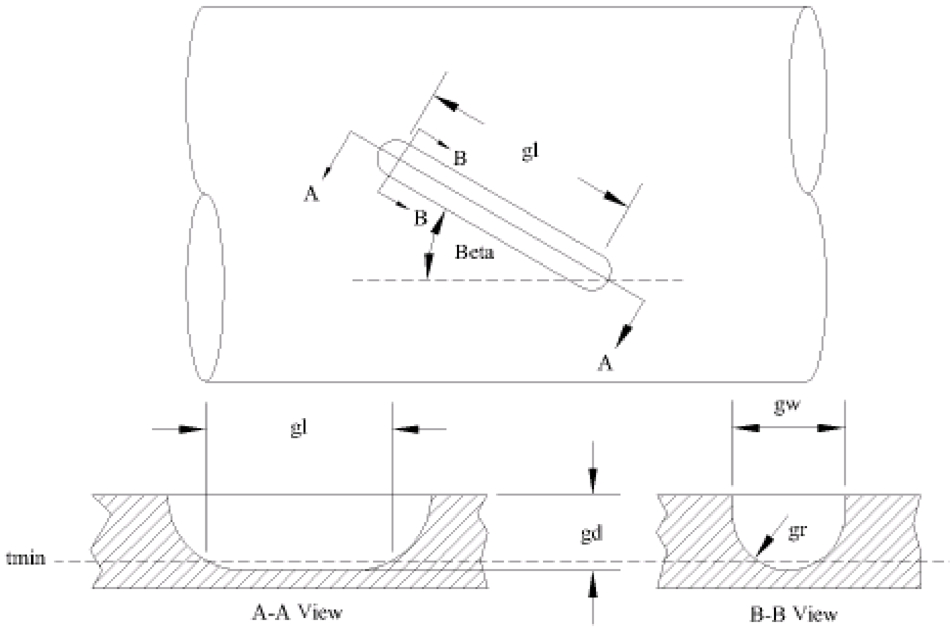
Orientation w/Respect to Long. Axis [beta]

Enter the angle to the groove-like flaw from the longitudinal axis.



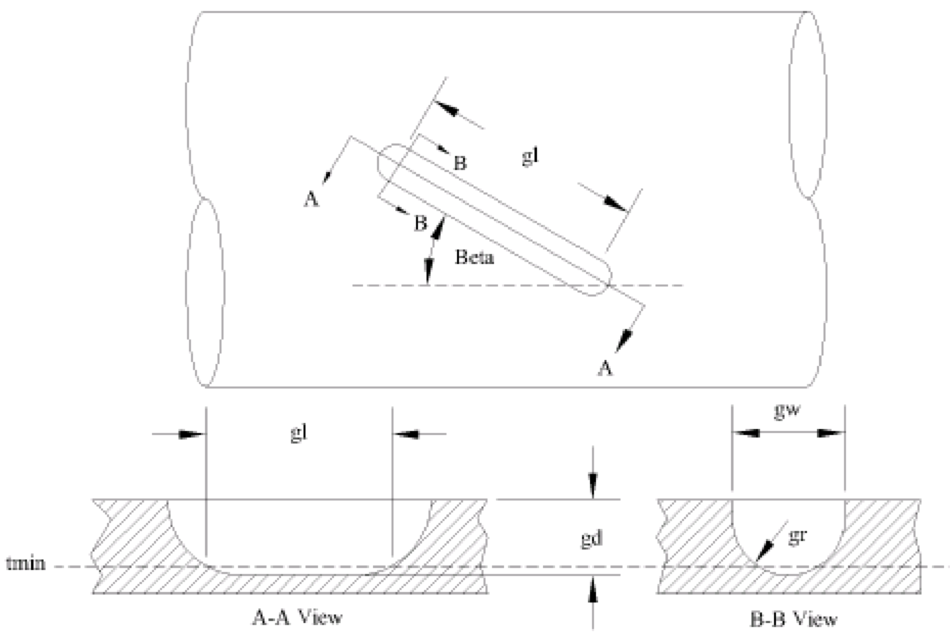
Length of Groove-like Flaw [gl]

Enter the length of the flaw. This dimension is based on the corresponding critical thickness profile (CTP), which is measured parallel and perpendicular to the groove and should include the projected future corrosion growth.



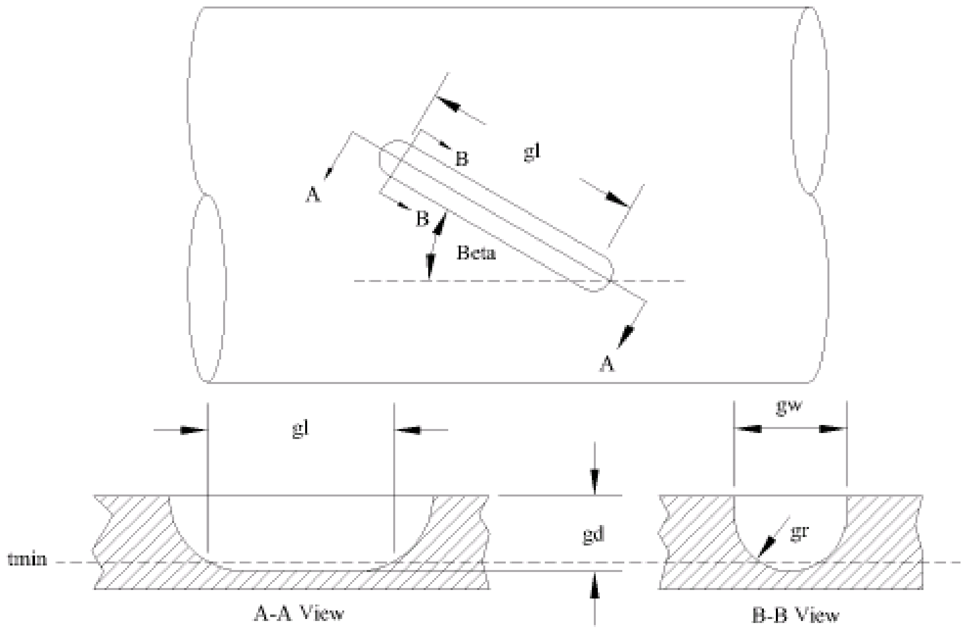
Width of Groove-like Flaw [gw]

Enter the width of the flaw. This dimension is based on the corresponding critical thickness profile (CTP), which is measured parallel and perpendicular to the groove and should include the projected future corrosion growth.



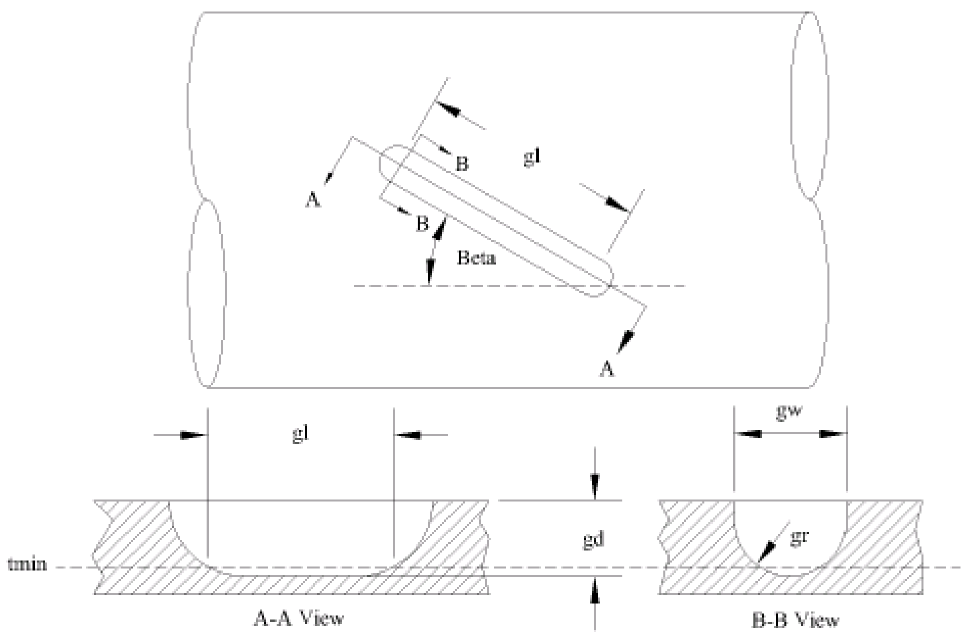
Radius at Base of Groove-like Flaw [gr]

Enter the radius at the base of the flaw. This dimension is based on the corresponding critical thickness profile (CTP), which is measured parallel and perpendicular to the groove and should include the projected future corrosion growth.



Depth of Groove-like Flaw [d]

Enter the depth of the flaw. This dimension is based on the corresponding critical thickness profile (CTP), which is measured parallel and perpendicular to the grove and should include the projected future corrosion growth.



Local Thinning Area Longitudinal Dimension [s]

Enter the longitudinal extent or length of the region of local metal loss at the time of the inspection.

Local Thinning Area Circumferential Dimension [c]

Enter the circumferential extent or length of the region of local metal loss at the time of the inspection.

Distance to Nearest Major Structural Discontinuity [Lmsd]

Enter the distance to the nearest major structural discontinuity. This box indicates that the software checks the limiting flaw size criteria.

Measurement Data

Point Thickness Readings (Part 4 only)

Select this option to use point thickness readings in the assessment. This option is only available when Flaw Type is set to **Part 4 (General Metal Loss)**.

If you select this option, you are confirming the assumption of uniform metal loss. Point thickness readings may be used to characterize the metal loss in a component if there are no significant differences in the thickness reading values obtained at inspection monitoring locations.

A minimum of 15 thickness readings should be used unless the level of NDE used can confirm that the metal loss is general. In some cases, additional readings might be required based on the size of the component, the construction details used, and the nature of the environment resulting in the metal loss. A sample data sheet to record thickness readings is shown in Table 4.2.

If the Coefficient of Variation (COV) of the thickness readings is greater than 10%, then thickness profiles shall be considered for use in the assessment (see paragraph 4.3.3.3). The COV is defined as the standard deviation divided by the average.

Critical Thickness Profile

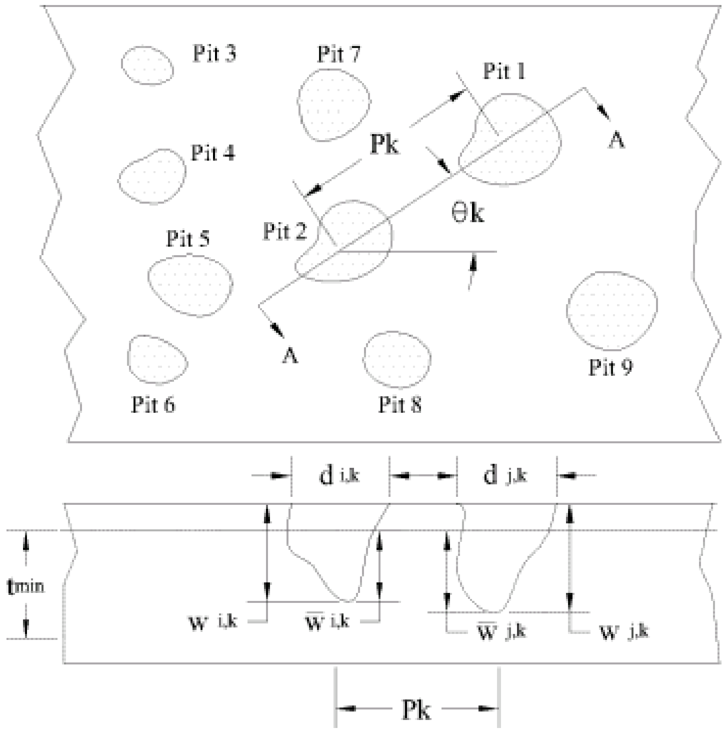
Select this option to use thickness profiles to characterize the metal loss in a component. You must use this option if there is a significant variation in the thickness readings, which can indicate the metal loss might be localized. Thickness profiles should be used to characterize the remaining thickness and size of the region of metal loss.

Pitting Data (Part 6 Only)

Select this option to consider pitting data. This option is only available when Flaw Type is set to **Part 6 (Pitting)**.

- **Level 1 Assessment** - The surface damage is measured in terms of pitted area, and the maximum pit depth is used to quantify the extent of pitting damage. You can use the standard pit charts (see Figures 6.6 to 6.13) to compare the actual damage on the component to the damage represented on the pit chart. The pit chart and an estimate of the maximum pit depth are used to determine acceptability. A cross-sectional UT thickness scan can determine the pitting profile. Guidelines for determining the maximum pit depth are in paragraph 6.3.4.1 of the code.
- **Level 2 Assessment** - The surface damage is measured using a pit-couple (two pits separated by a solid ligament; see Figure 6.14). The metal loss of each pit in a pit-couple is modeled as an equivalent cylinder. The diameter and depth of each pit, and the distance between the pit centers are also required. The orientation of the pit-couple in the biaxial stress field can also be included in the assessment (see Figure 6.14). The depth and diameter of a pit should be carefully measured because of the variety of pit types that can occur in service (see Figure 6.5). If the pit has an irregular shape, you

should use a diameter and depth that encompasses the entire shape for the assessment.



Edit Measurement Data

Activates the **API 579 Point Thickness Readings** dialog box, **Critical Thickness Profile** dialog box, or the **API 579 Pitting Data** dialog box. Use the dialog box to enter the measurement data for your analysis.

Pitting

Pit Chart

Select the pit chart to use.

- Grade 1
- Grade 2
- Grade 3
- Grade 4
- Grade 5
- Grade 6
- Grade 7
- Grade 8

Maximum Pit Depth [wmax]

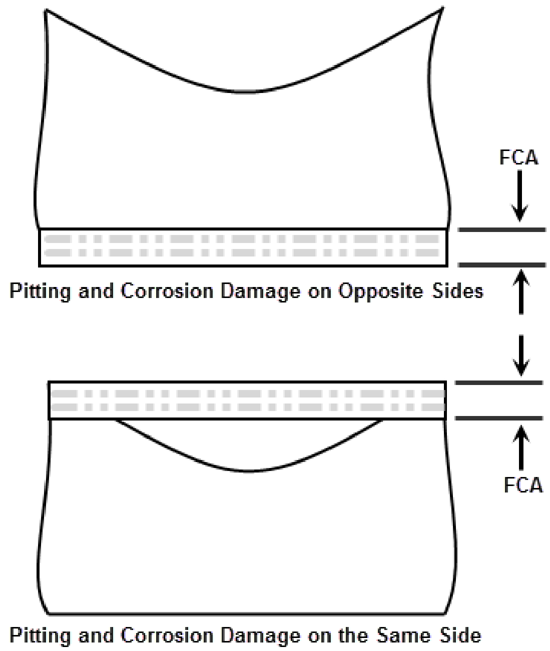
Enter the pit depth of the deepest pit. This option is only for a Part 6, Level 1 Assessment.

Maximum Pit Diameter [d]

Enter the pit diameter of the widest pit. This option is only for a Part 6, Level 1 Assessment.

Pitting and Corrosion

Select whether the pitting and corrosion act on the same side or on opposite sides of the vessel (see Figure 6.4).



Pit Category

Select the pit category:

- **Widespread Pitting** - Select this option when the pitting is widespread across the element.
- **Localized Pitting** - Select this option if the pitting is localized.
- **LTA in Region of Widespread Pitting** - Select this option when the region of local metal loss (LTA) is in an area of widespread pitting. When you select this option, the software calculates a combined Remaining Strength Factor for assessing damage.
- **Pitting Confined in Local Thin Area** - Select this option when the pitting damage is confined within the local thin area.

Remaining Life Assessment Options

Thickness Approach (Parts 4 and 5 only)

Select this option to calculate the remaining life of a component based on a minimum required thickness for the intended service conditions, thickness measurements from an inspection, and an estimate of the anticipated corrosion rate. This method is acceptable for calculating the remaining life for Type A Components (see paragraph 4.2.5). Because the thickness-based approach (this option) might produce non-conservative results for the remaining life when applied to Type B, Class 1 Components (see paragraph 4.2.5), you need to use the MAWP Approach for those components.

MAWP Approach

Select this option to calculate the remaining life of Type A or Type B, Class 1 components using a systematic method. You must select this option for Type B, Class 1 components. This approach also ensures that the design pressure is not exceeded during normal operation if the future corrosion rate is accurately established.

Increase in Pit Size

Select this option if the pitting corrosion propagates by pit size, that is the depth or the diameter of the pits increases. Enter the pit corrosion rates in the **Pit Depth [PPRpit-depth]** and **Pit Diameter [PPRpit-dia]** fields.

NOTE This option is only available when **Flaw Type** is set to **Part 6 (Pitting)**.

Increase in Pit Density

Select this option if the pitting corrosion propagates by the growth of existing pits or the formation of new pits. This corrosion method results in an increase in the pit density, that is the spacing between pits decreases. Enter the pit corrosion rate in **Pit Spacing (PPRpit-spa)**.

NOTE This option is only available when **Flaw Type** is set to **Part 6 (Pitting)** and **Assessment Level** is set to **Level 2 Only** or **Level 1 and Level 2.**

Increase in Pit Region

Select this option if the localized pitting region increases either longitudinally or circumferentially. An increase in the local pitting region is similar to an increase in a local thinning area (LTA). Enter the enlargement rates of the pitting region in the **Longitudinal [CrateS]** and **Circumferential [CrateC]** fields.

NOTE This option is only available when **Flaw Type** is set to **Part 6 (Pitting)** and **Assessment Level** is set to **Level 2 Only** or **Level 1 and Level 2**.

Previous Average Measured Thickness [tam_prev]

Enter the average measured thickness from the previous inspection. The software uses this value in the Remaining Life calculations.

Corrosion Rates

Corrosion Rate [Crate]

Enter the future corrosion rate determined from previous thickness data, corrosion design curves, or experience in similar services. This option is only for a Part 4 Assessment.

Away from Metal Loss [CrateRD]

Enter the anticipated future corrosion rate away from the local metal loss.

NOTE This option is only for a Part 5 Assessment.

Longitudinal [CrateS]

Enter the estimated rate of change of the meridional length of the region of local metal loss. This option is only for a Part 5 or a Part 6 assessment.

Circumferential [CrateC]

Enter the estimated rate of change of the circumferential length of the region of local metal loss. This option is only for a Part 5 or a Part 6 assessment.

Pit Depth [PPRpit-depth]

Enter the estimated rate of change of the pit characteristic depth. Pits can corrode in many forms and sometimes the pit depth is not affected by corrosion. If you leave this value at 0, the software does not use a pit depth corrosion rate in the remaining life pitting calculations. However, if corrosion affects both the pit diameter and the depth and you specify those values, the software takes both propagation rates into consideration for the remaining life calculation.

Pit Diameter [PPRpit-dia]

Enter the rate of change of the pit characteristic diameter. Pits can corrode in many forms, and sometimes the pit diameter is not affected by corrosion. If you leave this value at 0, the software does not use the pit diameter corrosion rate in the remaining life pitting calculations. However, if corrosion affects both the pit diameter and the depth and you specify those values, the software takes both propagation rates into consideration for the remaining life calculation.

Pit Spacing [PPRpit-spa]

Enter the rate of change, as a negative number, of the average spacing between two nearest neighboring pits. Measure the rate of change from the center of each pit.

Overriding Values

Activates the *API Overriding Values Dialog Box* (page 212). Use this dialog to specify settings for the corrosion allowance on remaining life, design condition specifics, and overriding force and moment details.

API Overriding Values Dialog Box

Click **Overriding Values** on the **API-579 Flaw/Damage Input/Analysis** dialog to activate the **API Overriding Values** dialog. Use this dialog to specify overriding values information for the selected element in areas such as the corrosion allowance on remaining life, design condition specifics, and overriding force and moment details.

Input/Output Panel

The following commands are available on the **Input/Output** panel on the **Home** tab.

	Input - Shows and hides the data input tabs located in the bottom-left of the interface. For more information, see *Input* (page 213).
	Component Analysis - Activates CodeCalc. For more information, see *Component Analysis* (page 213).
	Review Database - Opens the output database in Microsoft Access. For more information, see *Review Database* (page 213).

Input

Home tab: **Input / Output > Input**

Activates and hides the input tabs that appear on the bottom-left corner of the interface.

General Input

Enter data for a vessel element. For more information, see *General Input Tab* (page 264).

Report Headings

Enter page heading text and cover sheet text for reports. For more information, see *Report Headings (Heading Tab)* (page 350).

Design/Analysis Constraints

Enter data such as pressures and temperatures, hydrotest data, and wall thicknesses. For more information, see *Design Constraints Tab* (page 352).

Load Cases

Enter stress combination and nozzle pressure load cases. For more information, see *Load Cases Tab* (page 364).

Wind Loads

Select a wind design code and enter data required by that code. For more information, see *Wind Loads (Wind Data Tab)* (page 375).

Seismic Loads

Select a seismic design code and enter data required by that code. For more information, see *Seismic Loads (Seismic Data Tab)* (page 432).

Component Analysis

Home tab: **Input / Output > Component Analysis**

Opens CodeCalc and perform analyses separate from the PV Elite model. You can perform analyses such as local loads on nozzles and tubesheet calculations. For more information, see CodeCalc Help.

Review Database

Home tab: **Input / Output > Review Database**

Opens the database in Microsoft Access so you can review the contents. You need to create the database first using **Create Database** in the **Auxiliary** panel. For more information, see *Create Database* (page 220).

Utility Panel

Provides miscellaneous edit commands for functions such as insert, delete, update, and flip to edit elements. The following commands are available on the **Utility** panel on the **Home** tab.

	Insert Element - Insert an element before or after the current element.
	Delete Element - Delete the currently selected element.
	Propagate Element Diameter - Propagate element diameter to connected elements.
	Share Information - Share information between elements.
	Flip Element Orientation - Flips the orientation of the current element. Use this command when you want to change the orientation of just a single element. For more information, see *Flip Element Orientation* (page 215). Use **Flip Model Orientation** on the **Tools** tab to flip the entire model.
	Select Material - Select a material from the materials database.
	Zoom Mode - Zoom in and out of elements in 2D.
	View Element - View 2D element plan or layout view.
	Compute Ligament Efficiencies - Calculates ASME VIII-1 UG-53.x or 4.10.x ligament efficiencies for tube spacing. For more information, see *Compute Ligament Efficiencies* (page 252).
	Retirement Limit Calculation - Generate retirement limit calculations. For more information, see *Retirement Limit Calculations* (page 215).

Share Information

Home tab: **Utility** > **Share Information**

If your vessel has multiple elements and you need to copy element data from one element to the other elements, use this command option.

Select the element that has the data you want to share, and then click the (Share Information) icon. In the **Data Share Dialog** window, type the node number of the starting element for which to copy data in **from "From Node"**. Type the node number of the ending element for which to copy data in **to "From Node"**. Click the check boxes for the data items you want to share, such as the diameter, thickness, corrosion allowance, internal or external pressure, and material properties, and then click **OK**.

To change nozzle data, select **Change Nozzle Materials**. The **Select New Nozzle and Nozzle Flange Material** window displays. You can select a new nozzle, pad, or flange material. You can also change the class or grade of a nozzle flange. To exclude any nozzles from being updated, click the box next to the nozzle in the **Exclude Nozzle** list box. Click **Update all Nozzles Now** to apply the changes. Click **Exit** to close the window without applying the changes.

Flip Element Orientation

Home tab: **Utility** > **Flip Element Orientation**

Vessels are defined one element to the next (from bottom-to-top for vertical vessels and from left-to-right for horizontal vessels). If the vessel begins with a skirt element, it is a vertical vessel. Vertical vessels on legs and horizontal vessels start with a head element. If that first element is improperly oriented for the vessel that you want to model, use this command to correct the orientation.

After the second element is added, the vessel can no longer be flipped between horizontal and vertical using this command. However, you can use this command later if heads, body flanges, or cone elements need to be flipped. To flip the entire model after the second element has been added, use **Flip Model Orientation** on the **Tools** tab.

Retirement Limit Calculations

Home tab: **Utility** > **Retirement Limit Calculation**

After a vessel or exchanger has been in service, it experiences corrosion, fouling, erosion as well as other issues during the course of its use. As a result of corrosion, the thickness of the vessel wall, nozzles, and so forth decreases over time. At some point, the thickness of the component might not fulfill the requirements of the code. This could simply be that the wall thickness is no longer acceptable for internal or external pressure or that a nozzle might no longer meet code requirements for basic minimum structural wall thickness or area replacement.

It is useful to determine the thickness of these components to determine their retirement limit. To do this in the past, you had to iterate on the element’s corrosion allowance until it failed in some way. The element thickness minus the iterated corrosion allowance yields the retirement limit. This process can be very tedious and time consuming especially considering the interaction between nozzles and their host elements. Depending on the complexity of the vessel or exchanger, a rerate can take several days. Cone to shell junction analysis, horizontal vessel analysis and others are also considered during the re-rating analysis. Days of manual work are collapsed into just a few seconds of analysis using this feature.

Generate retirement limit calculations for this model

Select to perform retirement limit calculations during analysis runtime and displays element retirement limits on the status bar.

If the joint efficiency of the element is less than one, there will be two extra calculations shown on the status bar: **Tmin, as** (Tmin at seam) and **Tmin, afs** (Tmin away from seam). Because the thickness calculation involves the joint efficiency, the application shows both results for your convenience. Below is an excerpt from the internal pressure report for a cylinder:

Element/Component Rerate Information:

```
Corrosion Allowance at Retirement Limit:  0.7124 in.
Retirement Limit (Corroded Condition):    0.2876 in.
Rerate Corrosion Allowance:               0.1250 in.
Corroded Retirement Limit:                0.8750 in.
```

```
Retirement Limit, MDMT:                    0.4520 in.
Retirement Limit, Away from Seam:          0.2445 in.
```

Enter advanced rerating information

Select to enable **Minimum UT Thickness** and **Future Corrosion Allowance at next Inspection Interval**.

This additional information can be entered for the main elements and the nozzles. Doing so allows the application to perform additional calculations that can be found in the re-rating analysis report.

Generate Maximum Safe Operating Temperature Curves

Select to compute a pressure temperature curve known as an MSOT (Maximum Safe Operating Temperature) curve. If selected, the application computes pressure versus temperature curves for the shells, heads and nozzles where an MDMT calculation is required. For each pressure, the governing temperature is consolidated for each component and the governing value is chosen. The curve data is the then plotted by the graph processor. There is an option to plot the curve in the output processor after the analysis is finished. If the point of actual process temperature and pressure is plotted and it falls outside of the curve, this is an unsafe condition.

Consider design MDMT when determining tmin values for elements

Select to check the MDMT while performing the rerate analysis. Because calculation of MDMT values is a function of the stress in the vessel component as well as the thickness, this can also influence the retirement limit results. Selecting this option might not be required however, especially if the control system has safeties that will shut the process down in the event that the temperature is below a certain set point.

After the analysis is run and you return to input, the Tmin values can be shown graphically by holding down the CTRL key and moving the mouse to the component. Release the control key to display a tooltip window with various element information including the Tmin value and governing reason. If the model is changed in any way, the analysis must be re-run to obtain the new Tmin values.

Auxiliary Panel

The following commands are available from the **Auxiliary** panel on the **Home** tab.

	Pipe Properties - Open the Seamless Pipe Selection dialog box. For more information, see *Seamless Pipe Selection Dialog Box* (page 217).
	List Dialog - Open the Detail Listing dialog box. For more information, see *List Dialog* (page 218).
	Write Foundation 3D File - Create a Foundation 3D file after the model is analyzed. For more information, see www.dimsoln.com.
	Export to DXF File - Export the vessel geometry to a Release 12 Data Exchange File (CAD file). For more information, see *Setting Up the Required Parameters* (page 219).
	Rigging Results - Display rigging results.

	Create Database - Create database of input files. For more information, see *Create Database* (page 220).
	Element Properties - Display a list of element weights, volumes, and surface areas.
	Set Configuration Parameters - Open the Configuration dialog box. For more information, see *Configuration* (page 224).
	Create/Review Units - Create or review unit files. For more information, see Create/Review Units.
	Calculator - Open the Windows calculator.
	Switch Datum Input - Enable/Disable nozzle data entry from the datum line.
	Create 3D PDF Files - Enable/Disable creation of 3D model PDF files. For more information, see *Create 3D PDF Files* (page 220).

Seamless Pipe Selection Dialog Box

Home tab: **Auxiliary > Pipe Properties**

Opens the **Seamless Pipe Selection** dialog box, where you can select piping properties for the shell, nozzle, or leg from the currently selected piping database.

Pipe Schedule

Select the pipe schedule from the drop-down menu.

Nominal Pipe Diameter

This value automatically gets populated after the pipe schedule is selected. This value can then be modified if needed.

Deduct Mill Tolerance from Thickness?

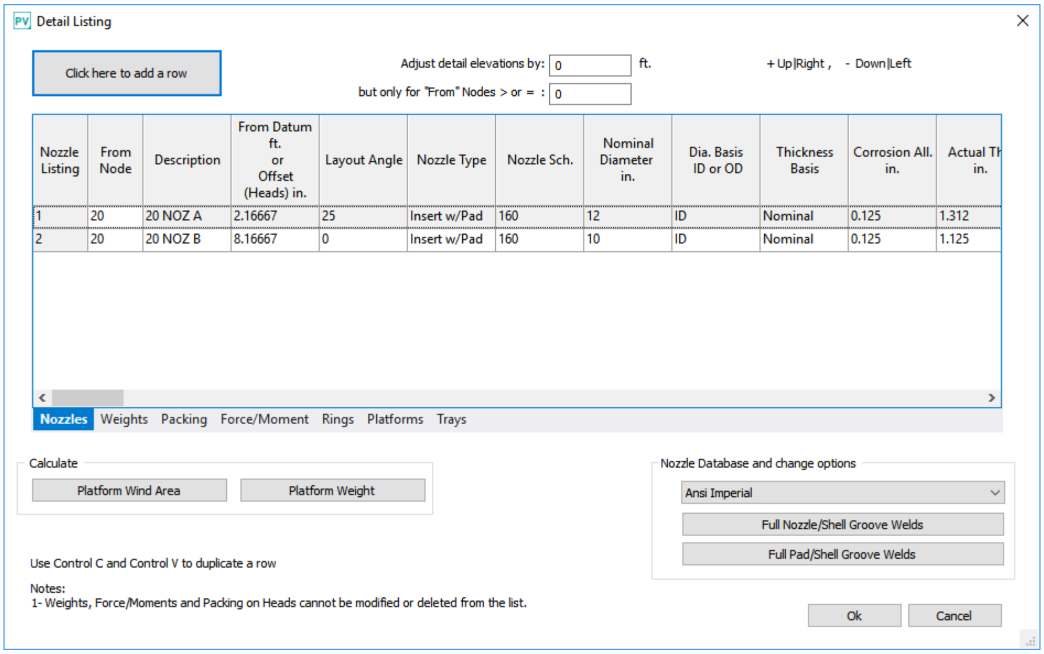
Check this box if you want to deduct the mill tolerance from the thickness.

NOTE **Deduct Mill Tolerance from Thickness?** has no effect on nozzles or pipe legs.

List Dialog

Home tab: **Auxiliary > List Dialog**

Opens the **Detail Listing** dialog box, where you add details for platforms, nozzles, weights, packing, forces/moments, trays, and pressure rings.



The location of the detail can be specified from the datum position. Select the type of detail to edit by clicking its tab. Enter the needed data for each row. Press **+** to add a row.

The software automatically assigns the value for **From Node**.

IMPORTANT **Description** is required. If you do not enter one, the software ignores the row and the data is lost.

All other data must be entered as required.

Rows of data can be duplicated from one row to the next. Click on the listing number of the item to copy to highlight the row. Copy the row and paste it to a blank row. Change any data that might be different for that detail.

Adjust detail elevations by

Enter a value to shift the position of all details by the specified elevation distance. A negative value will move details down or left. A positive value moves the details up or right. This option is useful when all of the details such as rings, nozzles, and trays need to be adjusted by a specified amount. This may happen if an element is inserted into the model after it has been completed and the detail elevations need to be kept constant.

CAUTION If the adjustment moves a detail (such as a tray) into an element (such as a body flange), the software does not allow this, and the detail is lost and cannot be recovered.

But only for "From" nodes > or =

Enter the **From Node** number where you would like the change in position to start. All details on this element and the following elements are affected. A value of zero affects all elements.

Setting Up the Required Parameters

Home tab: **Auxiliary > Export to DXF File**

Instructs **PV Elite** to generate DXF files during an analysis run. Optionally, you can use **File > Import/Export > Export Vessel Geometry to R12 DXF File** to set this option. If the scale factor is not set, the **DXF Options** dialog appears prompting for the scale factor and any other necessary options. These options should be entered after the vessel has been completely modeled because the scaling factor is based on the overall height and length of the vessel. It is best to check the scaling factor at the conclusion of the data input and before the model is analyzed.

Create Default Border

Select to put a border around the drawing. The border style differs based on the border size. You can create your own border styles. The borders are located in the **PV Elite\System** folder. They are named ANSI_A.txt and so forth. These text files are essentially the core of ACAD Release 12 DXF files. See the user border creation section for more information.

Create Nozzle Schedule

Select to create a Nozzle Schedule. The nozzle schedule contains information pertaining to the size and thickness of nozzles, their mark number and the necessity of reinforcing.

Create Bill of Material

Select to generate a Bill of Material for the major components of the vessel, such as shells, heads, and conical sections.

Show Only OD Lines

Normally the DXF file will contain ID as well as OD lines for the major shell sections. If you do not want to see the ID lines, then check this box.

Show Dimensions

Select if you want tail dimensions for the major shell courses. The element diameters and thicknesses are shown in the BOM.

Insert Design Data and General Notes

Select if you want the software to include design data and general notes in the DXF file.

Drawing Size

Select A, B, C or D. Each size has a different style.

Scale Factor

Enter the scale factor. We recommend letting the program select this value by clicking **Compute and Insert Scale Factor**. We then recommend rounding up to the nearest typical scale factor.

User Border Creation

In order to do the following, you must use Windows Explorer, AutoCad, and Notepad.

1. Start AutoCAD and open your border. The border should be ANSI standard dimensions (8½ by 11, and so forth) scaled for the non-printable area of the paper.
2. After the border drawing is open, save it as a release 12 DXF file.
3. After the file has been saved it will be necessary to edit it with a text editor such as Notepad. Because the main drawing will have a DXF header, it will be necessary to delete the one in the border drawing. The DXF header ends on about line 960 with the word Entities. Delete through this line.
4. Next delete the last four lines in the file. This is the end of file marker.
5. Save the file with a txt extension.
6. Next rename the file in the **PV Elite\System** folder that you will be replacing. We suggest putting a new extension on it.
7. Save/Copy your border in the **PV Elite\System** folder and then rename it replacing our default border. You should now have new ANSI_?.txt file in the **PV Elite**\System subdirectory.

Review our border drawing text files before you start. Note that the border drawings must not contain any block attributes. These are not supported in our current implementation.

Create Database

Home tab: **Auxiliary > Create Database**

Creates a Microsoft Access database that contains all the input values.

Use **Review Database** in the **Input / Output** panel to review the database. For more information, see *Review Database* (page 213).

Create 3D PDF Files

Home tab: **Auxiliary > Create 3D PDF Files**

Instructs **PV Elite** to create or update the 3D PDF files during analysis. When PV Elite creates the 3D PDF files, the software activates the **Insert 2D Page** and **Insert 3D Page** options on the **Preview/Print** pane of the Output Processor.

Select this option and create the 3D PDF files prior to generating your final reports. If you make modifications to the geometry of your vessel after creating the 3D PDF files, select this option to turn it on prior to reanalyzing the vessel to update your 3D PDF files. After selecting reports from the Report List, click **Print to PDF** on the **Preview/Print** pane of the Output Processor and select **Insert 3D Page** to include the 3D model in the output report PDF. You can also click **Print to 3D PDF** on the **Preview/Print** pane of the Input Processor to print the 3D model as a separate PDF file.

NOTE You must have Adobe Acrobat version 11 (or higher) to utilize the 3D PDF feature.

Analyze Panel

The following commands are available on the **Analyze** panel on the **Home** tab.

	Analyze - Analyze the vessel and produce reports and forms. For more information, see *Analyze* (page 221).
	Error Check - Error check the vessel input. For more information, see *Error Check Only* (page 221).
	Review - Review the analysis data output for the vessel from the last analysis. For more information, see *Review Reports* (page 222).
	Review the DXF File - Opens the .dxf file of the model in any software installed on your computer system that is capable of viewing .dxf-formatted files. For more information, see *DXF File Generated by PV Elite During Runtime* (page 222).

Analyze

Home tab: **Analyze > Analyze**
Keyboard: **F12**

Analyzes the current model and creates the output files. Click reports in the **Report List** to see results of the analysis. For more information about the reports, see *Review Reports* (page 222) and *Output Processor* (page 544).

For more information about how PV Elite performs an analysis, see *PV Elite Analysis* (page 535).

TIP Although not required, we recommend that you run *Error Check Only* (page 221) before you run an analysis.

Error Check Only

Home tab: **Analyze > Error Check Only**

Runs only error checking on the model without running the analysis. Select **Warnings and Errors** in the **Report List** to see results of the error check.

Error Checking

The **Input Processor** makes many data consistency checks during the input session. For example, the processor creates an error message if you try to specify a nozzle 20 feet from the bottom of a 10-foot shell element. However, not all data can be confirmed on input so a general error processor is run prior to the analysis. This error processor can be run as a stand-alone from the **Analyze** panel, *Error Check Only* (page 221).

In addition to the notes that are presented on the screen during error checking, these error messages appear in the output report and are accessible through the output review processor.

NOTE As with all engineering and designing, the vessel analyst must use common sense to insure the model is basically correct. This is a great advantage of the 3D graphics as it reveals obvious errors.

Review Reports

Home tab: **Analyze > Review**

Displays the results of your analysis and output that results to a Microsoft Word file or an ASME Form using Microsoft Excel. For more information about reports, see *Output Processor* (page 544).

DXF File Generated by PV Elite During Runtime

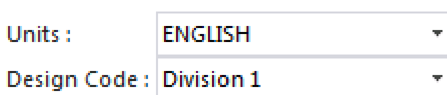
Home tab: **Analyze > Review the DXF File**

Opens the DXF file using drawing software installed on your computer that supports DXF files. If this command is available, the DXF file for this job was created during the last run. Clicking this command submits the file to Windows, which in turn launches your drawing software. If the input is altered, the analysis must be run in order to generate a new DXF file.

Use *Setting Up the Required Parameters* (page 219) to define the DXF settings to use.

Units/Code Panel

The following commands are available on the **Units/Code** panel on the **Home** tab.



Units

Selecting a units file converts all the properties in the model to that set of units. This can also be changed by **Tools > Select Units** (page 234).

Design Code

Determines which design code is used for analysis.

SECTION 5

Tools Tab

The following utility commands are available on the Tools tab:

	Set Configuration Parameters - Set configuration options for this analysis. For more information, see *Configuration* (page 224).
	Select Units - Select a new units file. For more information, see *Select Units* (page 234).
	Lock/Unlock the Input File - Protect the current input file from having modifications made to it. For more information, see *Lock/Unlock the Current Input File* (page 234).
	Create/Review Units - Review the current units file, or create a new units file. For more information, see *Create / Review Units* (page 234).
	Units Conversion - Open the PV Elite Units Conversion Utility that you can use to values from one unit of measure to another.
	Edit/Add Materials - Edit a materials database. For more information, see *Edit/Add Materials* (page 237).
	File Extraction Utility - Open the File Extraction Utility that you can use to extract input and output data files for compressed PV Elite files. For more information, see *File Extraction Utility* (page 247).
	Export to VUE Format - Open the VUE File Exporter Utility that you can use to convert PV Elite input files into VUE format, which can be used in Intergraph S3D applications. For more information, see *VUE File Exporter* (page 250).
	Calculator - Open the Windows *Calculator* (page 252).
	Renumber the Nodes - Resequence the From Node and To Node numbers of the elements in the vessel.
	Flip Model Orientation - Flip the orientation of the entire model. Use Flip Element Orientation on the Home tab to flip just a single element.
	Enter in U-1 Form Information for This Vessel - Create an ASME U-1 form for the current vessel. For more information, see *Enter in U-1 Form Information for This Vessel* (page 252).

	Compute Ligament Efficiencies - Calculate ligament efficiencies for tube spacing. For more information, see *Compute Ligament Efficiencies* (page 252).
	Display Driver - Select your display driver. If you then have display issues, try the other option.

Configuration

Tools tab: **Set Configuration Parameters**
Home tab: **Auxiliary > Set Configuration Parameters**

Sets job-specific parameters in the **Configuration** dialog box. Many parameters affect the analysis results. Always review the configuration at the start of a new job.

Topics

Job Specific Setup Parameters Tab (Configuration Dialog)

Print Water Volume in Gallons

Select to print out the element volume in gallons, instead of the default **Volume** cubic diameter unit. This command is independent of the selected units system.

Round Thickness to Nearest Nominal Size

Select to round thicknesses to the nearest 1/16 of an inch (if you are in English units) or the nearest 1mm (if you are in SI or MM units). The software increases the thickness of an element if you so specify in **Design Constraints Tab** (page 352) and the element thickness is inadequate.

Print Equations and Substitutions

Select to print formulas and substitutions for internal and external pressure calculations in analysis reports.

Increase Blind Flange Thickness for Reinforcement

Select to bypass reinforcement of a single opening of a flat end connection, according to Section VIII Division 1, paragraph UG-39(d)(2). This effectively increases the required thickness of the blind flange cover. This option can only be used if there is only one nozzle located in the blind flange.

Print Flange Calcs for External Pressure

Select to always print external pressure calculations on flanges in addition to the default internal pressure calculations. When this option is *not* selected, the software does not print external pressure calculations unless the required thickness generated by the external pressure exceeds the thickness generated by the internal pressure.

Disregard Bolt Space Correction Factor for ASME Flange Design

Select this option to avoid using the ASME Section VIII, Appendix 2 Equation (7) or the PD 5500 bolt space correction factors in the design of heat exchanger flanges and tubesheets, like Taylor-Forge. Using the correction factor is an industry-standard practice that is applied in other pressure vessel design codes. When the actual bolt spacing exceeds the allowable bolt spacing, the software multiplies the correction factor by the moment to design a thicker flange. In these cases, do *not* select this option.

Use ASME Code Case 2260/2261

Select to use modified equations to calculate the required thickness of elliptical/torispherical heads, as defined in ASME Section VIII, Division 1, Code Case 2260, May 20, 1998, Alternate Design Rules for Ellipsoidal and Torispherical Formed Heads. A thinner head is typically designed with these rules.

Use Eigen Solver

Select to use an Eigen solver for natural frequency calculations.

The natural frequency of a structure can be calculated using more than one method. When this option is not selected, the traditional method is to use the analysis technique of Freese or Rayleigh-Ritz. For a skirt-supported free standing vessel, this method provides acceptable results. When the support configuration is not a skirt/base type, such as legs, lugs or intermediate skirt, this method may not provide accurate results. By default, this option is selected and the Eigen solver is used.

The natural frequency Eigen solver uses numerical methods to solve the general equation of motion. Namely, the software solves the following matrix problem:

$$[[K] - w^2[M]]\{a\} = \{0\}$$

which for the general case is a set of *n* homogeneous (right-hand side equal to zero) algebraic system of linear equations with *n* unknown displacements a_i and an unknown parameter w^2. This is known as an Eigen problem. This iterative solution, for which not all a_i = 0, requires that the determinant of the matrix factor of *{a}* be equal to zero, in this case:

$$abs([[K] - w^2[M]]) = 0$$

After building stiffness *[K]* and mass *[M]* matrices of the model with appropriate boundary conditions (such as, anchors at skirts, bottom of legs, and at support lugs), the software can extract modes that are meaningful in the solution of the dynamics problem, particularly the modal response spectrum analysis. Using this generic-frequency Eigen solution method, the software can accurately extract modes of vibration for models that do not fit neatly into the cantilever beam model required for the Freese integration method. The natural frequency of the vessel is used in several of the wind and seismic codes.

NOTE If the selected earthquake code uses response spectrum, the software automatically uses the Eigen Solver, even if this option is not selected.

Use Pre-99 Addenda (ASME VIII-1 only)

Select if you are re-rating an older vessel to the pre-99 ASME addenda, and would like to use the older material allowables. As of January 2000, the 1999 addenda of ASME Division 1, Section 2, Part D is mandatory. The revision includes changes to the material properties of many materials, such as increases of allowable stresses in some ranges. By default, this option is not selected and the higher allowable stress database is used.

IMPORTANT Select this option before any vessel modeling occurs. For an existing file, you must access the material database for each existing element to update material properties. Other design codes are not affected.

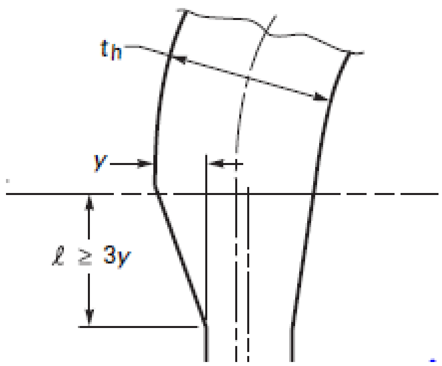
Use 2004 A-06 Addenda for Division 2

Select if the model is ASME Section VIII< Division 2, to use the older code rules in the analysis.

★ **IMPORTANT** Select *prior* to entering any vessel data.

For ASME VIII, Shell Head Joints are Tapered

Select if the shell to head junction is tapered as shown in the figure (ASME VIII-1 Figure UW-13.1).



For ASME VIII-1, Use Table G instead of exact equation for "A"

PV Elite can determine the strain factor ‘A’ used in ASME VIII-1 for external pressure calculations by either:

- Formula
- Table G in Section II Part D

Both selections yield a nearly identical result in most cases. However, PV Elite runs faster when using the formula option. The default is to use the data from Table G.

Use Nominal Head Thickness When Computing Longitudinal Stress

Select to use the nominal thickness of the head in the longitudinal stress calculation. Using this option effectively lowers the longitudinal stresses and increases the compressive allowable stress at the intersection of the skirt and bottom head.

By default, PV Elite uses the finished (minimum) thickness when determining longitudinal stresses on elliptical and torispherical heads due to pressure and bending moments. This is a conservative calculation of the longitudinal stress at the base of the straight flange.

No MDMT Calculations

Select to use the MDMT check in the analysis.

No MAWP Calculations

Select to use the MAWP check in the analysis.

Use Bolt Load Instead of Bolt Area Times Bolt Stress

Select to use the calculated value of the bolt load instead of the bolt area times its allowable stress in the design of annular baserings. This leads to a less conservative basering/bolting/chair cap thickness calculation.

Syntax Highlighting in Output Reports

Select to highlight failures and problem areas in data reports.

No Extended ASCII Characters in Output

Select to replace extended ASCII characters with multiple characters of the same value in equations. For example, ½ is replaced with 1/2. Some non-English versions of Windows do not display these characters correctly.

Metric Output is in Consistent Units

Select to allow the software to change units so that they are displayed consistently. For example, units of stress may be displayed in MPa and pressure in Bar. For meaningful output, these units should be the same.

Use ASME Code Case 2286

Select to use ASME Code Case 2286-1, a set of alternative rules for determining allowable external pressure and compressive stresses for cylinders, cones, spheres, and formed heads. This code case is applicable for ASME Section VIII Divisions 1 and 2. In the current ASME code rules, there are no provisions for computation of allowables when external pressure acts with bending and other compressive loads simultaneously.

These rules were first published in the late 1990's and reaffirmed on January 1, 2004. Review the code case before selecting this option. The following sections are covered in PV Elite:

2.2 Stress Reduction Factors
3.1 External Pressure
3.2 Uniform Axial Compression
3.2.1 Local Buckling
3.2.2 Column Buckling
3.3 Axial Compression due to Bending Moment
3.4 Shear (Allowable in Plane Shear Stress)
4.1.1 Allowable Circumferential Compression Stresses for Cones
4.1.3 Cone-Cylinder Junction Rings
4.2.1 Allowable Longitudinal and Bending Stresses for Cones
4.2.2 Unstiffened Cone-Cylinder Junctions
4.2.3 Cone-Cylinder Junction Rings
4.3.1 Allowable in Plane Shear Stress for Cones
5.1.1 Allowables for Uniform Axial Compression and Hoop Compression
5.1.2 Allowables for Uniform Axial Compression and Hoop Compression (Column Buckling)
5.2 Allowables for Axial Compression due to Bending Moment and Hoop Compression
5.3 Allowables for Hoop Compression and Shear
5.4 Allowables for Uniform Axial Compression, Axial Compression due to Bending Moment, Shear in the presence of Hoop Compression
5.4.1 Local Buckling (Section 5.4)
5.4.2 Column Buckling (Section 5.4)
5.5 Allowables for Uniform Axial Compression, Axial Compression due to Bending Moment, Shear in the absence of Hoop Compression
5.5.1 Local Buckling (Section 5.5)
5.5.2 Column Buckling (Section 5.5)
6.0 Sizing of Rings
6.1 Sizing of Small Rings
6.4 Local Stiffener Geometry Requirements

8.1.1 Spherical Shell with Equal Biaxial Stresses

Does the use of 2286 guarantee thinner required thicknesses and lower stresses due to combined loading?

No, it does not. The results are a function of the geometry. However, the results for allowable external pressure are generally higher than those computed using the typical UG-28 external pressure methods.

What happens during the analysis when this option is selected?

The software follows several steps in determining values that are needed during the 2286 calculations. One of these steps is to determine the lengths, such as LB1 and LB2. These are the distances between major lines of support, such as heads, body flanges and conical sections. The software does not distinguish large rings that act as bulkheads, according to paragraph 6.1(b). For cones, **Line of Support Options** in **General Input Tab** must be set to **Both Ends a Line of Support**. The **ASME Steel Stack** option in **Design Constraints Tab (page 352)** is not compatible with this option and is also ignored if both are checked.

The first calculations that appear for 2286 are in the **External Pressure Calculations** report, with the calculation of the allowable pressure at the given thickness. After completing this step, the software iterates to determine the required thickness for the given external pressure. If the element is a cylinder, the maximum length between stiffeners is calculated. These results are displayed in the summary at the end of the **External Pressure Calculations** report. Because Factor A and Factor B are not applicable, they are set to "No Calc".

After completing the external pressure calculations, the software calculates individual stresses for each stress category and summarizes them in the **Stress Due to Combined Loads** report. The allowable stresses are calculated for the combined loads (including external pressure). Unity checks according to 5.4 and 5.5 are made. The software also compares direct axial, shear and bending stresses to their respective allowables. If any of these are higher than the combined unity check, the maximum value is reported as the Unity Check.

A supplemental table in the **Stress Due to Combined Loads** report displays the results of λ_c, L and the calculated allowable stresses for each element for each load case. In some cases, the allowables may not be calculated, especially when there is no external pressure or when the load is only tensile. Some elements, such as welded flat heads and flanges, are not applicable and do not have any results. If this is the case, the printed stress value may read "No Calc" or may be blank.

If there any conical transitions in the model, the software calculates the necessity of junction rings and their requirements according to section 4.2.3. Cone-to-cylinder junction rings must satisfy inertia requirements from equations (4-1) and (4-5). The net area of the junction must be greater than or equal to the result of equation (4-4). The maximum distance from the cone-cylinder junction is defined in paragraph 4.2.3. The distance is defined as, "The nearest surface of the stiffening ring shall be located with a distance of T_r or 1 in., whichever is greater from the cone junction." T_r is the contact or stem width of the ring. Because the software handles arbitrary sections, the check value is one inch (25.4 mm). If the ring is farther than one inch from the junction, it is ignored.

External length 'L' includes Flange Thickness

Select to make the external length L for cylinders connected to weld neck flanges.

For ASME VIII-1, Compute "K" in corroded condition

The ‘K’ value is the stress concentration factor used in determining the required thickness of

Elliptical Heads. If the elliptical head has an internal corrosion allowance, the computed value of K decreases resulting in a lower required thickness. If that is what you want, select this option.

Use ASME Code Case 2695 (Div 1 allowables using Div 2 Part 4.5)

ASME Code Case 2695 allows the use of ASME VIII-2 formulas in a Division 1 design. In addition, the allowable stress values from Division 1 must also be used. Using the Code Case generally results in a more economical design, especially in the area of head thickness and nozzle reinforcement. As when using any Code Case, all of the restrictions must be understood so read the Code Case before using this option.

Moments Add at Lug Supports Due to Seismic Loads

Use Commas Instead of Decimals in Numbers

Select to use the comma character (,) as the decimal indicator in numbers. Clear this option to use the period character (.) as the decimal indicator. For example, 0,35 or 0.35.

Nozzle Analysis Directives

No Corrosion on Inside Welds

Select if inside nozzle welds do not corrode or you do not wish to remove the corrosion allowance when computing the area. By default, the software always corrodes the inner fillet weld when calculating the area available in the inside weld. The default method is the most conservative because the area under the weld is corroded in accordance with figure UG-37 of the ASME Code. This option is not valid when using PD 5500.

Use AD-540.2 sketch b and not sketch d for normal limit (pre 2007)

Select to use sketch (b) for the calculation of the vertical nozzle thickness limit, according to ASME Section VIII, Division 2, paragraph AD-540.2. Sketch (b) shows an integral connection with a smooth radius. Sketch (d) shows a similar geometry with an alternative pad plate and fillet. By default, the software uses sketch (d) to calculate the vertical thickness limit.

Compute Increased Nozzle Thickness

Select to calculate the minimum nozzle wall to account for external loading. In many cases pressure vessels are designed and built long before the piping system is attached to them. This means that the nozzle loadings are unknown. When this option is selected, then the minimum nozzle thickness *trn* is maximum of:

```
trn = (.134, trn for internal pressure) <= Nps 18
trn = (OD/150, trn for internal pressure) > Nps 18
```

Using this requirement in addition to UG-45 provides some additional metal to work with to satisfy thermal bending stresses in the nozzle. You can also specify the minimum wall thickness of the nozzle *trn* in **Nozzle**. If you do so, that vale overrides this calculation.

NOTE These formulae are not in the ASME Code. They are used in industry.

Compute and Print Areas for Small Nozzles

Select to calculate the nozzle reinforcement areas and MAWPs of small nozzles when the requirements of code paragraph UG-36 are met. UG-36 discusses the requirement of performing area replacement calculations when small nozzles are involved. Openings in vessels not subject to rapid fluctuations in pressure do not require reinforcement other than that inherent in the construction under the following conditions:

- 3.5 inch finished opening in a shell or head 0.375 inches required thickness or less
- 2.375 inch finished opening in a shell or head greater than 0.375 inches required thickness

If your geometry meets this criteria and this option is not selected, then the nozzle reinforcement areas and MAWPs are not calculated. If you require a single nozzle to be checked, place the text **#SN** in the nozzle description to force the software to calculate the areas for the selected small nozzle.

Compute Chord Length in Hillside Direction

Select to use the chord length to calculate the included angle for hillside nozzle calculations. By default, the software uses the actual length of removed material.

Compute Areas per PD 5500 3.5.4.9

Select to perform the pressure times area calculations according to PD 5000:2018, 3.5.4.9. The standard calculations according to design section 3 are always Calculated.

Nozzle opening MWAP is not restricted by the Shell (ASME)

When using ASME VIII-1 UG-37 for nozzle reinforcement, the MAWP of the opening is iteratively computed based on several items including UG-45, UG-37 areas, and other considerations. The MAWP of the opening is the minimum of the overall UG-37 calculation and the MAWP of the parent component to which the nozzle is attached. In some cases, you might want to know the MAWP of the junction without regard to the parent’s MAWP. If that is the case, select this option.

No B31.3 Stress Checks on Nozzles (ASME)

Select if ASME B31.3 analysis should not be performed for nozzle neck calculations.

NOTE If this option is selected, a note displays on the Nozzle Calcs output report(s), stating the software did not compute external and pressure loads per the user's request.

ASME MDMT Directives

ASME VIII-1 MDMT Option

Select the needed option.

Use the MAWP to Compute the MDMT

Select to use the MAWP for the vessel when determining the required thickness to use in the temperature reduction calculation according to UCS 66.1 (Div. 1), or Section 3.11.2.5 (Div. 2 2007 Edition or later). The MAWP also includes MAWPs determined during nozzle calculations. If this option is not selected, the design pressure on the element is used to determine the MDMT reduction for that element.

NOTES

- If the pressure specified on each element is the MAWP, do not select this option. Otherwise, the temperature reduction is conservatively low.
- For PD 5500, EN-13445, and Division 2 pre-2007, this option is ignored.

Do Not Use Nozzle MDMT Interpretation VIII-1-01-37

Select to control the MDMT calculation of the nozzle to shell junction, according to Section VIII, Division 1 (1998 Edition, 2000 Addenda), Figure UCS-66.l, Interpretation VIII-1-01-37, March 9, 2001. The interpretation states that if a nozzle neck with a nominal noncorroded thickness that is heavier than that of the shell is attached to the shell with a corner joint, then the shell becomes the governing thickness as defined in UCS-66(a)(1)(b). When evaluating the nozzle joint per UCS-66(b) of Section VIII, Division 1, the tr and tn thicknesses are those of the shell.

Reduce the MDMT due to Lower Membrane Stress

Select to calculate the reduced MDMT value per UCS 66.1. If this option is not selected, the software calculates the basic MDMT per UCS 66.

UCS-66.1 ratios below 0.35 compute MDMT's of -155F

UCS-67(c) MDMT's can be less that -55F

Consider Longitudinal Stress when computing MDMT's

Allowable Tower Deflection

Enter a value for the allowable vertical tower deflection, if the default of 6 inches per 100 feet does not meet your design specification.

Wind Shape Factor

Enter a value for the wind shape factor, if your design specification requires a specific value that does not correspond to the software-calculated value. For cylindrical structures, the value is typically 0.7.

Operating Nat. Freq. (Hz) Optional

Enter a value for the operating natural frequency. This is typically a value that you calculate separately when you want to use a method different than the Rayleigh method used by the software. In general, the Rayleigh method is suitable for most vessel designs, and you do not need to enter a value.

Empty Nat. Freq. (Hz) Optional

Enter a value for the empty natural frequency. This is typically a value that you calculate separately when you want to use a method different than the Rayleigh method used by the software. In general, the Rayleigh method is suitable for most vessel designs, and you do not need to enter a value.

Test Nat. Freq. (Hz) Optional

Enter a value for the test natural frequency. This is typically a value that you calculate separately when you want to use a method different than the Rayleigh method used by the software. In general, the Rayleigh method is suitable for most vessel designs, and you do not need to enter a value.

Set Material Impact Test Temperatures

Displays the **Set Impact Test Exemption Temperatures Dialog Box (page 232) where you** set the MDMT for impact tested materials, and select which tables and charts are considered when the software performs MDMT calculations.

Use Metric ASME Material Database (if available for the year chosen)

Select to indicate the software uses the metric version of the ASME Material Database. Specify the year of the material database in the **Material Database Year** box. The software defaults to using the current year if you do not specify a material database year.

Material Database Selection

Select the database the software uses for tables of allowable stress versus temperature.

This entry is only valid for Section VIII Division 1 and Section VIII Division 2, and is not available if **Use Pre-99 Addenda (Division 1 only)** is selected.

IMPORTANT Select this option before entering other data.

Metric Constant Selection

ASME Code Section VIII Division 1 contains a number of constants used for comparison in certain calculations. These values are presented in both Imperial and Metric units. An example would be the minimum thickness per UG-16(b) of 1/16" (1.5mm) or 3/32" (2.5mm). Note that the values in mm have been rounded. PV Elite can work with either depending on the selection made in this pull down.

- **Imperial** – Use the Imperial constants
- **Metric** – Use the Metric constants
- **Determine at runtime** – Have PV Elite determine which value to use depending on the set of currently selected units. For new files, the default is the **Determine at Runtime**.

Input Echo Language

Select the language to use for the Input Echo report. Make this selection before you analyze the model.

Set Impact Test Exemption Temperatures Dialog Box

Consider Table UG-84 in MDMT Calculations

Select to have the software consider the tables in UG-84 when determining the MDMT for the materials.

Consider UCS-66(g) in MDMT Calculations

Select to have the software calculate whether a material can be exempt from impact testing for a lower MDMT based on UCS-66(g).

Consider UCS-66(i) in MDMT Calculations

Select to have the software determine whether to calculate an MDMT colder than the input test temperature based on UCS-66(i).

Set MDMTs for Low Temperature Materials

Select to manually enter the MDMT for low temperature materials. Note that impact tested materials will appear in the selection. Temperatures for impact tested materials that are lower than allowed by specification can be entered here as well.

NOTE Press the refresh button to update the list of materials that might have been added to the model since the last time the dialog was visited.

DXF Options Tab (Configuration Dialog)

Create Default Border

Select to put a border around the drawing. The border style differs based on the border size. You can create your own border styles. The borders are located in the **PV Elite\System** folder. They are named ANSI_A.txt and so forth. These text files are essentially the core of ACAD Release 12 DXF files. See the user border creation section for more information.

Create Nozzle Schedule

Select to create a Nozzle Schedule. The nozzle schedule contains information pertaining to the size and thickness of nozzles, their mark number and the necessity of reinforcing.

Create Bill of Material

Select to generate a Bill of Material for the major components of the vessel, such as shells, heads, and conical sections.

Show Only OD Lines

Normally the DXF file will contain ID as well as OD lines for the major shell sections. If you do not want to see the ID lines, then check this box.

Show Dimensions

Select if you want tail dimensions for the major shell courses. The element diameters and thicknesses are shown in the BOM.

Insert Design Data and General Notes

Select if you want the software to include design data and general notes in the DXF file.

Drawing Size

Select A, B, C or D. Each size has a different style.

Scale Factor

Enter the scale factor. We recommend letting the program select this value by clicking **Compute and Insert Scale Factor**. We then recommend rounding up to the nearest typical scale factor.

Default Value Settings Tab (Configuration Dialog)

Sets the initial values for commonly used properties. PV Elite defaults fields using the values that you define here. You can edit the default value during creation.

You can set default values for the following areas of the software:

- Basic shell and head properties, such as: diameter, pressure details, and corrosion allowance.
- Material defaults for specific piping and nozzle standards used with vessels.
- Automatic advancing of details when adding new details to a model.
  When you add a detail to the model, such as for a nozzle or stiffening ring, PV Elite generates a detail description for that new detail (for example, Ring 1:1). When selected, the software does not generate a new description for any subsequently added details. The software copies the previous detail's description. If you select this configuration setting, you must give each detail a unique name when you add a new detail.
- Units setting defaults, including a specification to use metric units for all ASME jobs.
- Display foundation loads data on a separate report.
- Nozzle database defaults.
- Angle direction and orientation for exterior vessel details (such as nozzles, platforms and masses).
  Based on the drawing layout, you can specify which direction is the principle direction and whether the rotation is clockwise or counter-clockwise from that position. This option simplifies entering vessel details, regardless of the drawing convention system.

- Automatic save options, save folder locations, and help file locations.
- Specific folder for PV Elite template (.pvpt) files.
- Compression of input files.
- Update option frequency details.
- System folder default location.
- Font color and background color of the tooltip that displays
- Number of significant values to display in equations and substitutions on output reports.

Select Units

Tools tab: **Select Units**

Selects a new units file and changes the units system of the current job. For example, if your current job is in English units and you would like to change the units to millimeters, then use this option. After selecting a new units file, the current input values are converted into that set of units. To get a set of reports in the new units, run **Analyze (page 221)** again.

Delivered units files have the .fil extension and are in the C:\Users\Public\Public Documents\Intergraph CAS\PVELITE\2019\System folder. Many unit systems are delivered, such as English, MM, SI, Inches, and Newtons. Unicode systems are delivered for use in China, Japan, Taiwan, and Korea, where multibyte character sets are used.

Lock/Unlock the Current Input File

Tools tab: **Lock/Unlock the Current Input File**

Locks or unlocks the current input file. For example, if a file is unlocked, you can use this option to lock the file and create a read-only file. You cannot save changes made to a read-only file or analyze the file. If a file is locked, you can use this option to unlock the file and save changes made to the file or perform an analysis.

If you attempt to save changes or perform an analysis on a locked file, a warning message displays to inform you that the file is locked and is read-only. To save the changes you can unlock the file, or you can use the **Save As** function to save the changes under a new file name.

NOTE A newly created file must first be saved before it can be locked.

Create / Review Units

Tools tab: Create\Review Units

The **Create/Review Units File** utility allows you to create a new custom units file or edit an existing units file for use with PV Elite or CodeCalc. The utility is available on the **Tools** tab > **Create/Review Units**. You can also double-click **MakeUnit.exe** in the product delivery folder.

Delivered units files have the .fil extension and are in the C:\Users\Public\Public Documents\Intergraph CAS\PVELITE\2019\System folder. Many unit systems are delivered, such as English, MM, SI, Inches, and Newtons. Unicode systems are delivered for use in China, Japan, Taiwan, and Korea, where multibyte character sets are used. You can save new units files to the system folder or to another folder.

TIPS

- Use **Tools** tab > **Configuration** to specify the units file to use at startup.
- Use **Tools** tab > **Select Units** to select a new units file. The data in your job file is immediately converted to the new units.

***Units File Dialog Box* (page 236)**

What do you want to do?

- *Create a new units file* (page 235)
- *Edit an existing units file* (page 236)

Create a new units file

1. On the **Tools** tab, click **Create/Review Units** .

   *The units file dialog box displays. **Constant** has a default value of **1** for each type of unit.*

2. Do one of the following for each type of unit:
   - Select defined values for **Constant** *or* **User Unit**.
   - Type values for or **Constant** *and* **User Unit**.

3. Click **Save and Exit** .

   *The **Save As** dialog box displays.*

4. Select a folder path and type a file name.

5. Click **Save**.

   *The **Save As** dialog box and the **Units File** dialog box close.*

Edit an existing units file

1. On the **Tools** tab > click **Create/Review Units** .

   The units file dialog box displays.

2. Click **Open** .

   *The **Open** dialog box displays.*

3. Select an existing .fil units file and click **Open**.
4. Change unit types as needed by doing one of the following:
   - Select defined values for **Constant** *or* **User Unit**.
   - Type values for or **Constant** *and* **User Unit**.
5. Click **Save and Exit** .

   *The **Save As** dialog box displays.*

6. Select a folder path and type a file name. You can also use the same file name to replace the open file with the new unit values.
7. Click **Save**.

   *The **Save As** and **Units File** dialog boxes close.*

Units File Dialog Box

Specifies units and constants for a units file.

Name

Displays the type of unit, such as **Length**, **Area**, or **Pressure**.

System Unit

Displays the default system unit used as a multiplier for conversions, such as **feet**, **sq-inches**, and **psig**.

Constant

Select a defined conversion constant used as a multiplier for conversions, or type your own value.

User Unit

Select a defined unit for the conversion from the drop-down, or click in the box and type your own unit.

IMPORTANT

- If you select a defined **Constant**, the software changes **User Unit** to the correct unit. If you select a defined **User Unit**, the software changes **Constant** to the correct value.
- If you type your own value for **Constant** and **User Unit**, you must manually ensure that the combination provides the needed conversion.

Open

Open an existing units file for editing.

Save

Saves the units file.

Save and Exit

Saves the units file and closes the dialog box.

Help

Opens the help.

Edit/Add Materials

Tools tab: **Edit/Add Materials**

- **Edit Material**
- **Edit PD 5500 Material Database**
- **Edit EN-13445 User Material Database**

The **Material Database Editor** (page 237) utility opens. You can edit existing material and create new material in the selected material database. The new material does not affect the current job. To change an element or detail to the new material, open the **Material Database Dialog Box** (page 547) for each element or detail.

Material Database Editor

The **Material Database Editor** utility allows you to add custom materials to a delivered ASME, PD 5500, or EN 13445 material database for use with PV Elite or CodeCalc. For a YouTube demonstration, visit: http://www.youtube.com/watch?v=GEtIRO4PwCw. While the video is centered around PV Elite, it works much the same way in CodeCalc.

The utility is available from:

- **Tools** > **Edit/Add Materials**
- **MatEdit.exe**, found in the *[Program Folder]*\Intergraph CAS\PV Elite*[Release Number]* folder

When you use this utility, material database files with the .bin extension are created in the *[Program Folder]*\Intergraph CAS\PV Elite*[Release Number]*\System folder. These files contain only the custom materials you have added. The custom materials can then be merged into the main material databases.

NOTES

- The delivered databases contain allowed material for the current codes. You typically only add custom material if you are required to use an outdated material, or need to add material from a different code.
- Have the appropriate code available when adding new material. You will enter code-based material properties such as Chart Data, Material Band, and S Factor. The properties needed vary with the database that you are editing.

Material Properties **(page 239)**

What do you want to do?

- *Create a new custom material* (page 238)
- *Create a custom material based on an existing material* (page 238)

Create a new custom material

1. Click **Tools** > **Edit/Add Materials** and select the **ASME**, **PD 5500**, or **EN-13445** material database.
2. Click **Add** .

   *A new row named **New Material** appears in the grid of the **Material Database** view in the right pane.*
3. In the **Material Properties** view in the left pane, type values for the new material.

   IMPORTANT As you type values, check the **Stress vs. Temperature** graph in the right pane. Stress must not increase as temperature decreases.
4. Repeat these steps for each new material that you want to add.
5. Click **Save** to save the new material to a user database file.
6. Click **Merge** to add the user database to the material database of the software.

NOTE After merging, the custom material now appears at the bottom of the material database list for any command using the material database in PV Elite or CodeCalc.

Create a custom material based on an existing material

1. Click **Tools** > **Edit/Add Materials** and select the **ASME**, **PD 5500**, or **EN-13445** material database.
2. Click **Edit** .

   *The contents of the software database appear in the grid of the **Material Database** view in the right pane.*
3. Select a material for the **Material Database** grid.
4. Click **Select** and click **Yes** on the confirmation dialog box.

   *The copied material appears in a new row in the grid of the **Material Database** view.*
5. In the **Material Properties** view in the left pane, type new values as needed.

   IMPORTANT

   - You must change **Material Name** so that the name is unique in the user database *and* in the material database after merging.

- As you type values, check the **Stress vs. Temperature** graph in the right pane. Stress must not increase as temperature decreases.

6. Click **Save** to save the new material to a user database file.
7. Click **Merge** to add the user database to the material database of the software.

NOTE After merging, the custom material now appears at the bottom of the material database list for any command using the material database in PV Elite or CodeCalc.

Material Properties

The following code-based values are typically used as material properties.

Material Name

Type an allowed external pressure chart name. The software uses the chart name to calculate the B value for all external pressure and buckling calculations. If you type a valid value for **Material Name**, the software will look into its database and determine the external pressure chart name for this material and enter it into this cell. The program will also determine this chart name when you select a material name from the material selection window.

The following are the allowed external pressure chart names:

Carbon Steel

CS-1	Carbon and Low Alloy Sy<30000
CS-2	Carbon and Low Alloy Sy>30000
CS-3	Carbon and Low Alloy Sy<38000
CS-4	SA-537
CS-5	SA-508, SA-533, SA-541
CS-6	SA-562 or SA-620

Heat-Treated Steel

HT-1	SA-517 and SA-592 A, E, and F
HT-2	SA-508 Cl. 4a, SA-543,B,C

Stainless Steel (High Alloy)

HA-1	Type 304
HA-2	Type 316, 321, 347, 309, 310, 430B

HA-3	Type 304L
HA-4	Type 316L, 317L
HA-5	Alloy S31500

Non-Ferrous Material

NFA-1	AL3003, O and H112
NFA-2	AL3003, H20
NFA-3	AL3004, O and H112
NFA-4	AL3004, H34
NFA-5	AL5154, O and H112
NFA-6	C62000 (Aluminum Bronze)
NFA-7	AL1060, O
NFA-8	AL5052, O and H112
NFA-9	AL5080, O and H112
NFA-10	AL5456, O
NFA-11	AL5083, O and H112
NFA-12	AL6061, T6, T651, T6510 and T6511
NFA-13	AL6061, T4, T451, T4510 and T4511
NFA-20	AL5454, O and H112
NFC-1	Annealed Copper
NFC-2	Copper Silicon A and C
NFC-3	Annealed 90-10 Copper Nickel
NFC-4	Annealed 70-30 Copper Nickel
NFC-5	Welded Copper Iron Alloy Tube
NFC-6	SB-75 and SB-111 Copper Tube

NFN-1	Low Carbon Nickel
NFN-2	Ni
NFN-3	Ni Cu Alloy
NFN-4	Annealed Ni Cr Fe
NFN-5	Ni Mo Alloy B
NFN-6	Ni Mo Cr Fe
NFN-7	Ni Mo Cr Fe Cu
NFN-8	Ni Fe Cr Alloy 800
NFN-9	Ni Fe Cr Alloy 800H
NFN-10	Ni Moly Chrome Alloy N10276
NFN-11	Ni Cr Fe Mo Cu Alloys G and G-2
NFN-12	Cr Ni Fe Mo Cu Co, SB-462, 463, and so on.
NFN-13	Ni Fe Cr Si Alloy 330
NFN-20	Ni Cr Mo Grade C-4
NFN-15	Ni Mo Alloy X
NFN-16	Ni Mo Alloy B2
NFN-17	Ni Cr Mo Co N06625 (Alloy 625)
NFN-18	Ni Mo Cr Fe Cu (Grade G3)
NFN-19	Ni Mo Cr Fe Cu (Grade G3, >3/4)
NFN-20	Work Hardened Nickel
NFT-1	Unalloyed Titanium, Grade 1
NFT-2	Unalloyed Titanium, Grade 2
NFT-3	Titanium, Grade 1
NFZ-1	Zirconium, Alloy 702

NFZ-2	Zirconium, Alloy 705

Elastic Modulus Reference

The elastic modulus reference number is a value that points to or corresponds to a set of data set forth in ASME Section II Part D, tables TM-1, 2 and so on. Unfortunately, many materials have a composition or UNS number that does not match the criteria of what is supplied in the ASME Code. In these cases, the reference number will be brought in as zero. If this happens, you will need to enter in an appropriate value.

Thermal Expansion Coefficient Reference

Reference Number	Table	Description/UNS Number
1	TE-1	Carbon & Low Alloy Steels, Group 1
2	TE-1	Low Alloy Steels, Group 2
3	TE-1	5Cr-1Mo and 29Cr-7Ni-2Mo-N Steels
4	TE-1	9Cr-1Mo
5	TE-1	5Ni-¼4Mo
6	TE-1	8Ni and 9Ni
7	TE-1	12Cr,13Cr and 13Cr-4Ni Steels
8	TE-1	15Cr and 17Cr Steels
9	TE-1	27Cr Steels
10	TE-1	Austenitic Group 3 Steels
11	TE-1	Austenitic Group 4 Steels
12	TE-1	Ductile Cast Iron
13	TE-1	17Cr-4Ni-4Cu, Condition 1075
14	TE-1	17Cr-4Ni-4Cu, Condition 1150
15	TE-2	Aluminum Alloys
16	TE-3	Copper Alloys C1XXXX Series

Reference Number	Table	Description/UNS Number
17	TE-3	Bronze Alloys
18	TE-3	Brass Alloys
19	TE-3	70Cu-30Ni
20	TE-3	90Cu-10Ni
21	TE-4	N02200 and N02201
22	TE-4	N04400 and N04405
23	TE-4	N06002
24	TE-4	N06007
25	TE-4	N06022
26	TE-4	N06030
27	TE-4	N06045
28	TE-4	N06059
29	TE-4	N06230
30	TE-4	N06455
31	TE-4	N06600
32	TE-4	N06625
33	TE-4	N06690
34	TE-4	N07718
35	TE-4	N07750
36	TE-4	N08031
37	TE-4	N08330
38	TE-4	N08800,N08801,N08810,N08811
39	TE-4	N08825

Reference Number	Table	Description/UNS Number
40	TE-4	N10001
41	TE-4	N10003
42	TE-4	N10242
43	TE-4	N10276
44	TE-4	N10629
45	TE-4	N10665
46	TE-4	N10675
47	TE-4	N12160
48	TE-4	R20033
49	TE-5	Titanium Gr 1,2,3,7,11,12,16 and 17
50	TE-5	Titanium Grade 9
51	TEMA	5Cr-1/2Mo
52	TEMA	7Cr-1/2Mo & 9Cr-1Mo
53	TEMA	Ni-Mo (Alloy B)
54	TEMA	Nickel (Alloy 200)
55	TEMA	Copper-Silicon
56	TEMA	Admiralty
57	TEMA	Zirconium
58	TEMA	Cr-Ni-Fe-Mo-Cu-Cb (Alloy 20Cb)
59	TEMA	Tantalum
60	TEMA	Tantalum with 2.5% Tungsten
61	TEMA	17-19 CR (TP 439)
62	TEMA	AL-6XN

Reference Number	Table	Description/UNS Number
63	TEMA	2205 (S31803)
64	TEMA	3RE60 (S31500)
65	TEMA	7 MO (S32900)
66	TEMA	7 MO PLUS (S32950)
67	TEMA	AL 29-4-2
68	TEMA	SEA-CURE
69	TEMA	80-20 Cu-Ni (C71000)

Minimum Thickness (in.)

Type the minimum allowable thickness for the material. If the material has no minimum thickness, type **-1**.

Maximum Thickness (in.)

Type the maximum allowable thickness for the material. If the material has no maximum thickness, type **-1**.

Creep Temperature (F)

Type the temperature at which the material is governed by time dependent properties.

MDMT Exemption Temperature (F)

When the material uses an impact tested product specification, type the impact temperature. Otherwise, type **1**.

Product Form

Type an integer that designates the product form of the material.

Form Value	Product Form
1	Plate
2	Forgings
3	Seamless pipe
4	Welded pipe
5	Welded tube

6	Seamless tube
7	Bolting
8	Castings
9	Fittings
10	Seamless/welded pipe
11	Seamless/welded tube
12	reserved
13	Seamless pipe and tube
14	Pipe
15	Bar
16	Sheet
17	Tube
18	Forged pipe
19	Seamless/welded fitting
20	Drawn seamless tube
21	Condenser & heat exchanger tubes
22	Seamless extruded tube
23	Rod
24	Seamless and welded fittings
25	Welded fittings
26	Seamless fittings
27	Finned tube
28	Seamless U-bend tube
29	Welded condenser tube

Impact Reduction Temperature (F)

When the material is eligible for a -5ºF temperature reduction according to UCS-66(g), type **-5**. Otherwise, type **0.**

Material Band

The material band is used to determine the modulus of elasticity and coefficient of thermal expansion for that type of material.

Material Band	Basic Material Type/composition
M0	Carbon steel
M1	Carbon manganese steel
M2	Carbon molybdenum steel
M4	Low alloy MG Cr Mo V steel
M5	3.5Ni
M6	9Ni
M7	1-1.5Cr .5Mo
M8	.5Cr .5Mo .25V
M9	2.25Cr 1Mo
M10	5Cr .5Mo
M11	9Cr1Mo
M12	12Cr1Mo1V

File Extraction Utility

Input and output data files for each project in PV Elite, CodeCalc, and TANK, when saved, are compressed into a single file. The File Extraction utility allows you to extract these files. After extracting a file, you can view or open a specific input or output file directly from its folder location on the computer.

File Extraction Dialog Box (page 249)

What do you want to do?

- *Extract a file* (page 249)

Extract a file

1. In the **File Type** field, select the file type of the file(s) you want to extract.
2. In the **Selected Folder Name** field, enter the file path of the folder that contains the compressed files. Click the ellipses to open the **Browse for Folder** dialog box and select a file folder.
3. In the **Files to Extract** pane, select the file(s) to extract. Select the check box at the top of the pane to select all the files.
4. Click **Extract** to extract your files.

   The extracted file(s) display in the Extracted Files pane.
5. Click **Exit** to close the utility.
6. Open the extracted file from its location on the computer.

File Extraction Dialog Box

Select and extract compressed files from within the **File Extraction** dialog box.

File Type

Allows for selection of the type of file (by file extension, such as .cc, .pv, .tk) you want to locate.

Selected Folder Name

Displays the file path of the folder that contains the compressed files. Select the ellipsis to browse for a folder on the computer that contains compressed program files.

Files to Extract

Displays the compressed files.

Extracted File Name

Displays the files extracted from those in the **Files to Extract** pane.

Extract

Extracts the selected file(s) in the **Files to Extract** pane.

Exit

Exits the application.

VUE File Exporter

The VUE File Exporter utility allows you to export PV Elite and TANK files into VUE format. After exporting your files into VUE format, you can view your vessel or tank and add it to projects in SmartPlant Review.

***VUE File Exporter Dialog Box* (page 251)**

What do you want to do?

- *Export a File* (page 250)

Export a File

1. Click the **PV Elite and TANK File Folder** ellipsis**.**

   *The **Browse For Folder** dialog box displays.*

2. Select the file folder containing the PV Elite or TANK file(s) you want to export, and click **OK**.
3. Click the **VUE File Destination Folder** ellipsis.

   *The **Browse For Folder** dialog box displays.*

4. Select the file folder in which to place the exported VUE file(s), and click **OK**.
5. In the **List of Files** pane, select the check box of the file(s) to export. Select the check box at the top of the pane to select all the files.
6. *(Optional)* In the **Colors** pane, click the ellipsis in the detail row to select the color in which the detail will display in SmartPlant Review.
   - To apply a color to the entire model, click the ellipsis in the **Apply Color to Entire Model** row and select a new color in the **Color Editor** dialog.
   - Select **PV and TK Colors** to apply your element and detail color scheme from PV Elite or TANK to the converted models.
   - Select **S3D Colors** to apply your color scheme from Intergraph Smart™ 3D to the converted models.
7. Click **Export** to export the selected file(s).
8. Open the converted file from its location on the computer.

VUE File Exporter Dialog Box

Use the VUE File Exporter dialog to select the files in which to convert to VUE format. You can also select the colors for which the vessel elements and details will display in SmartPlant Review.

Icon Drop Down Menu

Contains **Recent Source** and **Recent Destination** options to select previously used file paths. The **Exit** option closes the application.

PV Elite and TANK File Folder

Displays the file path of the folder that contains the PV Elite or TANK files to be converted. Click the ellipsis to browse for a folder on the computer that contains PV Elite or TANK files.

VUE File Destination Folder

Displays the file path in which to place the converted PV Elite and TANK files. Click the ellipsis to browse for a folder on the computer in which to place the converted VUE files.

VUE File Destination Folder same as PV Elite and TANK File Folder

Select this check box to have the software place converted VUE files in the same folder as the original PV Elite or TANK files.

List of Files Pane

Displays the PV Elite or TANK files in the selected folder that are available to be converted. Upon successful completion of the export process, a green check mark displays in the **Status** column. If a file is not successfully exported, a red exclamation mark displays in the **Status** column.

Colors Pane

Displays a list of the various vessel or tank details and the colors in which they will display in SmartPlant Review. To change a detail color, click the ellipsis in the detail row and select a new color in the **Color Editor** dialog. To apply a color to the entire model, click the ellipsis in the **Apply Color to Entire Model** row and select a new color in the **Color Editor** dialog.

Select the **PV and TK Colors** option to apply the color scheme from PV Elite or TANK to the converted models. Select the **S3D Colors** option to apply the color scheme from Intergraph Smart™ 3D to the converted models. Select the **Reset All** option to apply the default color scheme of the VUE File Exporter utility.

Export

Select this option to export the selected PV Elite or TANK files.

Calculator

Tools tab: **Calculator**

Opens the Windows-supplied calculator utility. Use **Ctrl-C** and **Ctrl-V** to copy and paste values between the calculator and PV Elite.

Enter in U-1 Form Information for This Vessel

Tools tab: **Enter in U-1 Form Information for This Vessel**

Opens the **Additional Vessel Information** dialog box into which you enter additional information to produce an ASME U-1 form for the vessel. After analysis, an intermediate results file (.pvu) is created. This file is read by a Microsoft Excel macro when **Create ASME Form** is clicked in **Review Reports**, and the worksheet fields are populated with the calculated results. For more information, see *Review Reports* (page 222).

Compute Ligament Efficiencies

Tools tab: **Compute Ligament Efficiencies**

Opens the **ASME VIII-1 UG-53 Ligament Efficiency Scratch Pad** dialog box which is used to calculate ASME VIII-1 UG-53.x or 4.10.x ligament efficiencies for tube spacing.

Ligament Efficiency Calculation per Figure

Select one of the following:

- UG-53.1 or 4.10.1
- UG-53.2 or 4.10.2
- UG-53.3 or 4.10.3
- UG-53.4 or 4.10.4

and Figure

When **UG-53.4 or 4.10.4** is selected for **Ligament Efficiency Calculation per Figure**, select one of the following as the secondary figure:

- UG-53.5 or 4.10.5
- UG-53.6 or 4.10.6

Diameter of Tube Holes (d), **Longitudinal Pitch of Tube Holes (p)**, **Unit Length of Ligament (p1)**, **Diagonal Pitch of tube Holes (p' or p*)**, **Number of Tube Holes in Length p1 (n)**, and **Angle of Diagonal with Longitudinal line (theta)**

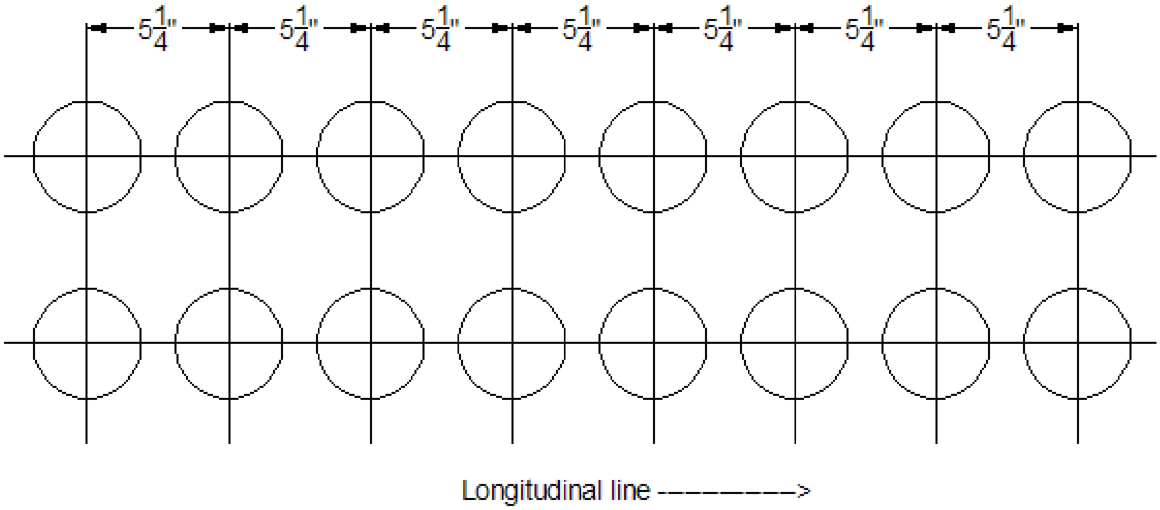
Based on the selected ligament efficiency figure, enter values for each dimension.

After all values are entered, the efficiencies are calculated and displayed as a percentage at the bottom of the dialog box. If there is an error, it is also displayed.

All figures are shown below.

UG-53.1 or 4.10.1

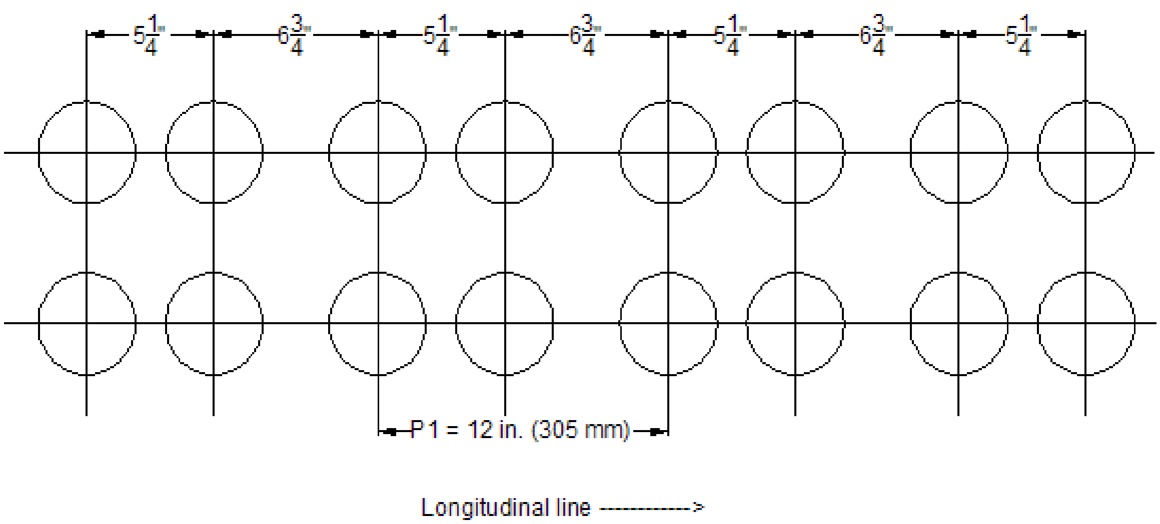
FIG-53.1 Example of Tube Spacing with Pitch of Holes Equal in Every Row



GENERAL NOTE: 5-1/4 in. = 133 mm

UG-53.2 or 4.10.2

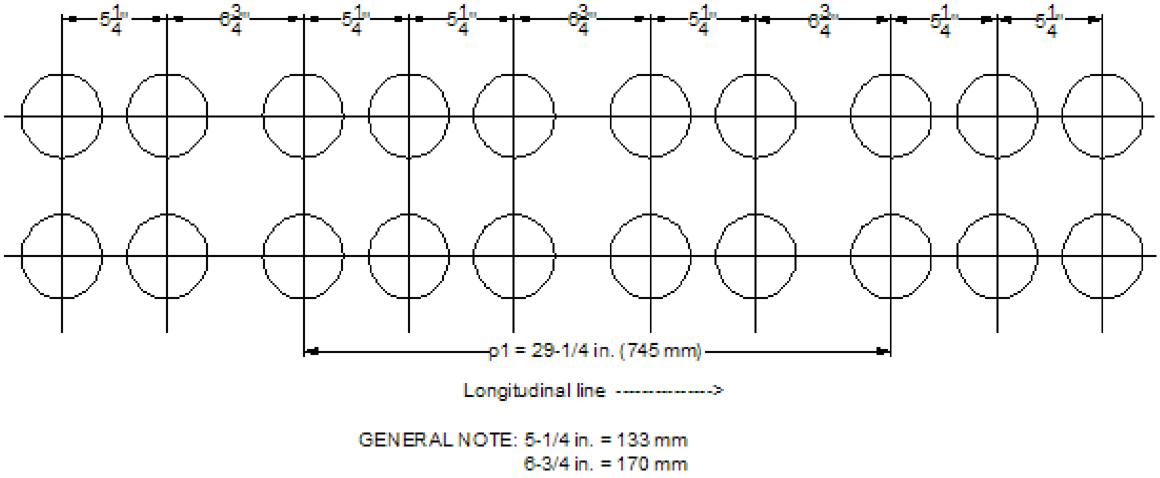
FIG-53.2 Example of Tube Spacing with Pitch of Holes Unequal in Every Second Row



GENERAL NOTE: 5-1/4 in. = 133 mm
6-3/4 in. = 170 mm

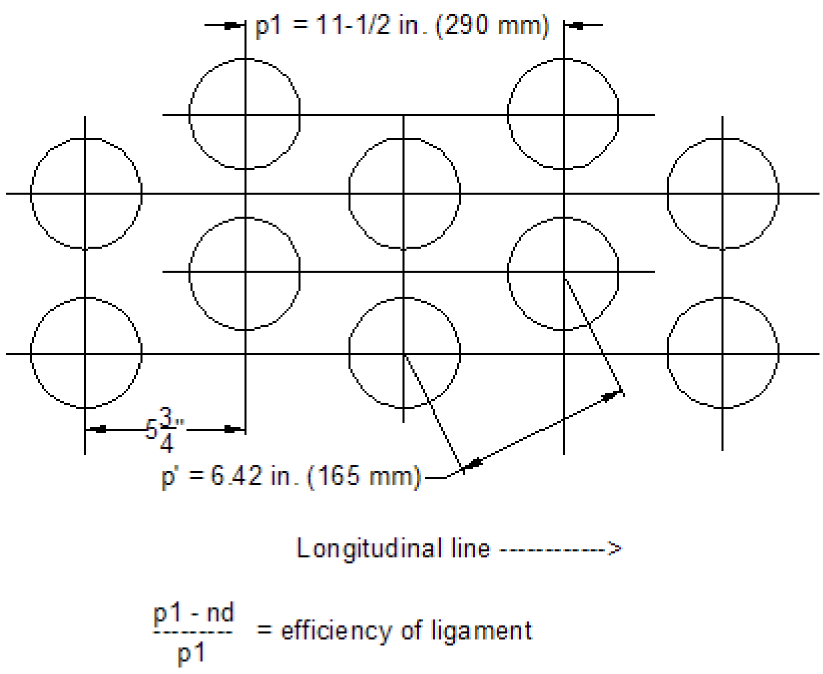
UG-53.3 or 4.10.3

FIG-53.3 Example of Tube Spacing with Pitch of Holes Varying in Every Second and Third Row



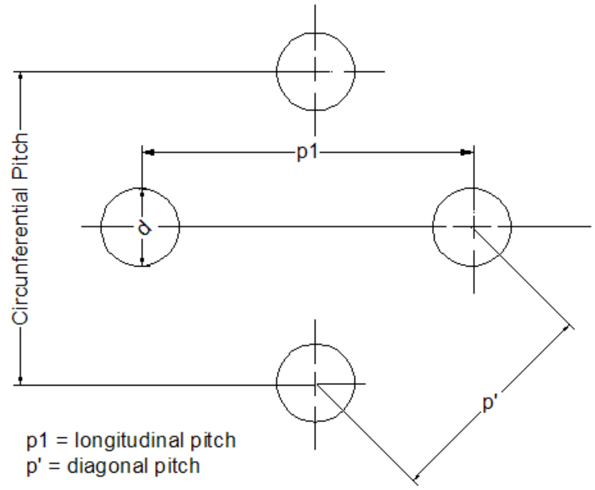
UG-53.4 or 4.10.4

FIG-53.4 Example of Tube Spacing with Pitch with Tube Holes on Diagonal Lines



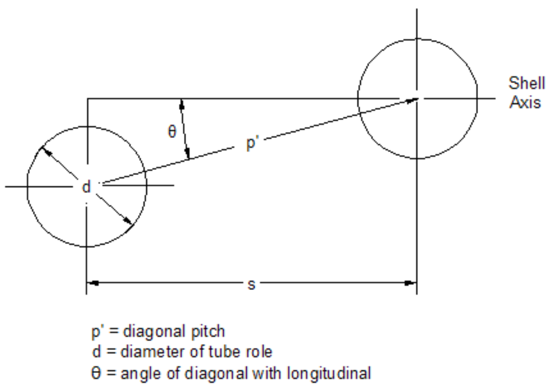
UG-53.5

FIG-53.5 Diagram for Determining the Efficiency of Longitudinal and Diagonal Ligaments Between Openings in Cylindrical Shells



UG-53.6

FIG-53.6 Diagram for Determining Equivalent Longitudinal Efficiency of Diagonal Ligaments Between Openings in Cylindrical Shells



SECTION 6

View Tab

The following commands are available on the **View** tab.

	Toggle Body Text - Click to display or hide the status bar at the bottom of the software window. The status bar displays information about the selected vessel element. 
	**Rigging Results** - Displays the calculated rigging results.
	**Reset Pane Layout** - Restores the software views to the default settings.

SECTION 7

3D Tab

The 3D tab contains commands that control the display of elements in the 3D View. The 3D View shows the actual vessel geometry in three dimensions. In addition to showing the outer surfaces, the model can also be viewed in wire frame and hidden line mode. Different shading modes such as Flat Shaded and Gourard are supported. Other operations, such as panning, zooming and model rotation are also supported.

Right-click anywhere on the 3D view to display the context menu. These same commands are also on the **3D** tab. The toolbar for performing some basic operations appears on the right side view. For more information, see *3D Graphics Toolbar* (page 258).

TIP When in 3D viewing mode, the dialog box for a detail can be activated by double-clicking a detail. It might be necessary to first select **Select by Single Click**.

	Visibility - Select and clear the different options to display and hide that element in the 3D graphics view. For example, you might want to hide the skirt to get a better view of a nozzle at the bottom of a vessel. The elements are not deleted from the model, just hidden.
	Wireframe - Shows lines and curves to show the object's borders and all its edges.
	Flat Shaded - Shows the objects with shaded surfaces and outlined edges.
	Gouraud Shaded - Shows the objects with shaded surfaces and smooth edges.
	Hidden Line - Shows the objects using lines and curves to show the object's borders.
	Orthographic View - Displays all objects the same relative size regardless of the distance from the observer.
	Perspective View - Specifies that the view should display the vessel using perspective rendering. Perspective rendering is characterized by: Objects being drawn smaller as their distance from the observer increases. The size of object dimensions along the line of sight are smaller relative to the dimensions across the line of sight.
	Delete Cutting Plane Grid - Removes a cutting plane from the view that you placed using the **Insert Cutting Plane** command on the 3D Graphics Toolbar.

	Fonts and Colors - Activates the Plot Properties dialog with which you can customize the colors of elements in the display.
	Fixed View - Turn on the fix the view.
	Shadow - Turns the vessel shadow on and off.
	Show Materials - Assigns a unique color for each material used in the vessel, and displays those colors in the view and legend grid.
	Show Wall Thickness - Assigns a unique color for each wall thickness value defined in the vessel, and displays those colors in the view and legend grid.
	Show Temperature - Assigns a unique color for each temperature value defined in the vessel, and displays those colors in the view and legend grid.
	Show Pressure - Assigns a unique color for each pressure value defined in the vessel, and displays those colors in the view and legend grid.

3D Graphics Toolbar

The 3D graphics toolbar controls how your model displays in the 3D View. By default, this toolbar displays vertically on the right side of the graphics window. You can toggle the 3D graphics toolbar off and on using the quick access toolbar customization command (black drop-arrow) in the top left-corner of the main window.

	Pre-defined Views - Changes the current view to front, back, top, bottom, left, right view or a standard isometric view.
	Zoom Extents - Resizes the model so that it fits in the current window.
	Zoom Window - Use the mouse to draw a window around the portion of the model that you want to zoom in on. This is a rubber band zoom. Alternately, spin the mouse wheel to zoom in and out.
	Orbit - Rotates the model in any direction using the mouse. Click the right mouse button and move the mouse to rotate the model.
	Turntable Orbit - Rotates the model about the Y-axis.

	Pan - Translates the model in the direction the mouse is dragged. Pressing the mouse wheel and holding it down while moving the mouse will also pan the model.
	Zoom Camera - Zooms in or out. Click this button, then press the left mouse button and move the mouse diagonally across the screen to zoom in or out. Alternately, spin the mouse wheel to zoom in and out.
	Select by Window - Selects details that are inside a fence that you define.
	Select By Click - Allows the selection of a detail for further manipulation.
	Translate Detail - Translates the selected detail in the view. NOTE This toolbar option is not available in the TANK product.
	Insert Cutting Plane - Inserts a cutting plane when you click this button and then click anywhere in the window. You can then rotate the cutting plane after it has been initiated. The rotating plane exposes the various layers of the vessel. The visibility of the cutting plane can then be turned off after the view is set. To restore the model, right- click in the 3D window and choose **Delete Cutting Plane**.
	Transparency - The main exterior shells of the model are transparent.
	Show Nozzle List - Displays list of nozzles in a list box. The list allows a nozzle to be located in the model for editing. NOTE This toolbar option is not available in the TANK product.
	Options - Element and detail colors are supported using the **Options** selection. After being set, the software recalls them in subsequent sessions. This option is also available when you right-click on the model window and select **Properties**. The **Options** dialog box displays as shown below. If any of the colors are changed, click **Apply** to update the new color selections.

SECTION 8

Diagnostics Tab

The following commands are available on the **Diagnostics** tab.

Crc Check - Performs a cyclic redundancy check on each of the delivered software DLLs and checks that executable files are correctly copied to the hard drive of your computer. Use this command if your software is behaving erratically.

Program Scanner - Checks the software build version of each executable file. See the *Intergraph web site www.coade.com/pvelite.htm* for the latest build information.

SECTION 9

ESL Tab

The following commands are available on the **ESL (External Software Lock)** tab.

	Show Data - Displays the encrypted data on your external software lock (ESL) key that allows you to check the status of the device. The data can also be saved to a log file. This information is useful for updating the software and for remaining current with your Intergraph license.
	Access Codes - Creates access codes needed to update the ESL when a new version of the software has been released.
	Authorization Codes - Allows ESL update codes to be entered. For more information, see *Authorization Codes* (page 262).
	Check Drivers- See Admin Control Center.
	Install Drivers- See Admin Control Center.
	Admin Control Center - Displays all information related to the HASP Driver. The **Sentinel Keys** option shows all available keys, whether local or on the network. The **Access Log** option displays all instances of a license being used on the network key on the host computer.

Phone Update

In general, each time a new version of PV Elite is released, data on the ESL must be updated to allow the new release to run properly. If the ESL is not updated, an error displays and software commands may not be active.

After installing the new version, use this option to generate four access codes. Phone, email, or fax the codes to Intergraph support. Updated codes are then provided that you enter using **Authorization Codes (page 262)** command.

NOTES

- The access codes constantly change.
- The time and date on the computer must be correct.

Authorization Codes

Enter one or more sets of update codes, usually to allow access to the latest software version. These codes can also change the client name, or change the last usage date.

Each set of codes received should be entered on a single line. Click **OK** to activate the new codes.

SECTION 10

Help Tab

The following commands are available on the **Help** tab.

	Help Topics - Opens the PV Elite help.
	Help Language - Allows you to select the language in which to view documentation; either English or Japanese.
	View - Opens the printable *PV Elite User's Guide, QA Manual,* and *QA certificate.*
	Quick Start - Opens the printable *PV Elite Quick Start.*
	Foundation 3D Help - Opens the help menu for all Foundation 3D related help topics.
	Check for Updates - Checks your version of the software against the most current version. You must be connected to the Internet for this option.
	Email Support - Creates an email with your system and software information. You can type your support question and send to support *ppmcrm@intergraph.com* (*mailto:ppmcrm@intergraph.com?subject=ICAS Support Request*). However, eCustomer is the best method to reach support. eCustomer provides a comprehensive knowledge base of information and allows Hexagon to track all customer queries, including bug reports, user issues, and user-generated ideas for improvement of the software.
	Online Registration - Registers this application. You must be connected to the Internet for this option. Hexagon does not give email addresses to third party solicitors. This information is solely used to inform customers regarding updates and release of the software.
	What's New - Opens a page in your default browser that describes the new features added in the most recent version of the software. **NOTE** For information about your software version, see the **Help** option on the **File** tab.

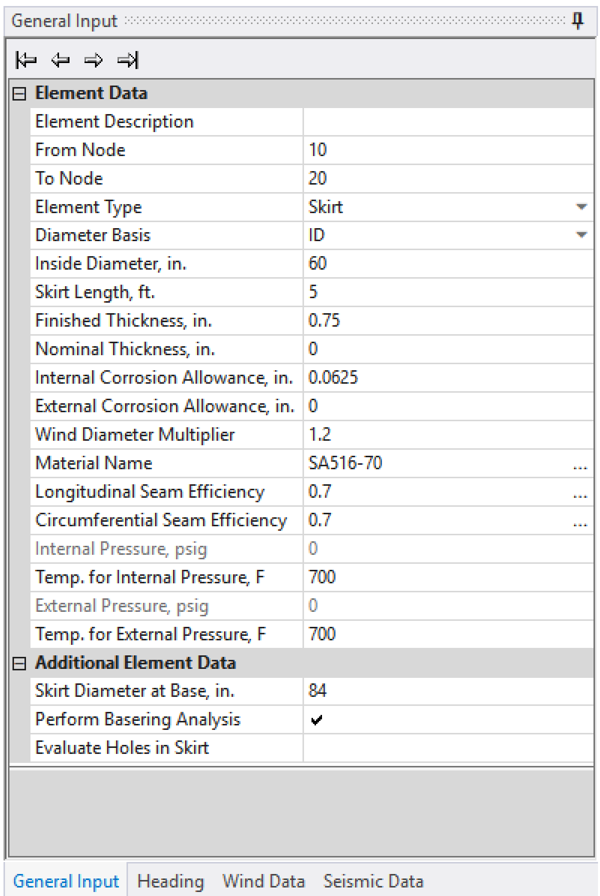
SECTION 11

General Input Tab

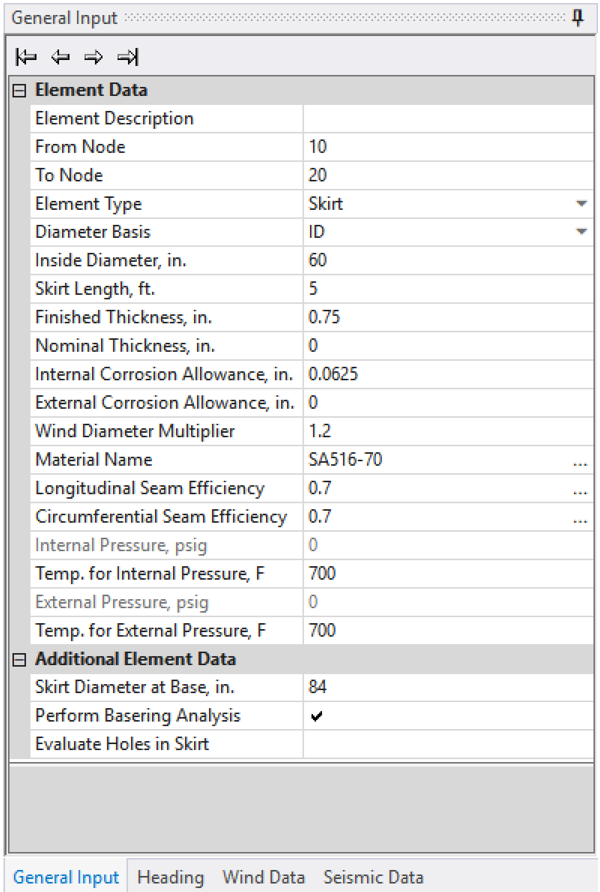
Select a vessel element in the graphics view to make it current, and then edit data for that element on the **General Input** tab.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > General Input** .

   *The **General Input** tab appears at the bottom-left of the PV Elite window.*



3. Use the element navigation arrows to navigate quickly between elements in the vessel.



In This Section

Element Data (General Input Tab)

All elements share a common set of parameters described below. A few of the parameters might be disabled depending on the active element.

Element Description

Enter a description for the element. The description can be up to 48 characters in length and can consist of both letters and numbers. The description is used in output reports. This entry is optional.

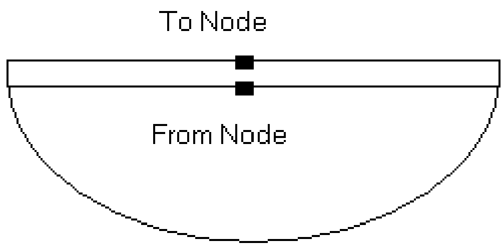
When descriptions are available, the software prints descriptions instead of node numbers. Descriptions are also seen in 3D graphics when you click **Flip to 3D View** on the **Auxiliary** toolbar.

NOTE This value is not a function of the selected vessel code (such as, PD 5500, EN or ASME).

From Node

Displays the software-generated node number describing the starting location of the element. The **From Node** value for this element is also used to define starting locations for details such as nozzles, insulation, and packing that are associated with this element.

The software defines a vertical vessel from bottom to top. If the vertical vessel is on skirt, the first element is the skirt. If it is on legs or lugs, the first element is a head and the legs or lugs are defined as details on the appropriate head or shell elements.



The software defines a horizontal vessel from left to right. The first element in a horizontal vessel is usually a head, and the support saddles are defined as details on the appropriate shell elements.

NOTE This value is not a function of the selected vessel code (such as, PD 5500, EN or ASME).

To Node

Displays the software-generated node number that describes the ending location of the element. The **To Node** value is incremented by 10 above the **From Node** value. **To Node** is **From Node** of the following element. For example, if the value of **To Node** for a head is 50, then the value of **From Node** for the shell is 50.

NOTE This value is not a function of the selected vessel code (such as, PD 5500, EN or ASME).

Element Type

Displays the type of the element. You can change the type by selecting one of the following from the list.

- **Cylinder** - A cylindrical shell
- **Elliptical** - An elliptical head
- **Torispherical** - A Torispherical (F&D) head
- **Spherical** - A spherical head
- **Conical** - A conical head or shell segment
- **Welded Flat** - CA welded flat head
- **Body Flange** - A body flange or blind flange
- **Skirt** - A skirt with an optional basering

CAUTION If the element type is changed, all detail data, such as nozzles, are lost.

NOTE This value is not a function of the selected vessel code (such as, PD 5500, EN or

ASME).

Diameter Basis

Select the type of diameter to use for the element. Select **ID** for the inside diameter. Select **OD** for the outside diameter. **ID** and **OD** are available for all design codes.

NOTES

- The ASME code provides different equations for required thickness based on whether the geometry is specified on **ID** or **OD**. By using the **ID** basis, the software computes a thinner required thickness, Tr, for the nozzle, such as in high-pressure, thick-wall geometries.
- If you are modeling a cylinder with welded flat heads on either end, and the welded flat heads sit just inside the cylinder shell, set **Diameter Basis** to **ID** and specify the **Inside Diameter** value on the welded flat heads to be the same size as the **Inside Diameter** of the cylinder. After you make these changes, if the flat head element still displays as sitting on the cylinder shell (instead of inside of the shell), select **Flip Orientation** twice. The software refreshes the model display to show the welded flat head inside the cylinder shell.

Inside Diameter

Enter a value for the inside diameter of the element, when **ID** is selected for **Diameter Basis:**

- Cylinders - Enter the diameter of the cylinder.
- Elliptical, torispherical and spherical heads - Enter the diameter of the straight flange.
- Skirts - Enter the diameter at the top of the skirt.
- Welded flat heads - Enter the large diameter of the flat head. The value is not a function of the selected code (such as, PD 5500, EN or ASME).

NOTE This option is not available when either **Body Flange** or **Conical is** selected for **Element Type**.

Outside Diameter

Enter a value for the outside diameter of the element, when **OD** is selected for **Diameter Basis:**

- Cylinders - Enter the diameter of the cylinder.
- Elliptical, torispherical and spherical heads - Enter the diameter of the straight flange.
- Skirts - Enter the diameter at the top of the skirt.
- Welded flat heads - Enter the large diameter of the flat head. The value is not a function of the selected code (such as, PD 5500, EN or ASME).

NOTE This option is not available when **Body Flange** or **Conical** are selected for **Element Type**.

"From" End Diameter

Enter a value for the inside or outside diameter at the **From Node** end of the cone, as needed by the selection for **Diameter Basis.** This option is only available when **Conical** is selected for **Element Type**.

Flange ID

Enter a value for the inside of a flange element. This option is only available when **Body Flange** is selected for **Element Type**

- Body flanges - Enter the inside diameter of the body flange.
- Blind flanges - Enter the inside diameter of the flange to which it is bolted. The inside diameter of a blind flange is zero, but the software uses this value to sketch the graphics.

Cylinder Length

Enter the distance between **From Node** and **To Node**. For a cylindrical shell, enter the length of the shell from seam to seam. This option is only available when **Cylinder** is selected for **Element Type**.

Straight Flange Length

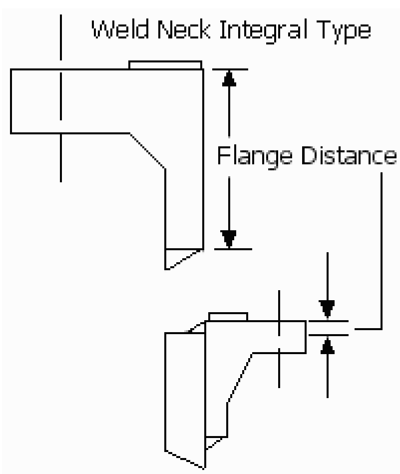
Enter the distance between **From Node** and **To Node**. For an elliptical, torispherical, or spherical head, enter the length of the straight flange. The software automatically includes the volume of the head and the depth of the head in calculations. This option is only available when **Elliptical**, **Torispherical**, or **Spherical** are selected for **Element Type**.

Overall Flange Length

Enter the distance between **From Node** and **To Node.**

For an integral weld neck type body flange, enter the through thickness of the flange including the weld neck, if any. If the flange is a slip-on, lap joint or similar, the distance is the length of the flange that protrudes past the shell to which it is attached. This value cannot be zero.

For a bolted blind flange, enter the thickness of the head/flange.



This option is only available when **Body Flange** is selected for **Element Type**.

Flat Head Thickness

Enter the distance between **From Node** and **To Node**. For a welded flat head, enter the thickness of the head/flange. The software also considers this value as the thickness of the weld between the flat head and attaching component, unless that thickness is specified in **Maximum Weld Thickness (Head or Junction)**.

This option is only available when **Welded Flat** is selected for **Element Type**.

Cone Length

Enter the distance between **From Node** and **To Node**.

For a conical head or shell segment, enter the length of the cone (including toriconical sections, if any) from seam to seam.

For a conical bottom head that is skirt-supported, enter the distance from the top of the skirt to the top of the conical head. You must also enter **Cone Length** in the **Additional Element Data** section. That is the length used in volume calculations and external pressure calculations.

This option is only available when **Conical** is selected for **Element Type**.

Skirt Length

Enter the distance between **From Node** and **To Node**. For a skirt support, enter the distance from the bottom of the base ring to the head tangent line. Because the software does not add the basering thickness to the element length, including the basering thickness in the skirt length provides the correct element elevations. This option is only available when **Skirt** is selected for **Element Type**.

Finished Thickness

Enter the finished thickness of the element. This is **Nominal Thickness** minus mill tolerance and thinning due to forming as described below. Finished thickness is a required input for each vessel element but you can allow PV Elite to increase the element thickness so that each element passes the requirements for internal pressure, external pressure, the combined loads of pressure, dead and live loads. Remember that the status bar lists internal pressure information about the current element including the required thickness.

For elliptical, torispherical and spherical heads, you may have to reduce the nominal thickness of the plate to account for thinning of the head due to forming.

For cylindrical shells made from pipe, you must subtract the maximum possible mill undertolerance from nominal pipe wall thickness. Use the pipe selection button to select standard pipe and insert the thickness into this field.

For welded flat heads this is simply the thickness of the plate from which the head is made.

For a skirt, this is typically the nominal thickness minus any mill undertolerance, and any thinning. For cylindrical skirts made from pipe, you will have to subtract the maximum possible mill undertolerance from the nominal pipe wall thickness.

For a body flange, this is the thickness of the flange. The ASME, EN and PD 5500 codes depict the thickness at the edge of the flange.

NOTE Do not include corrosion allowance. It is automatically subtracted from this thickness by the software when values are included for **Internal Corrosion Allowance** and **External Corrosion Allowance**.

Nominal Thickness

Enter the nominal (design) thickness of the element. For most calculations, the software uses **Finished Thickness** to determine MAWP and other results. However, when calculating element weight, it is theoretically more accurate to use the nominal thickness before the element it is formed. This entry is optional. If left blank, the software uses **Finished Thickness** to determine weight.

Normally, this value would only be applicable to formed heads, but the software allows this entry on all elements for greatest flexibility. When using **Nominal Thickness** to compute a result, the software always takes the greatest of the finished and nominal thicknesses. If the software designs the thickness, you are responsible for adjusting the value of nominal thickness before final calculations are made.

NOTES

- In May 2008, a change was made regarding the calculation of the inside diameter of a cylinder whose **Diameter Basis** is **OD**. In this case, the cylinder ID (usually pipe) is calculated as the nominal OD - 2 (Nominal wall thickness). This change affects only nozzle reinforcement calculations. If the diameter basis for the shell is specified as **ID**, there is no change.
- Use of **Nominal Thickness** applies to the selected vessel code (such as, PD 5500, EN or ASME).

Internal Corrosion Allowance

Enter the internal corrosion allowance for this element. Each dimension of the element (diameters and thicknesses) is modified by the corrosion allowance. Some elements in jacketed vessels may have both an internal and **External Corrosion Allowance**.

NOTE Use of **Internal Corrosion Allowance** applies to the selected vessel code (such as, PD 5500, EN 13445 or ASME).

External Corrosion Allowance

Enter the external corrosion allowance for this element. Most vessels do not have an external corrosion allowance specification, but some, such as jacketed vessels, need the consideration of an external corrosion allowance.

If an external corrosion allowance is specified, the software changes the dimension as needed. For example, the OD of a cylinder would be reduced by two times the external corrosion allowance for the external pressure calculation. For flanges, the large and small end hub dimensions are corroded in addition to the flange thickness. The external corrosion allowance is added to the final required thickness of the element.

NOTE Use of **External Corrosion Allowance** applies to the selected vessel code (such as, PD 5500, EN 13445 or ASME).

Wind Diameter Multiplier

Enter the wind load diameter multiplier. The value is multiplied by the element outside diameter in order to determine the overall diameter to be used in wind load calculations. The element outside diameter includes insulation.

When using a number greater than one, carefully account for the tributary area of external attachments such as nozzles, piping, or ladders. The typical multiplier used to determine wind load diameter is 1.2. Thus if the actual element OD is 50 inches, the overall wind load diameter for this element would be 50 * 1.2 = 60 inches.

The value of the wind load multiplier can be as low as 0. If a value of zero is used, then there will be no wind load on the element. This feature is useful when sections of vessels are not exposed to the wind.

When using the **Special Effective Wind Diameter Dialog Box (page 377)** dialog, you should set the value of the wind diameter multiplier to 1 for most models. For models in which the value is not 1, select **Calculate Effective Diameter for Wind Diameter Multipliers that are not 1 s**o the software accurately calculates the effective diameter.

NOTES

- If the element contains a platform, use **Platform Input** . The software automatically accounts for the load on the platform due to the wind pressure at the elevation of the platform. **Wind Diameter Multiplier** does not need to account for the platform.
- Use of **Wind Diameter Multiplier** applies to the selected vessel code (such as, PD 5500, EN-13445 or ASME).

Material Name

Enter the name of the material for this element. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, 5A and 5B. In addition, materials for PD 5500 and EN 13445 are included.

NOTES

- Alternatively, you can click **Select Material** on the **Utilities** toolbar to select a material directly from the **Material Database Dialog Box** (page 547).
- To modify the material properties of the selected element, click ... to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Longitudinal Seam Efficiency

Enter the efficiency of the welded joint for a shell section with welded longitudinal seams. This is the efficiency of the longitudinal seam in a cylindrical shell or any seam in a spherical shell. Elliptical and torispherical heads are typically seamless but may require a stress reduction which may be entered as a joint efficiency. Refer to Section VIII, Div. 1, **Table UW-12** (below) for help in determining this value. If you know the paragraph reference from UW-12, click ... to open the **Degree of Radiographic Examination and PWHT** dialog box and select the radiography and heat treatment for the joint. The longitudinal and circumferential joint efficiencies are then automatically selected by the software.

The joint efficiency in this (and all other) ASME code formulas is a measure of the inspection quality on the weld seam. In general, weld seams that receive full radiography have a joint efficiency of 1.0. Weld seams that receive spot radiography have a joint efficiency of 0.85. Weld seams that receive no radiography have a joint efficiency of 0.7. Seamless components usually have a joint efficiency of 1.0.

In addition to the basic rules described previously, the code requires that no two seams in the same vessel differ in joint efficiency by more than one category of radiography. For example, if circumferential seams receive no radiography (E = 0.7), then longitudinal seams have a maximum E of 0.85, even if they receive full radiography. In practice, circumferential seams, which are usually less highly stressed, may be spot radiographed (E = 0.85), while longitudinal seams are fully radiographed. This results in the same metal thickness at some savings in inspection costs.

For PD 5500 and Section VIII Div. II (pre-2007) this value is not used by the software except for skirt-to-shell welds. If Division II, 2007 edition or later is used, the joint efficiency will be 1 or 0.85 depending on the joint category, type of weld, and the extent of NDE. See section 4.3 and table 7.2 in Division 2 for more information.

Circumferential Seam Efficiency

Enter the efficiency of the welded joint for a shell section with welded circumferential seams. This is the efficiency of the circumferential seam in a cylindrical shell or any seam in a spherical shell. Elliptical and torispherical heads are typically seamless but may require a stress reduction which may be entered as a joint efficiency. Refer to Section VIII, Div. 1,

Table UW-12 (below) for help in determining this value. If you know the paragraph reference from UW-12, click ... to open the **Degree of Radiographic Examination and PWHT** dialog box and select the radiography and heat treatment for the joint. The longitudinal and circumferential joint efficiencies are then automatically selected by the software.

The joint efficiency in this (and all other) ASME code formulas is a measure of the inspection quality on the weld seam. In general, weld seams that receive full radiography have a joint efficiency of 1.0. Weld seams that receive spot radiography have a joint efficiency of 0.85. Weld seams that receive no radiography have a joint efficiency of 0.7. Seamless components usually have a joint efficiency of 1.0.

In addition to the basic rules described above, the code requires that no two seams in the same vessel differ in joint efficiency by more than one category of radiography. For example, if circumferential seams receive no radiography (E = 0.7), then longitudinal seams have a maximum E of 0.85, even if they receive full radiography. In practice, circumferential seams, which are usually less highly stressed, may be spot radiographed (E = 0.85), while longitudinal seams are fully radiographed. This results in the same metal thickness at some savings in inspection costs.

For PD 5500 and Section VIII Div. II (pre-2007) this value is not used by the software except for skirt-to-shell welds. If Division II, 2007 edition or later is used, the joint efficiency will be 1 or 0.85 depending on the joint category, type of weld, and the extent of NDE. See section 4.3 and table 7.2 in Division 2 for more information.

Table I: MAXIMUM ALLOWABLE JOINT EFFICIENCIES FOR ARC AND GAS WELDED JOINTS from Table UW-12

			Degree of Radiographic Examination			
Type No.	**Joint Description**	**Limitations**	**Weld Joint Category**	**(a) Full[2]**	**(a) Spot[3]**	**(c) None**
(1)	Butt joints as attained by double welding or by other means which will obtain the same quality of deposited weld metal on the inside and outside weld surfaces to agree with the requirements of UW-35. Welds using metal backing strips which remain in place are excluded.	None	A, B, C, D	1.00	0.85	0.70
(2)	Single welded butt joint with backing strip other than those included under (1)	(a) None except in (b) below (b) Circumferential butt joints with one plate offset; see UW-13(b)(4) and Fig. UW-13.1, sketch (k)	A, B, C, D A, B, C	0.90 0.90	0.80 0.80	0.65 0.65

Type No.	Joint Description	Limitations	Degree of Radiographic Examination Weld Joint Category	(a) Full[2]	(a) Spot[3]	(c) None
(3)	Single welded butt joint without use of backing strip.	Circumferential butt joints only, not over 5/16" in. (16 mm) thick and not over 24 in. (610 mm) outside diameter	A, B, C	NA	NA	0.60
(4)	Double full fillet lap joint	(a) Longitudinal joints not over 3/8 in. (10 mm) thick (b) Circumferential joints not over 5/8 in. (16 mm) thick	A B & C[6]	NA	NA	0.55
(5)	Single full fillet lap joints with plug welds conforming to UW-17	(a) Circumferential joints[4] for attachment of heads not over 24 in. (610 mm) outside diameter to shells not over 1.2 in. (13 mm) thick (b) Circumferential joints for the attachment to shells of jackets not over 5/8 in. (16 mm) in nominal thickness where the distance from the center of the plug weld to the edge of the plate is not less than 1 1/2 times the diameter of the hole for the plug	B	NA	NA	0.50

			Degree of Radiographic Examination			
Type No.	**Joint Description**	**Limitations**	**Weld Joint Category**	**(a) Full[2]**	**(a) Spot[3]**	**(c) None**
(6)	Single full fillet lap joints without plug welds	(a) For the attachment of heads convex to pressure to shells not over 5/8 in. (16 mm) required thickness, only with use of fillet weld in inside of the shell; or (b) for attachment of heads having pressure on either side, to shells not over 24 in. (610 mm) inside diameter and not over 1/4 in. (6 mm) required thickness with fillet weld on outside of head flange only	A, B A, B	NA	NA	0.45
(7)	Corner joints, full penetration, partial penetration, and/or fillet welded	As limited by Fig. UW-13.2 and Fig UW-16.1	C[7,] D[7]	NA	NA	NA
(8)	Angle joints	Design per U-2(g) for Category B and C joints	B, C, D	NA	NA	NA

Table Notes:
1 - The single factor shown for each combination of joint category and degree of radiographic examination replaces both the stress reduction factor and the joint efficiency factor considerations previously used in this Division
2 - See UW-12(a) and UW-51
3 - See UW-12(b) and UW-52
4 - Joints attaching hemispherical heads to shells are excluded
5 - E = 1.0 for butt joints in compression

			Degree of Radiographic Examination			
Type No.	**Joint Description**	**Limitations**	**Weld Joint Category**	**(a) Full[2]**	**(a) Spot[3]**	**(c) None**
6 - For Type No. 4 Category C joint, limitations not applicable for bolted flange connections.						
7 - There is no joint efficiency E in the design formulas of this Division for Category C and D joints. When needed, a value of E not greater than 1.00 may be used.						

Element is Post Weld Heat Treated

Select on the **Degree of Radiographic Examination and PWHT** dialog box if the element receives post-weld heat treatment. The code makes post weld heat treatment mandatory under certain conditions, such as lethal contents [UW-2] and thick materials [UCS-56]. In the case of carbon and low alloy steels (P1 to P10 materials) operating at low temperatures, the code gives credit if post-weld heat treatment is performed where it is not mandatory. Clause UCS-68 of the code discusses this in detail.

If the thickness ratio in Figure UCS-66.1 does not exceed 0.35, the code gives a temperature reduction credit of 30º F (16.66º C). The software determines the MDMT as follows:

a. Determine the minimum design metal temperature (MDMT) from figure UCS-66 using, the appropriate curve for the chosen material.

b. Determine the further reduction in temperature from figure UCS-66.1 where the calculated thickness is less than the actual part thickness.

c. If the ratio as determined by figure UCS-66.1 is less than 0.35, then the MDMT may then be reduced by a further 30º F (16.66º C).

For more information, see UW-2, UW-40. UCS-56, UCS-66 and UCS-68 of the code.

NOTE **Impact Testing (UG-84 and UCS-66)** - Under certain circumstances, the code makes impact testing of materials and weld metal mandatory. This is the case when the design temperature is lower than that determined using the code rules. The code provides rules in UCS-66 for determining the minimum design metal temperature (MDMT) of any part of the vessel. Where vessels are required to operate at a lower temperature than that calculated using the UCS-66 rules, then the materials must be impact tested according to UG-84. If the required energy levels are achieved for the impact test, then the vessel may be operated at the lower temperature.

You can enter the MDMT of a material for which you have performed impact testing at a temperature lower than that determined using the code rules. To enter the temperature for the material, go to the **Configuration** dialog box and click **Set Material Impact Test Temperatures.**

To include impact testing in your design, go to the **Materials Name** dialog box and check the box under "**Has this material been impact tested?**"

Internal Pressure

Enter the design internal pressure for the element. This pressure need not include any pressure due to liquid head, which is calculated automatically by the software when liquid is defined on this element.

Temperature for Internal Pressure

Enter the design temperature for internal pressure. This value is used as the metal design temperature for the element, especially in determining allowable stress values.

When you edit material properties by selecting a new material name, the program uses this temperature to determine the operating allowable stress value for the material. An exception is stiffening rings. The allowable stress for stiffeners uses **Temperature for External Pressure**.

NOTE For PD 5500 and EN 13445, this value must *never* be less than **Temperature for External Pressure**. The internal design temperature is the value that the software uses to determine the allowable stress for the element.

External Pressure

Enter the design pressure for external pressure analysis. This must be a positive value, such as 15 psig (0.103 MPa). If you enter a zero, the software determines the External MAWP, not the required thickness due to external pressure. For a skirt, zero is the only valid value, because there cannot be external pressure on a skirt.

External pressure definitions are the same for PD 5500, ASME and EN 13445. For example:

- 0 - No External Pressure
- 15 psig (0.1034 MPa) - Full Vacuum
- 0.3 psig - Partial Vacuum

NOTE If the element is an internal head, the external pressure equals the internal pressure on the previous element. If a different value is entered, the software will run, but prompts you with a warning message.

Temperature for External Pressure

Enter the design temperature for external pressure. This value is used as the metal design temperature for external pressure calculations. The software uses the external design temperature and **External Pressure Chart Name** on the **Material Properties Dialog Box** (page 611) to determine the allowable external pressure from the material tables.

When the software uses the materials tables to determine the allowable axial compression for a vessel, it uses the maximum of **Temperature for Internal Pressure** and **Temperature for External Pressure**. Axial compression may act in combination with either internal or external pressure unless **Vary Compressive Allowable for Internal/External Cases** is selected on the **Load Cases** tab.

NOTES

- For Divisions 1 and 2, this value must not exceed the maximum allowed by the external pressure charts in ASME Section II Part D.
- For PD 5500 and EN-13445, this value is not used to look up values for the allowable stress of the element. The internal temperature is used. If the vessel is a core or vacuum vessel and there is no internal pressure, **Temperature for Internal Pressure** should be set to the same value as **Temperature for External Pressure**.

Additional Element Data (General Input Tab)

Some element types require additional parameters.

Topics

Elliptical (Additional Element Data)

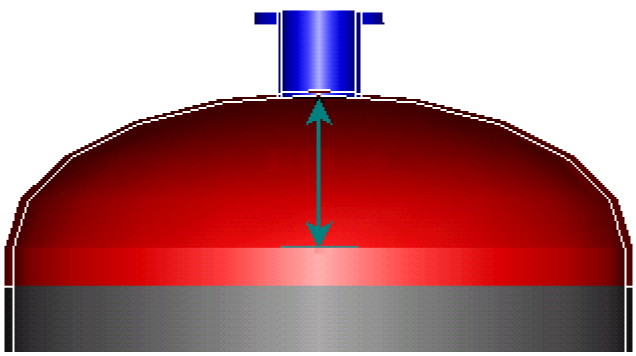
Defines additional data for elliptical head elements.

Head Factor

Enter the aspect ratio for the elliptical head. The aspect ratio is the ratio of the major axis to the minor axis for the ellipse. For a standard 2:1 elliptical head the aspect ratio is 2.0.

Inside Head Depth

Enter the inside depth of the elliptical head. This value is for a new head and does not include corrosion allowance. The software calculates the outer depth, *h*, and uses this term to calculate the required thickness of the ellipse. This option is only available for the PD 5500 code and you must select **Elliptical** for the **Element Type**.



Sump Head

Select if the head or flange is attached to the end of a nozzle. You must also select a **Parent Nozzle**. All of the nozzles on the vessel must be defined before the sump head. The best strategy is to completely define all other elements and details and then create the sump head element last. This option is only available when **Elliptical**, **Torispherical**, **Spherical**, or **Body Flange** are selected for **Element Type**.

Parent Nozzle

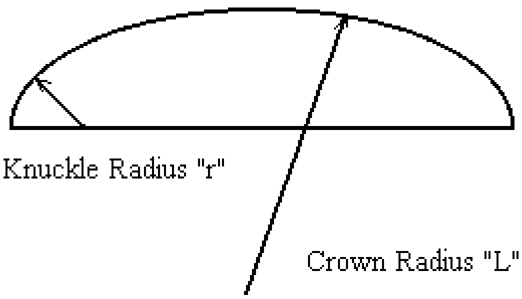
Select the type of nozzle for the sump head. Select **Inlet**, **Outlet**, **Shell Inlet**, or **Shell Outlet**. This option is only available when **Sump Head** is selected.

Torispherical (Additional Element Data)

Defines additional data for torispherical head elements.

Inside Crown Radius

Enter the crown radius *L* of the torispherical head. For a standard ASME flanged and dished head, this is equal to the outside diameter of the shell. See the ASME Code, Section VIII, Division 1, Appendix 1-4, figure 1-4(b). For PD 5500, this value is equal to the outside diameter measured to the tangent between crown and knuckle, as shown in Figure 3.5.2.1. For EN 13445 DIN 28013, this value is equal to the outside diameter times 0.8.



Inside Knuckle Radius

Enter the knuckle radius *r* for the toroidal portion of the torispherical head. For a standard ASME flanged and dished head, this is equal to six percent of the crown radius. Allowable values range from six percent of the crown radius to 100 percent of the crown radius (hemispherical head). See the ASME Code, Section VIII, Division 1, Appendix 1-4, figure 1-4(b). For EN 13445 DIN 28011, the knuckle radius is equal to 10% of the crown radius. For DIN 28013, the knuckle radius is the outside diameter times 0.154.

Sump Head

Select if the head or flange is attached to the end of a nozzle. You must also select a **Parent Nozzle**. All of the nozzles on the vessel must be defined before the sump head. The best strategy is to completely define all other elements and details and then create the sump head element last. This option is only available when **Elliptical**, **Torispherical**, **Spherical**, or **Body Flange** are selected for **Element Type**.

Parent Nozzle

Select the type of nozzle for the sump head. Select **Inlet**, **Outlet**, **Shell Inlet**, or **Shell Outlet**. This option is only available when **Sump Head** is selected.

Spherical (Additional Element Data)

Defines additional data for spherical head elements.

Sump Head

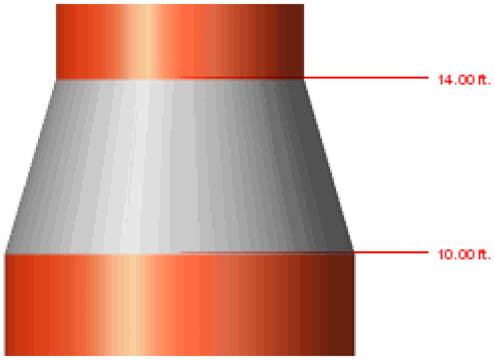
Select if the head or flange is attached to the end of a nozzle. You must also select a **Parent Nozzle**. All of the nozzles on the vessel must be defined before the sump head. The best strategy is to completely define all other elements and details and then create the sump head element last. This option is only available when **Elliptical**, **Torispherical**, **Spherical**, or **Body Flange** are selected for **Element Type**.

Parent Nozzle

Select the type of nozzle for the sump head. Select **Inlet**, **Outlet**, **Shell Inlet**, or **Shell Outlet**. This option is only available when **Sump Head** is selected.

Conical (Additional Element Data)

Defines additional data for conical head or shell elements.



'To' End Inside Diameter or **'To' End Outside Diameter**

Enter a value for the inside or outside diameter of the cone at the **To Node** end, as needed by the selection for **Diameter Basis.** For a conical head, either the **From Node or the To Node** has a diameter equal to zero or two times the small end knuckle radius. This is not the diameter at the point where a knuckle or flare intersects the conical section, but at the point where the knuckle or flare intersects the cylindrical section.

Cone Length

Enter the seam-to-seam or design length of the cone along the axis of the vessel. The software calculates the effective length of the cone for internal and external pressure calculations.

For cones without a knuckle or flared section, you must enter the design length of the cone. For cases where there is a knuckle or a flare, you must enter the seam-to seam-length. You can optionally enter in the **Half Apex Angle**.

Half Apex Angle

Enter a value if both **Cone Length** (in the **Element Data** section) and **Cone Length** (in the **Additional Element Data** section) are specified and you want to override the angle calculated by the software. If **0** is entered or no value is entered, the software uses the calculated value. This entry is optional.

Refer to the ASME code, Section VIII, Division 1, paragraph UG-33, figure UG-33.1 for a sketch of the half apex angle for some typical geometries. For internal pressure calculations, the half apex angle should not be greater than 30º, though the software will give results for up to 60º. For external pressure calculations, the angle should not be greater than 60º.

For cones without a knuckle, the software calculates discontinuity stresses according to the analysis technique by H. Bednar. For cones whose half apex angles are not within those limits prescribed by the code, this may help you decide if the geometry is acceptable. If the cone and attached cylinders do not have a common centerline, it may be necessary to calculate the greater of the angles and manually enter the value. For cones with a flare or knuckle, PV Elite calculates the half apex angle with the given seam-to-seam cone length. The overall cone length must include the knuckle dimensions.

Line of Support Options

Select a value to determine how the cone is taken as a line of support for external pressure calculations, according to ASME code Section VIII Division 1, Figure UG-28.1. Select one of the following:

- **Not a Line of Support** - The external pressure length for the surrounding sections includes the length of the cone.
- **Both Ends a Line of Support** - The external design length L does not include the cone length.
- **Small End a Line of Support** - The cone length up to small end is included.
- **Large End a Line of Support** - The cone length up to large end is included.

Based on the selection, it may not be necessary to perform moment of inertia calculations according to Appendix 1-8 (End is not a line of support).The code also does not force you to choose one option over the other; in some cases you may find that rings are not required at all. Optimize the design by experimenting with the options.

Has Flare or Knuckle?

Select if the cone is toriconical and has a flare at the small end or a knuckle at the large end. The **Toricone Dialog Box** (page 280) opens. See ASME code, Section VIII, Division 1, Paragraph UG-33, Figure UG-33.1 for an illustration of a toriconical section. This option is only available when **Conical** is selected for **Element Type**.

Is Concentric?

Select if the conical sections are concentric. Clear if the conical sections are eccentric. The software uses concentric sections in vertical geometries and eccentric conical sections in kettle type reboilers.

Shell Section

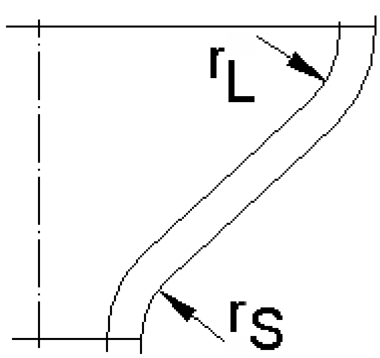
Select if the cone is a typical transition between two cylinders. Clear if this is a bottom or top conical head. This option applies to skirt-supported geometry.

Force Arl & Ars calculation (VIII-1)

For ASME VIII-1, select to force the Arl & Ars calculations when delta is greater than alpha when the cone end is attached to a flange. This option is only available when **Conical** is selected for **Element Type**.

Toricone Dialog Box

Defines knuckle data on a cone element. This dialog box opens when **Has Flare or Knuckle?** is selected.



Inside Knuckle Radius (rL)

Enter the bend radius of the toroidal knuckle at the large end. ASME Section VIII, Division 1, Paragraph UG-31(h) requires this radius to be no less than six percent of the outside diameter of the head, nor less than three times the knuckle thickness.

For ASME Section VIII Division 2 vessels, also choose the type of curvature of the large end knuckle: **Hemispherical**, **(2:1) Elliptical**, or **Torispherical**.

Knuckle Thickness

Enter the minimum thickness after forming the toroidal knuckle at the large end.

Crown Radius

When **Torispherical** is selected as the large end knuckle curvature, enter the radius of the torisphere crown. For standard geometry, click ... and the software calculates the value. ASME Section VIII, Division 1, Paragraph UG-31(h) requires this radius to be no less than six percent of the outside diameter of the head, nor less than three times the knuckle thickness.

Small end Knuckle Radius (rS)

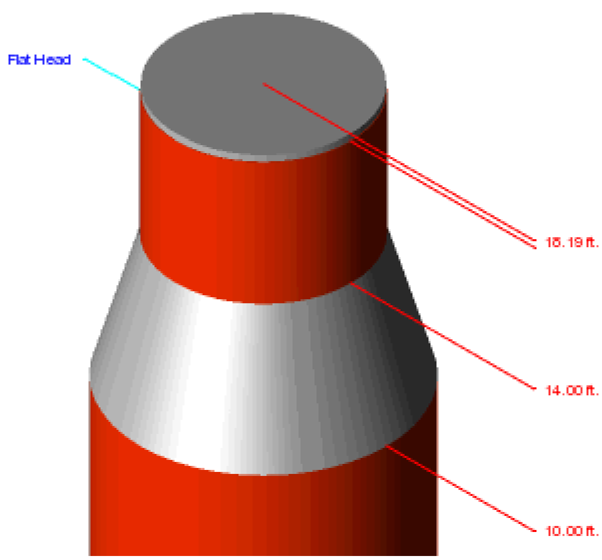
Enter the bend radius of the toroidal knuckle at the small end. ASME Section VIII, Division 1, Paragraph UG-31(h) requires this radius to be no less than six percent of the outside diameter of the head, nor less than three times the knuckle thickness.

Small End Knuckle Thickness

Enter the minimum thickness after forming the toroidal knuckle at the small end.

Welded Flat (Additional Element Data)

Defines additional data for welded flat head elements.



Welded Flat Head Attachment Sketch

Select the flat head attachment sketch, according to ASME code Section VIII, Division 1, Paragraph UG-34, Figure UG-34. Select from the following:

- **(a)** - Welded cover
- **(b-1)** - Head welded to vessel with generous radius
- **(b-2)** - Head welded to vessel with small radius
- **(c)** - Lap welded or brazed construction
- **(d)** - Integral flat circular heads
- **(e)**, **(f)**, **(g)** - Plate welded inside vessel (check 0.33*m)
- **(h)** - Plate welded to end of shell
- **(i)** - Plate welded to end of shell (check 0.33*m)
- **(m)**, **(n)**, **(o)** - Plate held in place by screwed ring
- **(q)** - Plate screwed into small diameter vessel
- **(r)**, **(s)** - Plate held in place by beveled edge

The sketch is for welded covers, not bolted blind covers.

Attachment Factor

Enter the welded flat head attachment factor for the sketch selected in **Welded Flat Head Attachment Sketch,** according to ASME code Section VIII, Division 1, Paragraph UG-34, Figure UG-34. Some typical attachment factors are below:

- **(b-1)** - 0.17 (Head welded to vessel with generous radius)
- **(b-2)** - 0.20 (Head welded to vessel with small radius)
- **(c)** - 0.20 (Lap welded or brazed construction)
- **(d)** - 0.13 (Integral flat circular heads)
- **(e)**, **(f)**, **(g)** - 0.20 (Plate welded inside vessel (check 0.33*m))
- **(h)** - 0.33 (Plate welded to end of shell)
- **(i)** - 0.20 (Plate welded to end of shell (check 0.33*m))
- **(j)**, **(k)** - 0.30 (Bolted flat heads (include bending moment))
- **(m)**, **(n)**, **(o)** - 0.30 (Plate held in place by screwed ring))
- **(p)** - 0.25 (Bolted flat head with full face gasket)
- **(q)** - 0.75 (Plate screwed into small diameter vessel)

IMPORTANT Consult Paragraph UG-34 before using these values.

NOTES

- For sketches involving *m*, the attachment factor (C) must be calculated and checked per the Code. The software does not calculate *m*.

For PD 5500, the attachment factor (C) is calculated according to 3.5.5.3 for welded flat ends. The attachment factor for welded flat heads is taken from figure 3.5-36 and is a function of the (cylinder actual thickness)/(cylinder required thickness) ratio and the pressure/allowable stress ratio. Enter the value from the figure 3.5-36 graph.

Non-Circular Small Diameter

Enter a value for the smaller dimension of a non-circular flat head. The large diameter is entered in **Inside Diameter** in the **Element Data** section**.** The software then calculates values such as required thickness using the formulas in the ASME code, Section VIII, Division 1, Paragraph UG-34. Enter a value of **0** if the flat head is circular.

Maximum Weld Thickness (Head or Junction)

Enter the maximum thickness of the weld between the flat head and attaching component and the diametral weld in the head. This value is used to determine whether post weld heat treatment (PWHT) is needed based on the thickness of the weld between the attaching component and the flat head. If no value is entered, the software considers the value in **Flat Head Thickness** as the thickness of the weld. This option is only available when **Welded Flat** is selected for **Element Type**.

App. 14 Large Central Opening

Select if there is a single, large, centrally-located opening that creates a (hole diameter)/(head diameter) ratio greater than 0.5. This opens the **Integral Flat Head with a Large Centrally Located Opening Dialog Box** (page 283). This option cannot be used with **Evaluate Uniform Patterned Holes?** and is only available when **Welded Flat** is selected for **Element Type**.

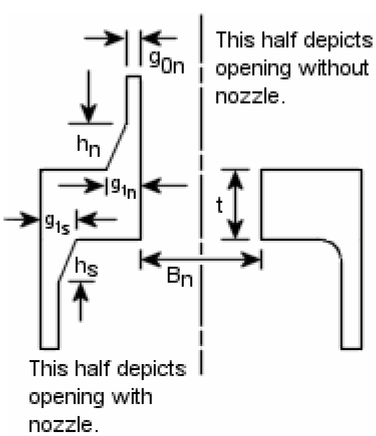
Evaluate Uniform Patterned Holes?

Select if there is a uniform series of holes in the flat head cover. The software then calculates the required thickness due to the ligament created by the hole pattern, according to ASME code Section VIII, Division 1, Paragraph UG-39. When selected, expand **Evaluate Uniform Patterned Holes?** and enter values for **Opening Diameter**, **Ligament Distance to Edge (U3)**, **Opening Diameter (d2)**, **Ligament Distance to Edge (U5)**, and **Pitch Distance (p).**

This option cannot be used when **App. 14 Large Central Opening** is selected. Nozzles are also not compatible and are not analyzed.

Integral Flat Head with a Large Centrally Located Opening Dialog Box

Defines values for calculating stresses and determining the required thickness of a welded flat cover with a large centrally located opening.



Opening Diameter Bn

Enter the diameter of the opening B_n centrally located in the welded flat head, as required by

ASME Section VIII, Division 1, Appendix 14 or ASME Section VIII, Division 2, Part 4.6.4. The diameter should also be greater than 1/2 of the welded flat head outside diameter. If your opening does not meet this criteria, do not use this analysis.

Large End Hub Thk (g1s)

Enter the hub thickness for the large end. The shell side dimensions must be entered in for the calculation to occur. Without proper values of G_{1s}, G_{0s} and h_s, the software does not calculate the required thickness of the head. Like the nozzle, the hub can be straight, and G_{1s} equals G_{0s}. The hub length is entered as stated for nozzles.

Small End Hub Thk (g0s)

Enter the hub thickness for the small end. The shell side dimensions must be entered in for the calculation to occur. Without proper values of G_{1s}, G_{0s} and h_s, the software does not calculate the required thickness of the head. Like the nozzle, the hub can be straight, and G_{1s} equals G_{0s}. The hub length is entered as stated for nozzles.

Hub Length (hs)

Enter the hub length. The shell side dimensions must be entered in for the calculation to occur. Without proper values of G_{1s}, G_{0s} and h_s, the software does not calculate the required thickness of the head. Like the nozzle, the hub can be straight, and G_{1s} equals G_{0s}. The hub length is entered as stated for nozzles.

Large End Hub Thk (g1n)

Enter the hub thickness for the large end. The nozzle side may or may not have a nozzle welded to the flat head. If there is no nozzle, the dimensions G_{1n}, G_{0n} and the hub length h_n are zero. If there is a nozzle, you must enter the hub dimensions as shown. For a straight neck, the hub dimensions are the same. If the hub dimensions are the same, the hub length is arbitrary, but cannot be zero. In this case one inch (25mm) is recommended.

Small End Hub Thk (g0n)

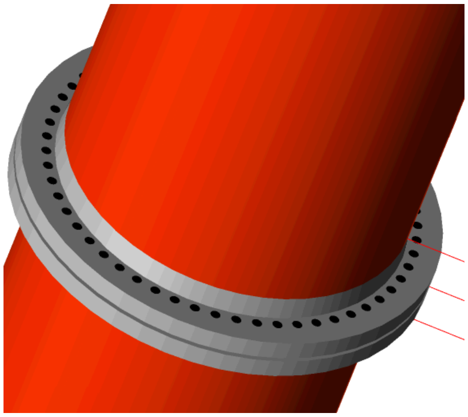
Enter the hub thickness for the small end. The nozzle side may or may not have a nozzle welded to the flat head. If there is no nozzle, the dimensions G_{1n}, G_{0n} and the hub length h_n are zero. If there is a nozzle, you must enter the hub dimensions as shown. For a straight neck, the hub dimensions are the same. If the hub dimensions are the same, the hub length is arbitrary, but cannot be zero. In this case one inch (25mm) is recommended.

Hub Length (hn)

Enter the hub length. The nozzle side may or may not have a nozzle welded to the flat head. If there is no nozzle, the dimensions G_{1n}, G_{0n} and the hub length h_n are zero. If there is a nozzle, you must enter the hub dimensions as shown. For a straight neck, the hub dimensions are the same. If the hub dimensions are the same, the hub length is arbitrary, but cannot be zero. In this case one inch (25mm) is recommended.

Body Flange (Additional Element Data)

Defines additional data for flange elements.



Perform Flange Calculation

Select to specify flange parameters and calculate the flange design, such as required thickness, MAWP, and MAPnc, according to the applicable design code. The **Flange Dialog Box** (page 303) opens. This option is only available when **Body Flange** is selected for **Element Type**.

Flange Weight

Enter the weight of the ANSI/ASME large diameter flange.

This option is only available when **Body Flange** is selected for **Element Type** and **Perform Flange Calculation** is not selected.

Approximate Weight (lbs.) of API 605 Flanges, Class Series B (According to Taylor Forge Cat. #571)

Nominal Size (in.)	75	150	300	400	600	900
26	80	120	400	360	550	1050
28	85	140	450	450	650	1520
30	90	150	550	530	810	1820
32	105	170	685	635	950	2065
34	110	210	750	690	1205	2450
36	145	240	840	855	1340	2520
38	160	290	915	935	1470	3385
40	170	310	990	1090	1630	3620

Nominal Size (in.)	75	150	300	400	600	900
42	185	345	1135	1190	2030	3960
44	230	370	1235	1375	2160	4300
46	245	435	1470	1525	2410	4640
48	270	480	1575	1790	2855	4980
50	290	520	1710	1950	3330	
52	310	550	1840	2125	3560	
54	340	620	1980	2565	3920	
56	400	650	2595	2710	4280	
58	430	780	2770	3230	4640	
60	475	850	2870	3820	5000	

Approximate Weight (lbs.) of MSS SP-44 Flanges, Class Series A (According to Taylor Forge Cat. #571)

Nominal Size (in.)	150	300	400	600	900
26	300	605	650	940	1525
28	345	745	785	1060	1810
30	400	870	905	1210	2120
32	505	1005	1065	1375	2545
34	540	1145	1200	1540	2970
36	640	1275	1340	1705	3395
38	720	695	935	1470	3385
40	775	840	1090	1630	3620
42	890	950	1190	2030	3960
44	990	1055	1375	2160	4300
46	1060	1235	1525	2410	4640
48	1185	1380	1790	2855	4980

Nominal Size (in.)	150	300	400	600	900
50	1270	1530	1950	3330	
52	1410	1660	2125	3560	
54	1585	2050	2565	3920	
56	1760	2155	2710	4280	
58	1915	2270	3230	4640	
60	2045	2470	3820	5000	

ANSI/DIN Class

Select the flange class from the pull down list. The software then finds the allowable pressure on the flange for both the operating and cold conditions of the selected class. This option is only available when **Body Flange** is selected for **Element Type** and **Perform Flange Calculation** is not selected.

ANSI/DIN Grade

Select the nozzle flange material grade. In the ANSI code, grades are assigned into groups.

★ IMPORTANT There are advisories on the use of certain material grades. Review the cautionary notes in the ANSI B16.5 code.

This option is only available when **Body Flange** is selected for **Element Type** and **Perform Flange Calculation** is not selected.

The following flange grades are available:

ANSI Groups

Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V 3½Ni C-Mn-Si	A 105 A 350 Gr.LF2 A 350 Gr. LF 6 Cl.1 A 350 Gr. LF3	A 216 Gr. WCB A 216 Gr. WCC A 352 Gr. LCC	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1

Group	Nominal Designation	Forgings	Castings	Plates
1.2	C-Mn-Si-V 2½Ni 3½Ni C-Si C-Mn-Si	A 350 Gr. LF 6 Cl.2	A 352 Gr. LC2 A 352 Gr. LC3 A 352 Gr. LCB	A 203 Gr. B A 203 Gr. E A 515 Gr. 65 A 516 Gr. 65
1.3	C-½Mo 2½Ni 3½Ni		A 217 Gr. WC1 A 352 Gr. LC1	A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-½Mo ½C-½Mo	A 182 Gr. F1 A 182 Gr. F2		A 204 Gr. A A 204 Gr. B
1.7	Ni-½Cr-½Mo ¾Ni-¾Cr-1Mo		A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1¼Cr-½Mo 1¼Cr-½Mo-Si	A 182 Gr. F22 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	2¼Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.11	Cr-½Mo			A 204 Gr. C
1.13	5Cr-½Mo	A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
1.17	1Cr-½Mo 5Cr-½Mo			

Group	Nominal Designation	Forgings	Castings	Plates
2.1	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304 A 182 Gr. F304H A 182 Gr. F316	A 351 Gr. CF3 A 351 Gr. CF8 A 351 Gr. CF3M	A 240 Gr. 304 A 240 Gr. 304H A 240 Gr. 316
2.2	18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316H A 182 Gr. F317	A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti 18Cr-10Ni-Cb	A 182 Gr. F321 A 182 Gr. F321H A 182 Gr. F347		A 240 Gr. 321 A 240 Gr. 321H A 240 Gr. 347
2.5		A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H		A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	23Cr-12Ni			A 240 Gr. 309H
2.7	25Cr-20Ni 20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N	A 182 Gr. F310 A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53	A 351 Gr. CK3MCuN	A 240 Gr. 310H A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750
2.8	24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F55	A 351 Gr. CE8MN A 351 Gr. CD4MCu A 351 Gr. CD3MWCuN	A 240 Gr. S32760
2.9	23Cr-12Ni 25Cr-20Ni			A 240 Gr. 309S A 240 Gr.310S
2.10	25Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	

Group	Nominal Designation	Forgings	Castings	Plates
2.11	18Cr-10Ni-Cb		A 351 Gr. CF8C	
2.12	25Cr-20Ni		A 351 Gr. CK20	
3.1	35Ni-35Fe-20Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe 64Ni-29.5Mo-2Cr-2Fe-Mn-W 54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe	B 462 Gr. N10665 B 462 Gr. N10675 B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001		B 333 Gr. N10665 B 333 Gr. N10675 B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001
3.8	70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu 55Ni-21Cr-13.5Mo 55Ni-23Cr-16Mo01.6Cu	B 573 Gr. N10003 B 574 Gr. 06455 B 564 Gr. N08825 B 462 Gr. N06022 B 462 Gr.N06200		B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. 08825 B 575 Gr. N06022 B 575 Gr. N06200

Group	Nominal Designation	Forgings	Castings	Plates
3.9	47Ni-22Cr-9Mo-I8Fe	B 572 Gr. N06002		B 435 Gr. N06002
3.10	25Ni-46Fe-21Cr-5Mo	B 672 Gr. N08700		B 599 Gr. N08700
3.11	44Fe-25Ni-21Cr-Mo 26Ni-43Fe-22Cr-5Mo	B 649 Gr. N08904 B 621 Gr. N08320		B 625 Gr. N08904 B 620 Gr. N08320
3.12	47Ni-22Cr-20Fe-7Mo 46Fe-24Ni-21Cr-6Mo-Cu-N	B 581 Gr. N06985 B 462 Gr. N08367	A 351 Gr. CN3MN	B 582 Gr. N06985 B 688 Gr. N08367
3.13	47Ni-22Cr-19Fe-6Mo Ni-Fe-Cr-Mo-Cu-Low C	B 581 Gr. N06975 B 462 Gr. N08031		B 582 Gr. N06975 B 625 Gr. N08031
3.14	47Ni-22Cr-19Fe-6Mo 40Ni-29Cr-15Fe-5Mo	B 581 Gr. N06007 B 462 Gr. N06030		B 582 Gr. N06007 B 582 Gr. N06030
3.15	33Ni-42Fe-21Cr	B 564 Gr. N08810		B 409 Gr. N08810
3.16	35Ni-19Cr-1¼Si	B 511 Gr. N08330		B 536 Gr. N08330
3.17	29Ni-20.5Cr-3.5Cu-2.5Mo		A 351 Gr. CN7M	

DIN Grades/Groups - Forgings and Flat Products

Group	Forgings			Flat Products		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	—	—	—	—	—	—
3E0	—	—	—	P235GH	EN 10028-2	1.0345

Group	Forgings			Flat Products		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
3E0	—	—	—	—	—	—
3E0	P245GH	EN 10222-2	1.0352	P265GH	EN 10028-2	1.0425
3E1	P280GH	EN 10222-2	1.0426	P295GH	EN 10028-2	1.0481
4E0	16Mo3	EN 10222-2	1.5415	16Mo3	EN 10028-2	1.5415
5E0	13CrMo4-5	EN 10222-2	1.7335	13CrMo4-5	EN 10028-2	1.7335
6E0	11CrMo9-10	EN 10222-2	1.7383	12CrMo9-10	EN 10028-2	1.7375
	-	-	-	10CrMo9-10	EN 10028-2	1.7380
6E1	X16CrMo5-1	EN 10222-2	1.7366	—	—	—
7E0	—	—	—	P275NL1	EN 10028-3	1.0488
	—	—	—	P275NL2	EN 10028-3	1.1104
7E1	—	—	—	P355NL1	EN 10028-3	1.0566
	—	—	—	P355NL2	EN 10028-3	1.1106
7E2	15NiMn6	EN 10222-3	1.6228	15NiMn6	EN 10028-4	1.6228
	—	—	—	11MnNi5-3	EN 10028-4	1.6212
	13MnNi6-3	EN 10222-3	1.6217	13MnNi6-3	EN 10028-4	1.6217
7E3	—	—	—			
	12Ni14	EN 10222-3	1.5637	12Ni14	EN 10028-4	1.5637

Group	Forgings			Flat Products		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
	X12Ni5	EN 10222-3	1.5680	X12 Ni 5	EN 10028-4	1.5680
	X8Ni9	EN 10222-3	1.5662	X8Ni9	EN 10028-4	1.5662
8E0	—	—	—	—	—	—
8E2	P285NH	EN 10222-4	1.0477	P275NH	EN 10028-3	1.0487
	P285QH	EN 10222-4	1.0478	—	—	—
8E3	P355NH	EN 10222-4	1.0565	P355N	EN 10028-3	1.0562
	P355QH1	EN 10222-4	1.0571	P355NH	EN 10028-3	1.0565
9E0	X20CrMoV11-1	EN 10222-2	1.4922	—	—	1.0565
9E1	X10CrMoVNb9-1	EN 10222-2	1.4903	X10CrMoVNb9-	EN 10028-2	1.4903
10E0	X2CrNi18-9	EN 10222-5	1.4307	X2CrNi18-9	EN 10028-7	1.4307
	—	—	—	X2CrNi19-11	EN 10028-7	1.4306
10E0	—	—	—	X1CrNi25-21	EN 10028-7	1.4335
10E1	X2CrNiN18-10	EN 10222-5	1.4311	X2CrNiN18-10	EN 10028-7	1.4311
11E0	X5CrNi18-10	EN 10222-5	1.4301	X5CrNi18-10	EN 10028-7	1.4301
	X6CrNi18-10	EN 10222-5	1.4948	X6CrNi18-10	EN 10028-7	1.4948

Group	Forgings			Flat Products		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	—	—	—	—	—	—
12E0	X6CrNiTi18-10	EN 10222-5	1.4541	X6CrNiTi18-10	EN 10028-7	1.4541
	X6CrNiNb18-10	EN 10222-5	1.4550	X6CrNiNb18-10	EN 10028-7	1.4550
	X6CrNiTiB18-10	EN 10222-5	1.4941	X6CrNiTiB18-10	EN 10028-7	1.4941
13E0	X2CrNiMo17-12-2	EN 10222-5	1.4404	X2CrNiMo17-12-2	EN 10028-7	1.4404
	X2CrNiMo17-12-3	EN 10222-5	1.4432	X2CrNiMo17-12-3	EN 10028-7	1.4432
	X2CrNiMo18-14-3	EN 10222-5	1.4435	X2CrNiMo18-14-3	EN 10028-7	1.4435
	X1NiCrMoCu25-20-5	EN 10222-5	Deleted text	X1NiCrMoCu25-20-5	EN 10028-7	1.4539
	—	—	—	X1NiCrMoCu31-27-4	EN 10028-7	1.4563
13E1	X2CrNiMoN17-11-2	EN 10222-5	1.4406	X2CrNiMoN17-11-2	EN 10028-7	1.4406
13E1	X2CrNiMoN17-13-3	EN 10222-5	1.4429	X2CrNiMoN17-13-3	EN 10028-7	1.4429
13E1	—	—	—	X2CrNiMoN17-13-5	EN 10028-7	1.4439
13E1	—	—	—	X1NiCrMoCuN25-20-7	EN 10028-7	1.4529
13E1	—	—	—	X1CrNiMoCuN20-18-7	EN 10028-7	1.4547
14E0	X5CrNiMo17-12-2	EN 10222-5	1.4401	X5CrNiMo17-12-2	EN 10028-7	1.4401

Group	Forgings			Flat Products		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	—	—	—	—	—	—
14E0	X3CrNiMo17-13-3	EN 10222-5	1.4436	X3CrNiMo17-13-3	EN 10028-7	1.4436
15E0	X6CrNiMoTi17-12-2	EN 10222-5	1.4571	X6CrNiMoTi17-12-2	EN 10028-7	1.4571
	—	—	—	X6CrNiMoNb17-12-2	EN 10028-7	1.4580
16E0	—	—	—	—	—	—
	—	—	—	X2CrNiN23-4	EN 10028-7	1.4362
	X2CrNiMoN22-5-3	EN 10222-5	1.4462	X2CrNiMoN22-5-3	EN 10028-7	1.4462
	X2CrNiMoN25-7-4	EN 10222-5	1.4410	X2CrNiMoN25-7-4	EN 10028-7	1.4410
	—	—	—	—	—	—

DIN Grades/Groups - Castings and Bars

Group	Castings			Bars		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
2E0	GP240GR	EN 10213-2	1.0621	—	—	—
3E0	GP240GH	EN 10213-2	1.0619	P235GH	EN 10273	1.0345
3E0	—	—	—	P250GH	EN 10273	1.0460
3E0	GP280GH	EN 10213-2	1.0625	P265GH	EN 10273	1.0425
3E1	—	—	—	P295GH	EN 10273	1.0481
4E0	G20Mo5	EN 10213-2	1.5419	16Mo3	EN 10273	1.5415

Group	Castings			Bars		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
5E0	G17CrMo5-5	EN 10213-2	1.7357	13CrMo4-5	EN 10273	1.7335
6E0	G17CrMo9-10	EN 10213-2	1.7379	11CrMo9-10	EN 10273	1.7383
	—			10CrMo9-10	EN 10273	1.7380
6E1	GX15CrMo5	EN 10213-2	1.7365	—	—	—
7E0	G17Mn5	EN 10213-3	1.1131	—	—	—
	G20Mn5	EN 10213-3	1.6220	—	—	—
7E1	—	—	—	—	—	—
	—	—	—	—	—	—
7E2	G9Ni10	EN 10213-3	1.5636	—	—	—
	—	—	—	—	—	—
	—	—	—	—	—	—
7E3	—	—	—	—	—	—
	G9Ni14	EN 10213-3	1.5638	—	—	—
	—	—	—	—	—	—
	—	—	—	—	—	—
8E0	—	—	—	—	—	—
8E2	—	—	—	P275NH	EN 10273	1.0487
				—	—	—
8E3	—	—	—	P355NH	EN 10273	1.0565
				P355QH	EN 10273	1.8867
9E0	GX23CrMoV12-1	EN 10213-2	1.4931	—	—	—
9E1	—	—	—	—	—	—

Group	Castings			Bars		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
10E0	GX2CrNi19-11	EN 10213-4	1.4309	X2CrNi18-9	EN 10272	1.4307
	—	—	—	X2CrNi19-11	EN 10272	1.4306
10E0	—	—	—	—	—	—
10E1	—	—	—	X2CrNiN18-10	EN 10272	1.4311
11E0	GX5CrNi19-10	EN 10213-4	1.4308	X5CrNi18-10	EN 10272	1.4301
	—	—	—	—	—	—
12E0	—	—	—	X6CrNiTi18-10	EN 10272	1.4541
	GX5CrNiNb19-11	EN 10213-4	1.4552	X6CrNiNb18-10	EN 10272	1.4550
	—	—	—	—	—	—
13E0	GX2CrNiMo19-11-2	EN 10213-4	1.4409	X2CrNiMo17-12-2	EN 10272	1.4404
	—	—	—	X2CrNiMo17-12-3	EN 10272	1.4432
	—	—	—	X2CrNiMo18-14-3	EN 10272	1.4435
	GX2NiCrMo28-20-2	EN 10213-4	1.4458	X1NiCrMoCu25-20-5	EN 10272	1.4539
	—	—	—	X1NiCrMoCu31-27-4	EN 10272	1.4563
13E1	—	—	—	X2CrNiMoN17-11-2	EN 10028-7	1.4406
13E1	—	—	—	X2CrNiMoN17-13-3	EN 10028-7	1.4429
13E1	—	—	—	X2CrNiMoN17-13-5	EN 10028-7	1.4439
13E1	—	—	—	X1NiCrMoCuN25-20-7	EN 10028-7	1.4529
13E1	—	—	—	X1CrNiMoCuN20-18-7	EN 10272	1.4547

Group	Castings			Bars		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
14E0	GX5CrNiMo19-11-2	EN 10213-4	1.4408	X5CrNiMo17-12-2	EN 10272	1.4547
14E0	—	—	—	X3CrNiMo17-13-3	EN 10272	1.4401
15E0	—	—	—	X6CrNiMoTi17-12-2	EN 10272	1.4436
	GX5CrNiMoNb19-11-2	EN 10213-4	1.4581	X6CrNiMoNb17-12-2	EN 10272	1.4571
16E0	GX2CrNiMoCuN25-6-3-3	EN 10213-4	1.4517	—	—	1.4580
	—	—	—	X2CrNiN23-4	EN 10272	—
	GX2CrNiMoN22-5-3	EN 10213-4	1.4470	X2CrNiMoN22-5-3	EN 10272	1.4362
	—	—	—	X2CrNiMoN25-7-4	EN 10272	1.4462
	GX2CrNiMoN26-7-4	EN 10213-4	1.4469	—	—	1.4410

DIN Grades/Groups - Seamless and Welded Tubes

Group	Seamless Tubes			Welded Tubes		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
2E0	—	—	—	—	—	—
3E0	P195GH	EN 10216-2	1.0348	P195GH	EN 10217-2	1.0348
3E0	P235GH	EN 10216-2	1.0345	P235GH	EN 10217-2	1.0345
3E1	P265GH	EN 10216-2	1.0425	P265GH	EN 10217-2	1.0425
4E1	16Mo3	EN 10216-2	1.5415	16Mo3	EN 10217-2	1.5415
5E0	13CrMo4-5	EN 10216-2	1.7335	—	—	—

Group	Seamless Tubes			Welded Tubes		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
6E0	10CrMo9-10	EN 10216-2	1.7380	—	—	—
6E0	11CrMo9-10	EN 10216-2	1.7383	—	—	—
6E1	11CrMo5+NT1	EN 10216-2	1.7362+1	—	—	—
7E0	P275NL1	EN 10216-3	1.0488	P275NL1	EN 10217-3	1.0488
7E0	P275NL2	EN 10216-3	1.1104	P275NL2	EN 10217-3	1.1104
7E1	P355NL1	EN 10216-3	1.0566	P355NL1	EN 10217-3	1.0566
7E1	P355NL2	EN 10216-3	1.1106	P355NL2	EN 10217-3	1.1106
7E2	12Ni14	EN 10216-4	1.5637	—	—	—
7E2	X10Ni9	EN 10216-4	1.5682	—	—	—
7E3	13MnNi6-3	EN 10216-4	1.6217	—	—	—
8E0	P275NL1	EN 10216-3	1.0488	P275NL1	EN 10217-3	1.0488
8E0	P275NL2	EN 10216-3	1.1104	P275NL2	EN 10217-3	1.1104
8E2	—	—	—	—	—	—
8E3	P355NH	EN 10216-3	1.0565	P355NH	EN 10217-3	1.0565
9E0	X20CrMoV11-1	EN 10216-2	1.4922	—	—	—
9E1	X10CrMoVNb9-1	EN 10216-2	1.4903	—	—	—
10E0	X2CrNi18-9	EN 10216-5	1.4307	X2CrNi18-9	EN 10217-7	1.4307
	X2CrNi19-11	EN 10216-5	1.4306	X2CrNi19-11	EN 10217-7	1.4306
10E0	X1CrNi25-21	EN 10216-5	1.4335	—	—	—
	X2CrNiN18-10	EN 10216-5	1.4311	X2CrNiN18-10	EN 10217-7	1.4311
11E0	X5CrNi18-10	EN 10216-5	1.4301	X5CrNi18-10	EN 10217-7	1.4301

Group	Seamless Tubes			Welded Tubes		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
11E0	X6CrNi18-10	EN 10216-5	1.4948	—	—	—
12E0	X6CrNiTi18-10	EN 10216-5	1.4541	X6CrNiTi18-10	EN 10217-7	1.4541
	X6CrNiNb18-10	EN 10216-5	1.4550	X6CrNiNb18-10	EN 10217-7	1.4550
12E0	X7CrNiTi18-10	EN 10216-5	1.4940	—	—	—
12E0	X7CrNiTiB18-10	EN 10216-5	1.4941	—	—	—
12E0	X7CrNiNb18-10	EN 10216-5	1.4912	—	—	—
	X8CrNiNb16-13	EN 10216-5	1.4961	—	—	—
13E0	X2CrNiMo17-12-2	EN 10216-5	1.4404	X2CrNiMo17-12-2	EN 10217-7	1.4404
	—	—	—	X2CrNiMo17-12-3	EN 10217-7	1.4432
	X2CrNiMo18-14-3	EN 10216-5	1.4435	X2CrNiMo18-14-3	EN 10217-7	1.4435
13E0	X1NiCrMoCu25-20-5	EN 10216-5	1.4539	X1NiCrMoCu25-20-5	EN 10217-7	1.4539
	X1NiCrMoCu31-27-4	EN 10216-5	1.4563	X1NiCrMoCu31-27-4	EN 10217-7	1.4563
	—	—	—	X2CrNiMoN18-15-4	EN 10217-7	1.4438
	X6CrNiMo17-13-2	EN 10216-5	1.4918	—	—	—
13E1	X2CrNiMoN17-13-3	EN 10216-5	1.4429	X2CrNiMoN17-13-3	EN 10217-7	1.4429

Group	Seamless Tubes			Welded Tubes		
	Material Name	Standard	Material Number	Material Name	Standard	Material Number
	X2CrNiMoN17-13-5	EN 10216-5	1.4439	X2CrNiMoN17-13-5	EN 10217-7	1.4439
	X1CrNiMoN25-22-2	EN 10216-5	1.4466	—	—	—
	X1CrNiMoCuN20-18-7	EN 10216-5	1.4547	X1CrNiMoCuN20-18-7	EN 10217-7	1.4547
	X1NiCrMoCuN25-20-7	EN 10216-5	1.4529	X1NiCrMoCuN25-20-7	EN 10217-7	1.4529
14E0	X5CrNiMo17-12-2	EN 10216-5	1.4401	X5CrNiMo17-12-2	EN 10217-7	1.4401
	X3CrNiMo17-13-3	EN 10216-5	1.4436	X3CrNiMo17-13-3	EN 10217-7	1.4436
15E0	X6CrNiMoTi17-12-2	EN 10216-5	1.4571	X6CrNiMoTi17-12-2	EN 10217-7	1.4571
	X6CrNiMoNb17-12-2	EN 10216-5	1.4580	—	—	—
16E0	X2CrNiMoS18-5-3	EN 10216-5	1.4424	—	—	—
	X2CrNiMoN22-5-3	EN 10216-5	1.4462	X2CrNiMoN22-5-3	EN 10217-7	1.4462
	X2CrNiN23-4	EN 10216-5	1.4362	X2CrNiN23-4	EN 10217-7	1.4362
	X2CrNiMoN25-7-4	EN 10216-5	1.4410	X2CrNiMoN25-7-4	EN 10217-7	1.4410
	X2CrNiMoCuN25-6-3	EN 10216-5	1.4507	—	—	—
	X2CrNiMoCuWN25-7-4	EN 10216-5	1.4501	X2CrNiMoCuWN25-7-4	EN 10217-7	1.4501
13E1	X2CrNiMoN17-13-3	EN 10216-5	1.4429	X2CrNiMoN17-13-3	EN 10217-7	1.4429

Group	Seamless Tubes			Welded Tubes		
	Material Name	**Standard**	**Material Number**	**Material Name**	**Standard**	**Material Number**
	X2CrNiMoN17-13-5	EN 10216-5	1.4439	X2CrNiMoN17-13-5	EN 10217-7	1.4439
	X1CrNiMoN25-22-2	EN 10216-5	1.4466	—	—	—
	X1CrNiMoCuN20-18-7	EN 10216-5	1.4547	X1CrNiMoCuN20-18-7	EN 10217-7	1.4547
	X1NiCrMoCuN25-20-7	EN 10216-5	1.4529	X1NiCrMoCuN25-20-7	EN 10217-7	1.4529
14E0	X5CrNiMo17-12-2	EN 10216-5	1.4401	X5CrNiMo17-12-2	EN 10217-7	1.4401
	X3CrNiMo17-13-3	EN 10216-5	1.4436	X3CrNiMo17-13-3	EN 10217-7	1.4436
15E0	X6CrNiMoTi17-12-2	EN 10216-5	1.4571	X6CrNiMoTi17-12-2	EN 10217-7	1.4571
	X6CrNiMoNb17-12-2	EN 10216-5	1.4580	—	—	—
16E0	X2CrNiMoS18-5-3	EN 10216-5	1.4424	—	—	—
	X2CrNiMoN22-5-3	EN 10216-5	1.4462	X2CrNiMoN22-5-3	EN 10217-7	1.4462
	X2CrNiN23-4	EN 10216-5	1.4362	X2CrNiN23-4	EN 10217-7	1.4362
	X2CrNiMoN25-7-4	EN 10216-5	1.4410	X2CrNiMoN25-7-4	EN 10217-7	1.4410
	X2CrNiMoCuN25-6-3	EN 10216-5	1.4507	—	—	—
	X2CrNiMoCuWN25-7-4	EN 10216-5	1.4501	X2CrNiMoCuWN25-7-4	EN 10217-7	1.4501

Flange Type

Select the ANSI flange type. Select **Weld Neck**, **Slip On**, or **Blind**. This option is only available when **Body Flange** is selected for **Element Type**.

Nominal Size Lookup

Select a nominal flange diameter. The flange diameter, length, and thickness are automatically updated. This value is not saved. This option is only available when **Body Flange** is selected for **Element Type**.

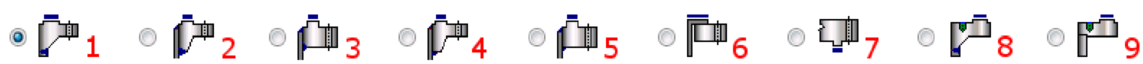
Flange Dialog Box

Defines values for calculating actual and allowable stresses for all types of flanges according to the ASME Code, Section VIII, Division 1, VIII-2; PD 5500; and EN-13445.

Select a Flange Type

Select the type of flange:

- 1 - Integral Weld Neck Flange
- 2 - Integral Slip on Flange
- 3 - Integral Ring Flange
- 4 - Loose Slip on Flange
- 5 - Loose Ring Flange
- 6 - Lap Joint Flange
- 7 - Bolted Blind Flange or TEMA channel Cover
- 8 - Reverse Geometry Weld Neck Flange
- 9 - Loose Reverse Flange



NOTE If the type 7 blind flange is selected, the modulus of elasticity at design temperature is needed. By default the software uses the external pressure charts to retrieve this information. Alternately, you can use the elasticity data supplied in the TEMA standard by typing in the TEMA identifier in **Flange ID**. The TEMA identifier is a number that ranges between 16 and 50, and depends on the composition of the flange material. Listings of the TEMA numbers can be found in *Tubesheet Type and Design Code Tab (Heat Exchanger Tubesheet Input Dialog Box)* (page 134).

Description

Enter an alphanumeric description for the item.

Flange Thickness

This option is used in the following ways:

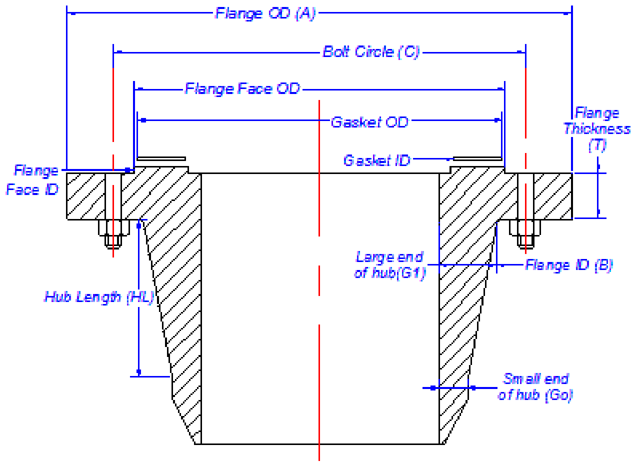
- Displays standard flange thickness after **Get Flange Dimensions Now** is clicked.
- Displays designed flange thickness after **Design** is clicked.
- Enter a manual flange thickness.

Flange ID

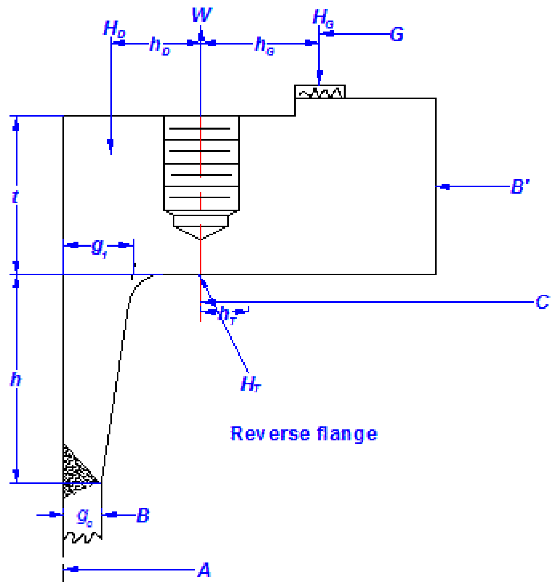
Enter the inside diameter of the flange. For integral-type flanges, this value is also the inner pipe diameter. This value is referred to as *B* in the ASME code. **Internal Corrosion**

Allowance is used to adjust this value (two times the corrosion allowance is added to the uncorroded value of **Flange ID**).

For blind flanges the value is 0.0.



For reverse flanges this is the *B*` dimension as shown in appendix 2 of the ASME Code.



Flange OD

Enter the outer diameter of the flange. This value is referred to as *A* in the ASME code. If the flange is being corroded from the outside, be sure this is a corroded dimension.

Face ID

Enter the inner diameter of the flange face. The software uses the maximum of the **Face ID** and **Gasket ID** to calculate the inner contact point of the gasket.

Face OD

Enter the outer diameter of the flange face. The software uses the minimum of **Face OD** and the **Gasket OD** to calculate the outside flange contact point, but uses the maximum in design when selecting the bolt circle. This is done so that the bolts do not interfere with the gasket.

Gasket ID

Enter the inner diameter of the gasket. The software uses the maximum of the **Flange ID** and the **Gasket ID** to calculate the inner contact point of the gasket.

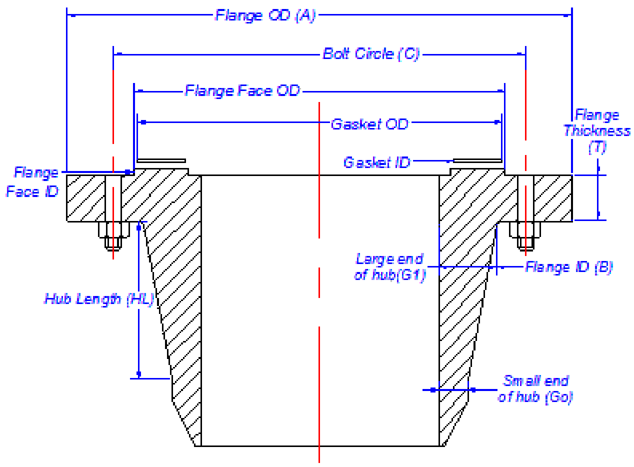
Gasket OD

Enter the outer diameter of the gasket. The software uses the minimum of **Flange OD** and **Gasket OD** to calculate the outside flange contact point, but uses the maximum in design when selecting the bolt circle. This is done so that the bolts do not interfere with the gasket.

Thickness Large

Enter the thickness of the large end of the hub. This value is referred to as *G1* in the ASME code. The corrosion allowance is subtracted from this value. It is permissible for the hub thickness at the large end to equal the hub thickness at the small end.

For flange geometry without hubs, this thickness may be entered as zero.



Thickness Small

Enter the thickness of the small end of the hub. This value is referred to as G_o in the ASME code. The corrosion allowance is subtracted from this value.

For weld neck flange types, this is the thickness of the shell at the end of the flange. For loose slip on flanges without hubs, this is typically the thickness of the attached shell or the hub dimensions can all be zero. For loose ring flanges with a hub, this is the thickness of the hub. For optional loose ring flanges with hubs analyzed as integral, this is the thickness of the attached shell. See ASME general note 2 of Figure 2-4 for more information.

If you are using PD 5500 or EN-13445, please check the code for correct input values.

Hub Length

Enter the hub length. This value is referred to as *H* in the ASME code.

For flange geometry without hubs, this length may be entered as zero. When analyzing an optional type flange that is welded at the hub end, the hub length should be the leg of the weld, and the thickness at the large end should include the thickness of the weld.

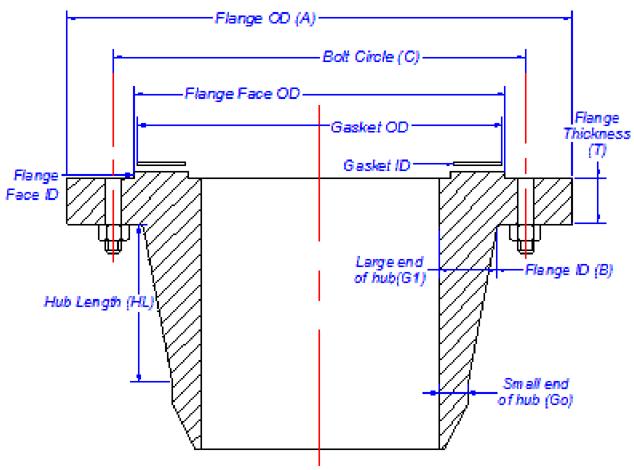
When you analyze a flange with no hub, such as a ring flange or a lap joint flange, you should enter zero for the hub length, the small end of the hub, and the large end of the hub. However, when you design a loose flange as a ring flange which has a fillet weld at the back, enter the size of a leg of the fillet weld as the large end of the hub. This insures that the software designs the bolt circle far enough away from the back of the flange to get a wrench around the nuts.

Bolt Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Bolt Circle Diameter

Enter the diameter of the bolt circle of the flange.



Thread Series

Select the thread series bolt table. Select **TEMA**, **TEMA Metric**, **UNC**, **BS 3643**, or **SABS 1700 1996.** If you have a bolt that is outside of the bolt table ranges, select **User Root Area** and enter the nominal size in **Nominal Bolt Diameter**.

Nominal Bolt Diameter

Click ... and select the nominal bolt diameter from the bolt table selected for **Type of Threads**. If you have a bolt that is outside of the bolt table ranges, enter the nominal size and select **User Root Area** in **Type of Threads**.

For **TEMA** and **UNC** bolts, the available bolt diameters are:

Bolt Size (inches)	Bolt Root Area	
	(cm.^2)	(in.^2)
0.500	0.8129	0.126

Bolt Size (inches)	Bolt Root Area	
0.625	1.3032	0.202
0.750	1.9484	0.302
0.875	2.7032	0.419
1.000	3.5548	0.551
1.125	4.6968	0.728
1.250	5.9935	0.929
1.375	7.4516	1.155
1.500	9.0645	1.405
1.625	10.8387	1.680
1.750	12.7741	1.980
1.875	14.8645	2.304
2.000	17.1096	2.652
2.250	22.0838	3.423
2.500	27.6903	4.292
2.750	33.9290	5.259
3.000	40.7999	6.324
3.250	48.3031	7.487
3.500	56.4450	8.749
3.750	65.2130	10.108
4.000	74.6192	11.566

NOTE This information is adapted from Jawad and Farr, *Structural Analysis and Design of Process Equipment*, pg 425.

Number of Bolts

Enter the number of bolts to be used in the flange analysis.

Root Area

For nonstandard bolts, enter the root cross-sectional area of the bolt.

Gasket Factor m | y

Enter values for the gasket material and contact facing. Enter the factor ratio (**m**) and minimum stress (**y**). Alternatively, click **...** to open the **Select a Gasket** dialog box, and select a standard gasket material.

ASME Table 2-5.1 - Gasket List

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Self-energizing types (O rings, metallic elastomer, and other types considered as self-sealing)	0.00	0.00	II
Elastomers WO/ fabric or high % of asbestos fiber:			
Below 75A Shore Durometer	0.50	0.0	II
75A or higher Shore Durometer	1.00	200.0	II
Asbestos with Suitable Binder			
1/8" thick	2.00	1600.0	II
1/16" thick	2.75	3700.0	II
1/32" thick	3.50	6500.0	II
Elastomers W/cotton fabric insert	1.25	400.0	II
Elastomers W/ asbestos fabric insert:			
3 ply	2.25	2200.0	II
2 ply	2.50	2900.0	II
1 ply	2.75	3700.0	II
Vegetable fiber	1.75	1100.0	II
Spiral-wound, asbestos filled:			
Carbon	2.50	10000.0	II
Stainless, Monel, Nickel alloys	3.00	10000.0	II
Corrugated metal, asbestos ins. or corrugated metal, jacketed asbestos:			

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
soft aluminum	2.50	2900.0	II
soft copper or brass	2.75	3700.0	II
iron or soft steel	3.00	4500.0	II
Monel or 4%-6% chrome	3.25	5500.0	II
SS/nickel base alloys	3.50	6500.0	II
Corrugated metal:			
soft aluminum	2.75	3700.0	II
soft copper or brass	3.00	4500.0	II
iron or soft steel	3.25	5500.0	II
Monel or 4%-6% chrome	3.50	6500.0	II
SS/nickel base alloys	3.75	7600.0	II
Flat metal, jacketed asbestos filled:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel	3.50	8000.0	II
4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	3.75	9000.0	II
Grooved metal:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel or 4%-6% chrome	3.75	9000.0	II

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
SS/nickel base alloys	4.25	10100.0	II
Solid flat metal:			
soft aluminum	4.00	8800.0	I
soft copper or brass	4.75	13000.0	I
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I
Ring joint:			
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I

NOTE *SS* refers to stainless steel

Sketch

Select the facing sketch number according to the following and using Table 2-5.2 of the ASME code (see **Gasket Factor m | y**):

Facing Sketch	Description
1a	Flat finish faces
1b	Serrated finish faces
1c	Raised nubbin-flat finish
1d	Raised nubbin-serrated finish
2	1/64 inch nubbin
3	1/64 inch nubbin both sides
4	Large serration, one side
5	Large serration, both sides

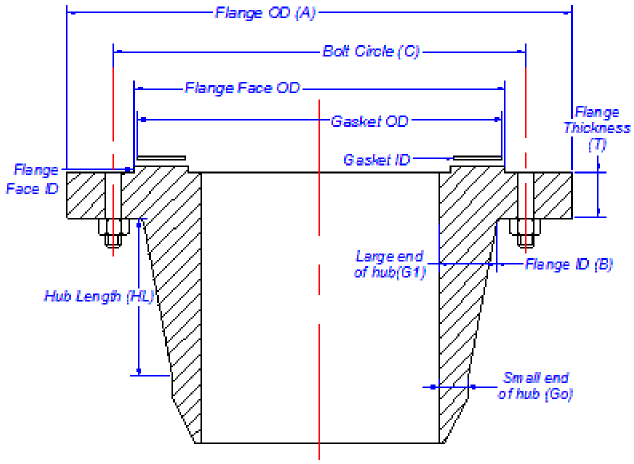
6	Metallic O-ring type gasket

Column

Select the facing column according to **ASME Table 2-5.1 - Gasket List**.

Gasket Thickness

Enter the gasket thickness. This value is only required for facing sketches 1c and 1d (PV Elite equivalents 3 and 4). For more information, see **Sketch**.



Nubbin or RTJ Width

Enter the nubbin width. This value is only required for facing sketches 1c, 1d, 2 and 6 (PV Elite equivalents 3, 4, 5, and 9). For sketch 9, this is not a nubbin width, this is the width of the ring that is in contact with the groove in which the ring gasket sits. For more information, see **Sketch**.

Base Required Thickness on Rigidity (ASME VIII-1)?

Select to calculate the flange rigidity index for both the mating flange **Operating, Wm1 and Seating, Wm2** (as it would normally). If either rigidity index is greater than 1.0, the software iteratively adjusts the flange required thickness until neither index is greater than 1.0. Using this method may prove useful for designing flanges with the new higher allowable stresses (ASME 99 addenda and later), because, as of November 1999, the code has not explicitly addressed the situation of using higher allowables for flange design.

NOTE In some cases, stress considerations govern over rigidity, and the thickness is not influenced by rigidity.

Include Corrosion in Flange Thickness Calculations?

Select to use the corrosion allowance (as is usual) when calculating the final stresses on the flange.

Clear to calculate stresses without subtracting corrosion from the flange thickness, typically producing a thinner flange that is not as highly stressed. The MAWP of the flange is also usually slightly higher. Because this is not directly addressed by any code, agreement is usually between the customer and the manufacturer.

Rigidity Calc. for Seating Case Uses Corroded Thickness?

Select to incorporate the corroded thickness into the rigidity calculation, especially for cases of old pipe already corroded.

Are the Hub and Attached Shell Materials the Same?

Select to indicate that the flange material is same as the attached shell material. The larger of the shell allowable and the flange allowable stress are used to calculate the required small-end hub thickness for the attached integral flange. Clear to consider only the flange allowable.

For some materials with relatively low yield strength (such as stainless steels), the ASME code has established higher stress values. These higher stress values (indicated by the presence of the note G5) can lead to higher deformation. As a result, these material allowables are not used for applications where flange deformation can cause failure or leakage.

When the flange allowables are lower as compared to the attached shell (for the same material), the small-end hub thickness is checked as a cylinder. The allowable stresses of the attached shell should then be used. Otherwise, the required hub thickness may be greater than that of the attached cylinder. If you want the higher shell allowables to be used, select **Are the Hub and Attached Shell Materials the Same?**.

Is this a Standard Flange (No calculation performed)?

Select to define the flange by using ANSI/DIN standards.

Weld Leg Back of Ring

Enter the length of the weld leg at the back of the ring. This value is added to the inside diameter during the design of ring type flanges to determine the minimum bolt circle.

Attached Shell Thickness

Enter the thickness of the shell attached to the flange.

Lap Joint Contact Point ID

Enter the inner diameter of the flange/joint contact surface.

Lap Joint Contact Point OD

Enter the outer diameter of the flange/joint contact surface.

Number of Splits in Ring

Enter the number of splits in the ring, if any, for loose-type flanges. This value must be **0**, **1,** or **2**. Typical split flanges are ring-type flanges. A split is used when the flange must be completely removable from the vessel. If the flange is split into two pieces by a single split, the design moment for the flange is multiplied by 2.0. If the flange consists of two separate split rings, each ring is designed as if it were a solid flange (without splits) using 0.75 times the design moment. The pair of rings is assembled so that the splits in one ring are 90º from the splits in the other.

Calculate Cover Deflection (multi-pass unit)?

Select if the flange is a TEMA channel cover (bolted blind flange). A separate thickness, deflection, and MAWP are calculated for channel covers.

Allowable Cover Deflection

For TEMA channel covers, enter the maximum allowed deflection at the center of the cover, as defined in TEMA paragraph RCB-9.21. For TEMA covers with nominal diameters less than or equal to 24 inches, the allowed deflection should be set to **.03** inches. For larger sizes, the deflection should be limited to .125% of the nominal diameter (nominal diameter / 800). If this field is left blank the appropriate allowable deflection will be used. Alternatively, click ... to open the **Compute Allowable Channel Cover Deflection** dialog box, and enter a value for **Nominal Diameter of Attached Shell**. The software then calculates the allowable cover deflection.

Diameter (Long Span)

Enter the head characteristic diameter, used to calculate the non-circular flange correction factor *Z* for ASME blind flanges. This factor is defined in paragraph UG-34 of the ASME code. This dimension is perpendicular to the short span dimension.

NOTE For circular blind flanges, the short span and long span dimensions are the same.

Diameter (Short Span)

Enter the head characteristic short diameter, used to calculate the non-circular flange correction factor *Z* for ASME blind flanges. This factor is defined in paragraph UG-34 of the ASME code. This dimension is perpendicular to the long span dimension. It is very important to enter this dimension correctly because it is used in the tangential flange stress computation.

NOTE For circular blind flanges, the short span and long span dimensions are the same.

Perimeter Along Bolt Circle Centerline

Enter the perimeter *L* of the bolted head measured along the centerline of the bolts. This value is needed for both noncircular and circular geometry. For a circular head, enter the value of (π * bolt circle diameter). For non-circular heads this value will have to be manually calculated and entered.

Class

Select the ANSI/DIN flange class, which is based on the pressure rating.

Grade

Select the flange material grade (group). Please note that there are certain advisories on the use of certain material grades. Please review those cautionary notes in the ANSI B16.5 code. ASME B16.5-2003 and ASME B16.5-1996 flange grades are available:

Table 1A List of Material Specifications (ASME B16.5-2003)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V 3½ Ni	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1 A 350 Gr. LF3	A 216 Gr. WCB	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 2½Ni 3½Ni	 A 350 Gr. LF 6 Cl.2	A 316 Gr. WCC A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	 A 203 Gr. B A 203 Gr. E

Material Group	Nominal Designation	Forgings	Castings	Plates
1.3	C-Si C-Mn-Si 2 ½Ni 3 ½Ni C-½Mo		A 352 Gr. LCB A 217 Gr. WC1 A 352 Gr. LC1	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1		A 204 Gr. A A 204 Gr. B
1.7	½C-½Mo Ni-½Cr-½Mo ¾Ni-¾Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1¼Cr-½Mo 1¼Cr-½Mo-Si	A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	2¼Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.11	Cr-½Mo			A 204 Gr. C
1.13	5Cr-½Mo	A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2
1.17	1Cr-½Mo 5Cr-½Mo	A 182 Gr. F12 Cl.2 A 182 Gr. F5		
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H A 182 Gr. F317	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H		A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	23Cr-12Ni			A 240 Gr. 309H

Material Group	Nominal Designation	Forgings	Castings	Plates
2.7	25Cr-20Ni	A 182 Gr. F310		A 240 Gr. 310H
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
2.9	23Cr-12Ni 25Cr-20Ni			A 240 Gr. 309S A 240 Gr. 310S
2.10	25Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	
2.11	18Cr-10Ni-Cb		A 351 Gr. CF8C	
2.12	25Cr-20Ni		A 351 Gr. CK20	
3.1	35Ni-35Fe-10Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe 64Ni-29.5Mo-2Cr-2Fe-Mn-W	B 462 Gr. N10665 B 462 Gr. N10675		B 333 Gr. N10665 B 333 Gr. N10675
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu 55Ni-21Cr-13.5Mo 55Ni-23Cr-16Mo-1.6Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825 B 462 Gr. N06022 B 462 Gr. N06200		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825 B 575 Gr. N06022 B 575 Gr. N06200
3.9	47Ni-22Cr-9Mo-I8Fe	B 572 Gr. N06002		B 435 Gr. N06002
3.10	25Ni-46Fe-21Cr-5Mo	B 672 Gr. N08700		B 599 Gr. N08700
3.11	44Fe-25Ni-21Cr-Mo	B 649 Gr. N08904		B 625 Gr. N08904
3.12	26Ni-43Fe-22Cr-5Mo 47Ni-22Cr-20Fe-7Mo 46Fe-24Ni-21Cr-6Mo-Cu-N	B 621 Gr. N08320 B 581 Gr. N06985 B 462 Gr. N08367	 A 351 Gr. CN3MN	B 620 Gr. N08320 B 582 Gr. N06985 B 688 Gr. N08367

Material Group	Nominal Designation	Forgings	Castings	Plates
3.13	49Ni-25Cr-18Fe-6Mo Ni-Fe-Cr-Mo-Cu-Low C	B 581 Gr. N06975 B 462 Gr. N08031		B 582 Gr. N06975 B 625 Gr. N08031
3.14	47Ni-22Cr-19Fe-6Mo 40Ni-29Cr-15Fe-5Mo	B 581 Gr. N06007 B 462 Gr. N06030		B 582 Gr. N06007 B 582 Gr. N06030
3.15	33Ni-42Fe-21Cr	B 564 Gr. N08810		B 409 Gr. N08810
3.16	35Ni-19Cr-1¼Si	B 511 Gr. N08330		B 536 Gr. N08330
3.17	29Ni-20.5Cr-3.5Cu-2.5Mo		A 351 Gr. CN7M	

Table 1A List of Material Specifications (ASME B16.5-1996)

Material Group	Nominal Designation	Forgings	Castings	Plates
1.1	C-Si C-Mn-Si C-Mn-Si-V	A 105 A 350 Gr. LF2 A 350 Gr. LF 6 Cl.1	A 216 Gr. WCB A 216 Gr. WCC	A 515 Gr. 70 A 516 Gr. 70 A 537 Cl. 1
1.2	C-Mn-Si C-Mn-Si-V 21/2Ni 31/2Ni	A 350 Gr. LF 6 Cl.2 A 350 Gr. LF3	A 352 Gr. LCC A 352 Gr. LC2 A 352 Gr. LC3	A 203 Gr. B A 203 Gr. E
1.3	C-Si C-Mn-Si 21/2Ni 31/2Ni		A 352 Gr. LCB	A 515 Gr. 65 A 516 Gr. 65 A 203 Gr. A A 203 Gr. D
1.4	C-Si C-Mn-Si	A 350 Gr. LF1 Cl. 1		A 515 Gr. 60 A 516 Gr. 60
1.5	C-1/2Mo	A 182 Gr. F1	A 217 Gr. WC1 A 352 Gr. LC1	A 204 Gr. A A 204 Gr. B A 204 Gr. C
1.7	C-1/2Mo 1/2Cr-1/2Mo Ni-1/2Cr-1/2Mo 3/4Ni-3/4Cr-1Mo	A 182 Gr. F2	A 217 Gr. WC4 A 217 Gr. WC5	
1.9	1Cr-1/2Mo 11/4Cr-1/2Mo 11/4Cr-1/2Mo-Si	A 182 Gr. F12 Cl.2 A 182 Gr. F11 Cl.2	A 217 Gr. WC6	A 387 Gr. 11 Cl.2
1.10	21/4Cr-1Mo	A 182 Gr. F22 Cl.3	A 217 Gr. WC9	A 387 Gr. 22 Cl.2
1.13	5Cr-1/2Mo	A 182 Gr. F5 A 182 Gr. F5a	A 217 Gr. C5	
1.14	9Cr-1Mo	A 182 Gr. F9	A 217 Gr. C12	
1.15	9Cr-1Mo-V	A 182 Gr. F91	A 217 Gr. C12A	A 387 Gr. 91 Cl.2

Material Group	Nominal Designation	Forgings	Castings	Plates
2.1	18Cr-8Ni	A 182 Gr. F304 A 182 Gr. F304H	A 351 Gr. CF3 A 351 Gr. CF8	A 240 Gr. 304 A 240 Gr. 304H
2.2	16Cr-12Ni-2Mo 18Cr-13Ni-3Mo 19Cr-10Ni-3Mo	A 182 Gr. F316 A 182 Gr. F316H	A 351 Gr. CF3M A 351 Gr. CF8M A 351 Gr. CG8M	A 240 Gr. 316 A 240 Gr. 316H A 240 Gr. 317
2.3	18Cr-8Ni 16Cr-12Ni-2Mo	A 182 Gr. F304L A 182 Gr. F316L		A 240 Gr. 304L A 240 Gr. 316L
2.4	18Cr-10Ni-Ti	A 182 Gr. F321 A 182 Gr. F321H		A 240 Gr. 321 A 240 Gr. 321H
2.5	18Cr-10Ni-Cb	A 182 Gr. F347 A 182 Gr. F347H A 182 Gr. F348 A 182 Gr. F348H	A 351 Gr. CF8C	A 240 Gr. 347 A 240 Gr. 347H A 240 Gr. 348 A 240 Gr. 348H
2.6	25Cr-12Ni 23Cr-12Ni		A 351 Gr. CH8 A 351 Gr. CH20	A 240 Gr. 309S A 240 Gr. 309H
2.7	25Cr-20Ni	A 182 Gr. F310	A 351 Gr. CK20	A 240 Gr. 310S A 240 Gr. 310H
2.8	20Cr-18Ni-6Mo 22Cr-5Ni-3Mo-N 25Cr-7Ni-4Mo-N 24Cr-10Ni-4Mo-V 25Cr-5Ni-2Mo-3Cu 25Cr-7Ni-3.5Mo-W-Cb 25Cr-7Ni-3.5Mo-N-Cu-W	A 182 Gr. F44 A 182 Gr. F51 A 182 Gr. F53 A 182 Gr. F55	A 351 Gr. CK3McuN A 351 Gr. CE8MN A 351 Gr. CD4Mcu A 351 Gr. CD3MWCuN	A 240 Gr. S31254 A 240 Gr. S31803 A 240 Gr. S32750 A 240 Gr. S32760
3.1	35Ni-35Fe-20Cr-Cb	B 462 Gr. N08020		B 463 Gr. N08020
3.2	99.0Ni	B 160 Gr. N02200		B 162 Gr. N02200
3.3	99.0Ni-Low C	B 160 Gr. N02201		B 162 Gr. N02201
3.4	67Ni-30Cu 67Ni-30Cu-S	B 564 Gr. N04400 B 164 Gr. N04405		B 127 Gr. N04400
3.5	72Ni-15Cr-8Fe	B 564 Gr. N06600		B 168 Gr. N06600
3.6	33Ni-42Fe-21Cr	B 564 Gr. N08800		B 409 Gr. N08800
3.7	65Ni-28Mo-2Fe	B 335 Gr. N10665		B 333 Gr. N10665
3.8	54Ni-16Mo-15Cr 60Ni-22Cr-9Mo-3.5Cb 62Ni-28Mo-5Fe 70Ni-16Mo-7Cr-5Fe 61Ni-16Mo-16Cr 42Ni-21.5Cr-3Mo-2.3Cu	B 564 Gr. N10276 B 564 Gr. N06625 B 335 Gr. N10001 B 573 Gr. N10003 B 574 Gr. N06455 B 564 Gr. N08825		B 575 Gr. N10276 B 443 Gr. N06625 B 333 Gr. N10001 B 434 Gr. N10003 B 575 Gr. N06455 B 424 Gr. N08825

Nom

Select the nominal value for the flange inside diameter.

ANSI Series

Select the ANSI flange series. Select **ANSI Series A** for general-use flanges. Select **ANSI Series B** for compact flanges.

Obtain Dimensions

Click to look up flange dimensions from ANSI/DIN tables based on the selections for **Is this a Standard Flange (No calculation performed)?**, **Class**, **Grade**, **Nom**, and **ANSI Series**. The flange thickness table value is shown in **Flange Thickness**.

NOTE Click **Quick Results** to see a report of the flange dimensions.

Use Full Bolt Load in Calc (Sa*Ab)?

Select to allow the full bolt load to be used on just the area of the bolt itself, instead of Am+ Ab, which is the area of the bolt (Ab) plus the required bolt area (Am). You use this option for allowable stress calculations.

Just Like

Select the node of an adjacent element to use the properties of that element. After selecting, click **Copy Now**.

Copy Now

Click to copy the properties of an adjacent element selected in **Just Like**. The **Select Items to Copy** dialog box opens. You then select the properties to copy.

Design

Click to design the flange based on all properties selected in the **Flange** dialog box.

Loads\Partition Gasket Information

Opens the **Additional Flange Data Dialog Box**, where you can enter more details about the selected flange.

Quick Results

Click to see a report of flange results.

Delete

Click to delete all data entered in the **Flange** dialog box, restore default flange values, and close the dialog box.

OK

Click to save all data entered in the **Flange** dialog box and close the dialog box.

Cancel

Click to close the **Flange** dialog box without saving the entered data.

Plot

Click to open the **Flange Graphics** dialog box. A cross-section view of the flange design is shown.

Additional Flange Data Dialog Box

Specifies additional flange data information for the selected flange. This dialog box appears when you select **Loads\Partition Gasket Information** on the **Flange Dialog Box** (page 303).

Operating, Wm1

Enter the bolt load from the mating flange for the operating case.

Seating, Wm2

Enter the bolt load from the mating flange for the seating condition.

Design, W

Enter the design bolt load for the mating flange.

Axial Force

Enter the magnitude of the external axial force acting on the flange. Because the axial load rotates the flange and causes stress in the hub, this value should always be positive.

Bending Moment

Enter the magnitude of the external bending moment acting on the flange. This is typically the square root of the sum of the squares of moments that contribute to bending (thus. the torsional moment would not be considered). The axial force and resolved moment increase the design pressure on the flange. This is because only pressure and bolt loads cause loads on flange. The ASME code does not consider the effect of bending and external forces on flanges.

Length

Enter the length of the partition gasket. This is the cumulative length of all heat exchanger pass partition gaskets associated with this flange.

NOTE If the flange does have a partition gasket, enter **0** for **Length** and **Width**.

Width

Enter the width of the pass partition gasket. Using **Width**, **Sketch**, **Column, and Gasket Factor m | y,** the software calculates the effective seating width and the gasket loads contributed by the partition gasket.

NOTE If the flange does have a partition gasket, enter **0** for **Length** and **Width**.

Sketch

Select the facing sketch number according to the following and using Table 2-5.2 of the ASME code (see **Gasket Factor m | y**):

Facing Sketch	**Description**
1a	Flat finish faces
1b	Serrated finish faces
1c	Raised nubbin-flat finish

1d	Raised nubbin-serrated finish
2	1/64 inch nubbin
3	1/64 inch nubbin both sides
4	Large serration, one side
5	Large serration, both sides
6	Metallic O-ring type gasket

Column

Select the facing column according the table below:

ASME Table 2-5.1 - Gasket List

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Self-energizing types (O rings, metallic elastomer, and other types considered as self-sealing)	0.00	0.00	II
Elastomers WO/ fabric or high % of asbestos fiber:			
Below 75A Shore Durometer	0.50	0.0	II
75A or higher Shore Durometer	1.00	200.0	II
Asbestos with Suitable Binder			
1/8" thick	2.00	1600.0	II
1/16" thick	2.75	3700.0	II
1/32" thick	3.50	6500.0	II
Elastomers W/cotton fabric insert	1.25	400.0	II
Elastomers W/ asbestos fabric insert:			
3 ply	2.25	2200.0	II
2 ply	2.50	2900.0	II
1 ply	2.75	3700.0	II
Vegetable fiber	1.75	1100.0	II

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Spiral-wound, asbestos filled:			
Carbon	2.50	10000.0	II
Stainless, Monel, Nickel alloys	3.00	10000.0	II
Corrugated metal, asbestos ins. or corrugated metal, jacketed asbestos:			
soft aluminum	2.50	2900.0	II
soft copper or brass	2.75	3700.0	II
iron or soft steel	3.00	4500.0	II
Monel or 4%-6% chrome	3.25	5500.0	II
SS/nickel base alloys	3.50	6500.0	II
Corrugated metal:			
soft aluminum	2.75	3700.0	II
soft copper or brass	3.00	4500.0	II
iron or soft steel	3.25	5500.0	II
Monel or 4%-6% chrome	3.50	6500.0	II
SS/nickel base alloys	3.75	7600.0	II
Flat metal, jacketed asbestos filled:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel	3.50	8000.0	II
4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	3.75	9000.0	II
Grooved metal:			

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel or 4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	4.25	10100.0	II
Solid flat metal:			
soft aluminum	4.00	8800.0	I
soft copper or brass	4.75	13000.0	I
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I
Ring joint:			
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I

NOTE *SS* refers to stainless steel

Gasket Factor m | y

Enter values for the gasket material and contact facing. Enter the factor ratio (**m**) and minimum stress (**y**). Alternatively, click ... to open the **Select a Gasket** dialog box, and select a standard gasket material.

ASME Table 2-5.1 - Gasket List

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Self-energizing types (O rings, metallic elastomer, and other types considered as self-sealing)	0.00	0.00	II

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
Elastomers WO/ fabric or high % of asbestos fiber:			
Below 75A Shore Durometer	0.50	0.0	II
75A or higher Shore Durometer	1.00	200.0	II
Asbestos with Suitable Binder			
1/8" thick	2.00	1600.0	II
1/16" thick	2.75	3700.0	II
1/32" thick	3.50	6500.0	II
Elastomers W/cotton fabric insert	1.25	400.0	II
Elastomers W/ asbestos fabric insert:			
3 ply	2.25	2200.0	II
2 ply	2.50	2900.0	II
1 ply	2.75	3700.0	II
Vegetable fiber	1.75	1100.0	II
Spiral-wound, asbestos filled:			
Carbon	2.50	10000.0	II
Stainless, Monel, Nickel alloys	3.00	10000.0	II
Corrugated metal, asbestos ins. or corrugated metal, jacketed asbestos:			
soft aluminum	2.50	2900.0	II
soft copper or brass	2.75	3700.0	II
iron or soft steel	3.00	4500.0	II
Monel or 4%-6% chrome	3.25	5500.0	II
SS/nickel base alloys	3.50	6500.0	II
Corrugated metal:			

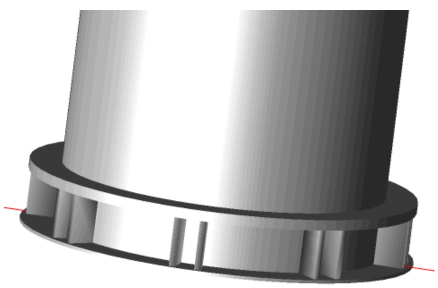
Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
soft aluminum	2.75	3700.0	II
soft copper or brass	3.00	4500.0	II
iron or soft steel	3.25	5500.0	II
Monel or 4%-6% chrome	3.50	6500.0	II
SS/nickel base alloys	3.75	7600.0	II
Flat metal, jacketed asbestos filled:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel	3.50	8000.0	II
4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	3.75	9000.0	II
Grooved metal:			
soft aluminum	3.25	5500.0	II
soft copper or brass	3.50	6500.0	II
iron or soft steel	3.75	7600.0	II
Monel or 4%-6% chrome	3.75	9000.0	II
SS/nickel base alloys	4.25	10100.0	II
Solid flat metal:			
soft aluminum	4.00	8800.0	I
soft copper or brass	4.75	13000.0	I
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I

Gasket Material	Gasket Factor m	Min. Stress y	Facing Column
SS/nickel base alloys	6.50	26000.0	I
Ring joint:			
iron or soft steel	5.50	18000.0	I
Monel or 4%-6% chrome	6.00	21800.0	I
SS/nickel base alloys	6.50	26000.0	I

NOTE *SS* refers to stainless steel

Skirt (Additional Element Data)

Defines additional data for a skirt support element with basering.



Skirt Diameter at Base

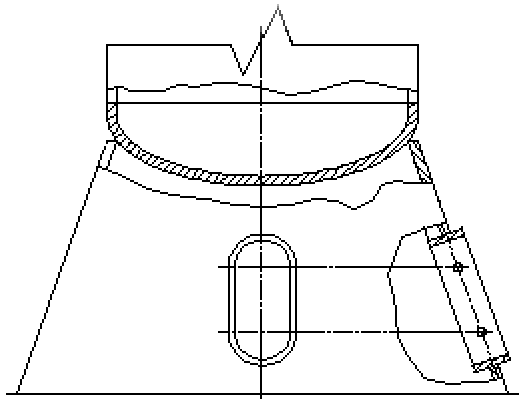
Enter the inside diameter at the bottom of the skirt. This value must be larger than or equal to the inside diameter at the top of the skirt. This option is only available when **Skirt** is selected for **Element Type**.

Perform Basering Analysis

If there is a basering on the skirt, select to specify basering parameters and calculate the basering design. The **Basering Dialog Box** (page 326) opens. This option is only available when **Skirt** is selected for **Element Type**.

Evaluate Holes in Skirt

Select to specify skirt access opening parameters and analyze the openings. Openings are analyzed when they are of reasonable size, are on tall towers, or are under sizable wind/seismic loads. This option is only available when **Skirt** is selected for **Element Type**.



This option opens the **Skirt Access Openings Dialog Box** (page 347).

For more information, see, *Skirt Opening Analysis Considerations* (page 341).

Basering Dialog Box

This dialog box defines data for baserings on a skirt element. Basering analysis performs thickness calculations and design for annular plate baserings, top rings, bolting, and gussets found on skirts for vertical vessels. These calculations are performed using industry standard calculation techniques. For more information, see *Basering Analysis Considerations* (page 326).

The Brownell and Young Method

The Brownell and Young method of design calculates the required thickness of the baseplate, the gussets, and the top plate or top ring. Brownell and Young discuss this method in their book *Process Equipment Design*. Dennis R. Moss also discusses it in the book *Pressure Vessel Design Manual*. This baseplate design method is based on the neutral axis shift method and generally results in a thinner basering design than the method discussed in *Basering Analysis Considerations* (page 326).

Basering Design Data Tab (Basering Dialog Box) (page 332)

Tailing Lug Data Tab (Basering Design Data Tab) (page 339)

Basering Analysis Considerations

Thickness of a Basering under Compression

The equation for the thickness of the basering is the equation for a simple cantilever beam. The beam is assumed to be supported at the skirt, and loaded with a uniform load caused by the compression of the concrete due to the combined weight of the vessel and bending moment on the down-wind / down-earthquake side of the vessel.

Thickness of a cantilever, t:

$$t = l * \sqrt{\frac{3\,f_c}{s_{allow}}}$$

Where

fc = Bearing stress on the concrete
l = Cantilever length of basering
s = Allowable bending stress of basering (typically 1.5 times the code allowable)

There are two commonly accepted methods of determining the stress from the vessel and base-ring acting on the concrete. The simplified method calculates the compressive stress on the concrete assuming that the neutral axis for the vessel is at the centerline.

Stress acting on the concrete, fc:

$$fc = -\left(\frac{W}{A} + \frac{Mc}{I}\right)$$

Where:

W = Weight of the vessel together with the basering
M = Maximum bending moment on vessel
A = Cross-sectional area of basering on foundation
c = Distance from the center of the basering to the outer edge of the basering
I = Moment of inertia of the basering on the foundation

However, when a steel skirt and basering are supported on a concrete foundation, the behavior of the foundation is similar to that of a reinforced concrete beam. If there is a net bending moment on the foundation, then the force upward on the bolts must be balanced by the force downward on the concrete. Because these two materials have different elastic moduli, and because the strain in the concrete cross section must be equal to the strain in the basering at any specific location, the neutral axis of the combined bolt/concrete cross section will be in the direction of the concrete. Several authors, including Jawad and Farr (*Structural Analysis and Design of Process Equipment,* pg 428 - 433) and Megyesy (*Pressure Vessel Handbook,* pg 70 - 73), have analyzed this phenomenon. The software uses the formulation of Singh and Soler (*Mechanical Design of Heat Exchangers and Pressure Vessel Components*, pg 957 - 959). This formulation seems to be the most readily adaptable to computerization, as there are no tabulated constants. Singh and Soler provide the following description of their method:

In this case, the neutral axis is parallel to the Y-axis. The location of the neutral axis is identified by the angle alpha. The object is to determine the peak concrete pressure (*p*) and the angle alpha.

For narrow base plate rings, an approximate solution may be constructed using numerical iteration. It is assumed that the concrete annulus under the base plate may be treated as a thin ring of mean diameter *c*. Assuming that the foundation is linearly elastic and the base plate is relatively rigid, Brownell and Young have developed an approximate solution which can be cast in a form suitable for numerical solution. Let the total tensile stress area of all foundation bolts be *A*. Within the limits of accuracy sought, it is permissible to replace the bolts with a thin shell of thickness *t* and mean diameter equal to the bolt circle diameter *c*, such that:

Thickness, t:

$$t = \frac{A}{Pl} * c$$

Where:

A = Total cross-sectional area of all foundation bolts
P = Peak concrete pressure

l = Width of basering
c = Thin ring diameter

We assume that the discrete tensile bolt loads, acting around the ring, are replaced by a line load, varying in intensity with the distance from the neutral plane.

Let *n* be the ratio of Young's moduli of the bolt material to that of the concrete; *n* normally varies between 10 and 15. Assuming that the concrete can take only compression (non-adhesive surface) and that the bolts are effective only in tension (untapped holes in the base plate), an analysis, similar to that given above, yields the following results:

$$p = \frac{2W + r_2 tcs}{(t_3 - t) r_1 c}$$

$$s = \frac{2(M - W r_4 c)}{r_2 r_3 t c^2}$$

$$\alpha = \text{acos}\left(\frac{s - np}{s + np}\right)$$

Where:

n = Ratio of elastic modulus of the bolt, Eb, to that of the concrete, Ec:

$$n = \frac{Eb}{Ec}$$

t_3 = Width of the basering, similar to the cantilever length, l, in Jawad and Farr's thickness equation previously mentioned
c = Bolt circle diameter
r_1 - r_4 = Four constants based on the neutral axis angle and defined in Singh and Soler's equations 20.3.12 through 20.3.17, not reproduced here.

These equations give the required seven non-linear equations to solve for seven unknowns, namely *p*, *c*, α, and the *ri* (*i* = 1 - 4) parameters. The iterative solution starts with assumed values of *s* and *p,* s_o and p_o, taken from an approximate analysis performed first. Then α is determined using the above equation. Knowing α the dimensionless parameters r_1, r_2, r_3, and *r4* are computed. This enables computation of corrected values of *p* and *s,* named p_o' and s_o'). The next iteration is started with s_1 and p_1 where we choose:

$$s_1 = 0.5\,(s_o + s_0')$$

$$p_1 = 0.5\,(p_0 + p_0')$$

This process is continued until the errors *ei* and *Ei* at the iteration stage are within specified tolerances --*ei = Ei = 0.005* is a practical value,

Where:

$$e_i = \frac{s_i' - s_i}{s_i}$$

$$E_i = \frac{p_i' - p_i}{p_i}$$

After the new values of bolt stress and bearing pressure are calculated, the thickness of the basering is calculated again using the same formula given above for the approximate method.

Thickness of Basering under Tension

On the tensile side, if there is no top ring but there are gussets, then there is a discrepancy on how to do the analysis. For example, while Megyesy uses Table F (*Pressure Vessel Handbook,* pg 78) to calculate an equivalent bending moment, Dennis R. Moss uses the same approach but does not give a table (*Pressure Vessel Design Manual,* pg 126-129), and Jawad & Farr use a 'yield-line' theory (*Structural Analysis and Design of Process Equipment,* pg 435-436). Since the Jawad and Farr equation for thickness, t, is both accepted and explicit, the program uses their equation 12.13:

Thickness, t:

$$t = \sqrt{\frac{3.91\ P_t}{s_y * (x + y + z)}}$$

Where:

Bolt Load, Pt:

$$P_t = s_{bolt,allow} * A_{bolt}$$

$$x = 2 * \frac{b}{a}$$

$$y = \frac{a}{2l}$$

$$z = d\ \left(\frac{2}{a} + \frac{1}{2l}\right)$$

s_y = Yield strength of the bolt
a = Distance between gussets
b = Width of base plate that is outside of skirt
l = Distance from skirt to bolt area
d = Diameter of bolt hole

Thickness of Top Ring under Tension

If there is a top ring or plate, its thickness is calculated using a simple beam formula. Taking the plate to be a beam supported between two gussets with a point load in the middle equal to the maximum bolt load, we derive the following equation:

Thickness, t:

$$t = \sqrt{6 * \frac{M}{s * Wt}}$$

Where:

Allowable stress, s:

$$s = 1.5\ s_{plate,allow}$$

Bending moment, M:

$$M = \frac{2 * Pt * Cg}{8}$$

Where:

Cg = Center of gravity, depending on the geometry of the plate

Bolt Load, Pt:

$$P_t = \frac{Allowable\,Stress}{Area}$$

Section Modulus, Z:

$$Z = \frac{W_t * t^2}{6.0}$$

Width of Section, Wt:

$$W_t = \left(\frac{D_o}{2}\right) - \left(\frac{D_s}{2}\right) - db$$

Required Thickness of Gussets in Tension

If there are gussets, they must be analyzed for both tension and compression. The tensile stress, T, is the force divided by the area, where the force is taken to be the allowable bolt stress times the bolt area, and the area of the gusset is the thickness of the gusset, t_{gusset}, times one half the width of the gusset, W_{gusset} (because gussets normally taper):

$$T = \frac{P}{A_{gusset}}$$

Where:

$$P = s_{bolt,allow} * A_{bolt}$$

$$A_{gusset} = t_{gusset} * \frac{1}{2} W_{gusset}$$

Required Thickness of Gussets in Compression

In compression (as a column) we must iteratively calculate the required thickness. Taking the actual thickness as the starting point, we perform the calculation in AISC 1.5.1.3. The radius of gyration for the gusset is taken as 0.289 t per Megyesy's *Pressure Vessel Design Handbook,* page 404. The actual compression is calculated as described above, and then compared to the allowed compression per AISC. The thickness is then modified and another calculation performed until the actual and allowed compressions are within one half of one percent of one another.

Basering Design

When you request a basering design, the software performs the following additional calculations to determine the design geometry:

- **Selection of Number of Bolts**

This selection is made on the basis of Megyesy's table in *Pressure Vessel Handbook* (Table C, page 67). Above the diameter shown, the selection is made to keep the anchor bolt spacing at about 24 inches.

- **Calculation of Load per Bolt**

  This calculation of load, Pb, per bolt:

  $$P_b = -\frac{W}{N} + \frac{2M}{NR}$$

  Where:

  W = Weight of vessel
  N = Number of bolts
  R = Radius of bolt area
  M = Bending moment

- **Calculation of Required Area for Each Bolt**

  This is the load per bolt divided by the allowable stress:

  $$A_r = \frac{P}{s}$$

- **Selection of the Bolt Size**

  The software has a table of bolt areas and selects the smallest bolt with area greater than the area calculated above.

- **Selection of Preliminary Basering Geometry**

  The table of bolt areas also contains the required clearances in order to successfully tighten the selected bolt (wrench clearances and edge clearances). The software selects a preliminary basering geometry based on these clearances. Values selected at this point are the bolt circle, basering outside diameter, and basering inside diameter.

Analysis of Preliminary Basering Geometry

Using the methods described previously for the analysis section, the software determines the approximate compressive stress in the concrete for the preliminary geometry.

Selection of Final Basering Geometry

If the compressive stress calculated above is acceptable then the preliminary geometry becomes the final geometry. If not, then the bolt circle and basering diameters are scaled up to the point where the compressive stresses are acceptable. These become the final basering geometry values.

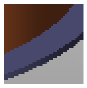
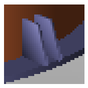
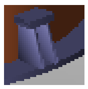
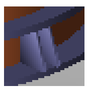
Analysis of Basering Thicknesses

The analysis then continues through the thickness calculation described above, determining required thicknesses for the basering, top ring, and gussets.

Basering Design Data Tab (Basering Dialog Box)

Basering Type

Select the type of basering design:

-  - Simple basering (without gussets)
-  - Basering with gusset plates
-  - Basering with gussets and top plate
-  - Basering with gussets and continuous top ring

Set Default Dimensions

Set default dimensions for all properties in the **Basering** dialog box based on the selected **Basering Type.**

Design Temperature

Enter the design temperature of basering. This value is used to determine the allowable stress values. When you select a value for **Basering Material**, the software uses this temperature to determine the operating allowable stress value for the material.

Basering Design Option

Select a value to either analyze an existing basering or design a new basering. Select **Analyze**, **Design**, **Brn & Young Analyze**, **Brn & Young Design**, or **ADM Analyze**. When a design option is selected, the software may change **Number of Bolts, Nominal Bolt Diameter, Basering ID,** and **Basering OD.**

Method for Thkness Calc

Select a method for basering thickness calculation:

- **Simplified or Steel on Steel** - Use for baserings located on a steel substructure.
- **Neutral Axis Shifted** - Use to design a thinner basering than the simplified method.

For a traditional basering on concrete, either method can be used.

NOTE If you select **Brn & Young Analyze** or **Brn & Young Design** in the **Basering Design Option** field, the software will only analyze your vessel using the **Neutral Axis Shifted** method.

E for Plates

If a basering with gussets is selected, enter the elastic modulus *E*, used to determine the allowable stress for plates in compression according to AISC. This is a required value. For most common steels, this value is 29 x 10^6 psi.

Sy for Plates

Enter the yield stress for the gusset plates at the design temperature of the base. For tables of yield stress versus temperature, see the ASME Code, Section II, Part D.

Gusset Thickness

Enter the thickness of the gusset plates used for this basering. Any allowances for corrosion should be considered.

Dist. Between Gussets

Enter the distance between the gussets. This dimension is used by the software to calculate the bending moment in the top plate. After the bending moment and bending stress are calculated, the software calculates the required thickness of the top plate.

Bottom Gusset Width

Enter the average width of the bottom gusset plate.

Top Gusset Width

Enter the average width of the top gusset plate.

Height of Gussets

Enter the gusset dimension from the basering to the top of the gusset plate. The forces in the skirt are transmitted to the anchor bolts through the gussets.

Top Plate Thickness

Enter the plate thickness for a basering with a top ring. If a value greater than **0.0** is entered, the software calculates the required thickness of the top plate. If no value is entered, the software does not perform top ring thickness calculations.

Top Plate Width

Enter the plate width for a basering with a top ring. This value is usually equal to the distance between the gusset plates plus two times the gusset plate thickness plus any additional width beyond the gussets.

Radial Width of Top Plate

Enter the radial width if the basering has a top plate. This value is: (top ring OD - top ring ID) / 2. This value must be entered and must be positive.

Bolt Hole Dia in Plate

Enter the diameter of the bolt hole that is in the plate.

Basering Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Basering Thickness

Enter the actual thickness of basering, not including allowances for corrosion or mill tolerance.

NOTE The software calculates the required basering thickness based on the selection in **Method for Thkness Calc**. The value enter here is only used for comparison.

Basering ID

Enter the inside diameter of the basering. The value must be greater than 0 and less than **Bolt Circle Diameter** and **Basering OD**. Enter a good approximation when **Analyze** or **Design** is selected for **Basering Design Option**. If you select **Design** for **Basering Design Option**, the software may change this value.

Basering OD

Enter the outside diameter of the basering. The value must be greater than **Bolt Circle Diameter** and **Basering ID**. Enter a good approximation when **Analyze** or **Design** is selected for **Basering Design Option**. If you select **Design** for **Basering Design Option**, the software may change this value.

Bolt Circle Diameter

Enter the diameter of the bolt circle. This is the diameter passing through the center of each bolt on the basering. The value must be greater than **Basering ID** and less than **Basering OD**. If you select **Design** for **Basering Design Option**, the software may change this value.

Corrosion Allowance

If needed, enter an external corrosion allowance. This allowance is added to **Basering Thickness** in the calculations.

Type of Threads

Select the thread series bolt table. Select **TEMA** (8-thread series), **UNC** (coarse threads, adapted from *Mark's*), **BS 3642**, or **SABS 1700 1996.** If you have a bolt that is outside of the bolt table ranges, select **User Root Area** and enter the nominal size in **Nominal Bolt Diameter**. This information can be obtained from a standard engineering handbook.

Nominal Bolt Diameter

Click ... and select the nominal bolt diameter from the bolt table selected for **Type of Threads**. If you have a bolt that is outside of the bolt table ranges, enter the nominal size and select **User Root Area** in **Type of Threads**.

For **TEMA** and **UNC** bolts, the available bolt diameters are:

Bolt Size (inches)	**Bolt Root Area**	
	(cm.^2)	**(in.^2)**
0.500	0.8129	0.126
0.625	1.3032	0.202
0.750	1.9484	0.302
0.875	2.7032	0.419
1.000	3.5548	0.551
1.125	4.6968	0.728

Bolt Size (inches)	Bolt Root Area	
1.250	5.9935	0.929
1.375	7.4516	1.155
1.500	9.0645	1.405
1.625	10.8387	1.680
1.750	12.7741	1.980
1.875	14.8645	2.304
2.000	17.1096	2.652
2.250	22.0838	3.423
2.500	27.6903	4.292
2.750	33.9290	5.259
3.000	40.7999	6.324
3.250	48.3031	7.487
3.500	56.4450	8.749
3.750	65.2130	10.108
4.000	74.6192	11.566

NOTE This information is adapted from Jawad and Farr, *Structural Analysis and Design of Process Equipment*, pg 425.

Number of Bolts

Enter the number of bolts required by the basering design. The number of bolts can be between 4 and 120. If you select **Design** for **Basering Design Option**, the software may change this value.

NOTE Bolts sizing is based on the maximum load per bolt in the operating case. The computation of the load per bolt is referenced in Jawad and Farr, equation 12.3, pg 422.

Bolt Material

Enter the name of the material. The software contains a database with most of the materials in ASME Code, Section II, Part D, Table 1A, 1B, and 3. **Matl...** allows a material to be selected directly from the **Material Database Dialog Box** (page 547). To modify the material properties of the selected element, click ▶ to open the **Material Properties Dialog Box** (page 611). Doing so only changes the properties of this element for this analysis. It does not modify the database.

Bolt C.A. (1/2 total)

Enter the bolt corrosion allowance. The software uses this value to corrode the radius of the root area and calculate a corroded root stress area based on the nominal bolt size and bolt table. This area is then used in the remainder of the bolt load/stress calculations.

The software calculates the required area of the bolt. If the bolt corrosion allowance is greater than zero, the software adds the corrosion allowance and recalculates the diameter based on the new required area:

Corroded Bolt Root Diameter = $(4 * \text{New Bolt Area}/\text{Pi})^{1/2} - 2 * \text{Bolt Corrosion Allowance}$

Bolt Root Area

Enter the root area of a single bolt if the basering design requires special bolts.

NOTE This option is mutually exclusive from the selection for **Basering Design Option**. If **Bolt Root Area** is available, the numbers from table 2 (UNC) are used. This value is used without modification by the software, so you should consider any corrosion allowance.

Bolt Shear Allowable

The allowable shear stress acting on the bolt, incorporating the bolt corrosion allowance.

Concrete Strength F'c/Fc <and> Modular Ratio Eplates/Ec

Enter the following for the concrete to which the base is bolted:

- **F'_c** - The nominal ultimate compressive stress of the concrete. This value is F'_c in Jawad and Farr or FPC in Meygesy. A typical entry is 3000 psi.
- **F_c** - The allowable compressive stress of the concrete
- **n** - The steel-to-concrete modulus of elasticity ratio, E_{plates}/E_c.

Average Values of Properties of Concrete Mixes (adapted from Brownell and Young)

Water Content (US Gallons per 94 lb Sack of Cement)	f'_c 28-day Ultimate Compressive Strength (psi)	f_c Allowable Compressive Strength = 0.45*f'c (psi)	n Modular Ratio (E_s/E_c)
7.5	2000	800	15
6.75	2500	1000	12
6	3000	1200	10
5	3750	1400	8

NOTE According to Jawad and Farr, Ec is equal to 57000 multiplied by the square root of f'_c psi. The modulus of elasticity of steel is assumed to be 30×10^6.

Lug Start Angle

Reassign the reference angle for positioning the lugs.

Use EIL Spec?

Select to use the EIL standard for basering design. This standard gives guidelines for selecting the number of bolts based on skirt diameter, as shown below:

Skirt Diameter (mm)	**Number of Bolts**
900	4
1200	8
1500	8
1800	12
2100	16
2400	16
2700	20
3000	24
3300	24
3600	28
4000	32

NOTE This option only applies if **Design** is selected for **Basering Design Option**.

Bolt Diameter

Enter a value when **Use EIL Spec?** is selected. This value provides additional tabular results for the standard thickness of basering components, such as the basering itself, gussets, top plate, and gusset height.

% Applied to Bolt Area * Bolt Stress

Enter a value for the percentage of force to apply to the bolt area times the bolt allowable stress in the calculation of the concrete stress. The default value of 100 percent generates the largest possible force. If the load cannot achieve this value, enter a smaller percentage.

Use AISI Design Method?

Select to perform calculations according to AISI Volume 2. The AISI (American Iron and Steel Institute) publishes *Steel Plate Engineering Data* volumes containing numerous useful calculations for steel structures including welding and thickness calculations for anchor bolt chairs.

Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?

Select to use AISC design manual 9th edition, Eqn. F3-1. The AISC (American Institute of Steel Construction) allows the use of 2/3rds the yield stress as the allowable for determination of the required thickness of the basering parts. A normal bending allowable is 1.5 times the allowable stress for the material at design temperature.

NOTES

- This option and **Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?** cannot be selected at the same time.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?

Select to use AISC design manual 9th edition, Eqn. F2-1. The AISC (American Institute of Steel Construction) allows the use of 75 percent of the yield stress as the allowable for determination of the required thickness of these parts. A normal bending allowable is 1.5 times the allowable stress for the material at design temperature.

NOTES

- This option and **Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?** cannot be selected at the same time.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use 1/3 Increase per ASIC A5.2?

Select to use AISC design manual 9th edition, Section A5.2. The AISC (American Institute of Steel Construction) allows a 1/3 increase in the allowable stress for parts that are subject to short-term sustained loads such as those due to wind or seismic.

NOTES

- This option is used in addition to the selection of **Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?** or **Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?**.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use Allowable Weld Stress per AISC Table J2.5?

Select to use AISC design manual 9th edition, Table J2.5. The weld allowable stress is then 0.3 times the minimum of the basering ultimate tensile strength and 58000 psi. If this option is cleared, the allowable weld stress is 0.4 times the minimum of the skirt and basering yield stress at ambient temperature.

NOTES

- This option is used in addition to the selection of **Use 2/3rds Yield for Basering/Top Plate Allowables per AISC F3-1?** or **Use 75% Yield for Basering/Top Plate Allowables per AISC F2-1?**.
- This section of AISC is used because ASME does not provide explicit design rules for baserings.

Use the skirt stress to determine the concrete stress for the simplified method?

Select this option to use the maximum skirt stress and the following formula to calculate the concrete stress:

$$s_{concrete} = \frac{s_{skirt} * t_{corroded,skirt}}{W_{baseplate}}$$

During the analysis of the skirt, the software calculates combination stresses due to different load types as outlined in the *Stresses due to combined loads* report. This option is only necessary when **Simplified** is selected for **Method for Thkness Calc.**

NOTES

- When **Use the skirt stress to determine the concrete stress for the simplified method?** is selected, **% Applied to Bolt Area * Bolt Stress** is inactive because this option is nearly identical to a percentage of 0.
- If **Neutral Axis Shifted** is selected for **Method for Thkness Calc**, then this option is not needed because the concrete stress is calculated in iterations. The concrete stress is presented for information.

Determine the Basering design bolt load accounting for Load Case Factors?

Select to calculate the bolt load based on moments and load case factors, depending on the local scalars used. The bolt load is calculated by the highest moment due to the bending stress from the different combination of loads.

IMPORTANT Be sure to select this option if you have also selected **Use and Allow editing of Local Scalars in the Load Cases** on the **Load Cases** tab.

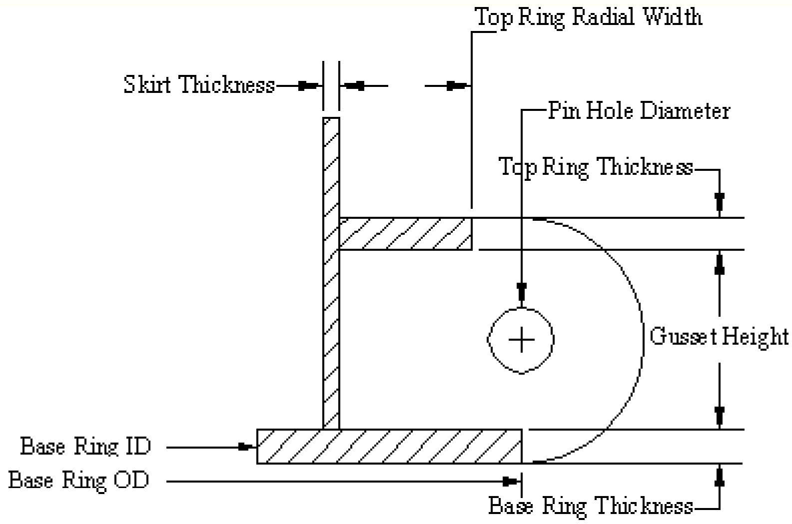
Plot

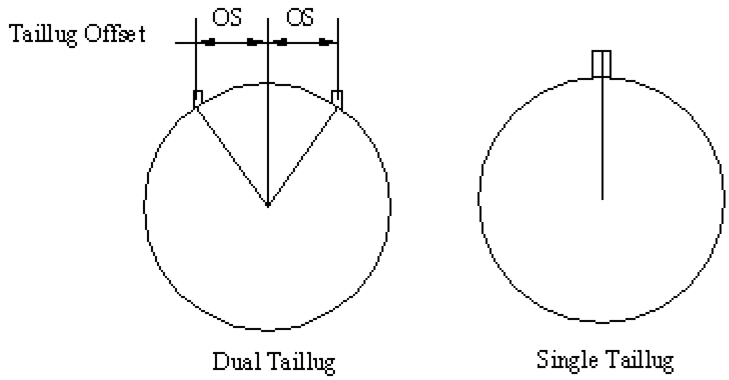
Click to open the **Basering Graphics** dialog box. A cross-section view of the basering design is shown.

Tailing Lug Data Tab (Basering Design Data Tab)

Tailing lug analysis is included in the basering analysis for a single or dual type design, as shown below. The design is based on a lift position where no bending occurs on the tailing lug. The location of the center of the pinhole is radial at the edge of the outer most of the top ring or the basering, whichever is larger. The tailing lug material is assumed to be the same material as the gusset or basering. The main considerations for the design are section modulus, shear and bearing stress at the pin hole, and weld strength.

NOTE For the software to generate the lifting load on the lug, the rigging data must be entered. To enter the rigging data, click **Installation | Misc. Options** on the **Load Cases** tab of the main window**.**





Perform Tailing Lug Analysis?

Select to enter data for the tailing lug analysis.

Type of Tailing Lug

Select **Single Lug** or **Dual Lug**, as shown above.

Centerline Offset

Enter the offset dimension from centerline *OS*. This entry is only valid for **Dual Lug**.

Lug Thickness

Enter the thickness of the tailing lug.

Pin Hole Diameter

Enter the pinhole diameter. The center of the pinhole is placed radially inline with the larger of the outermost edge of the top ring or the base ring OD.

Weld Size Thickness

Enter the leg size of the weld connecting the lug to the basering and the skirt.

Lug Height

Enter the tailing lug height measured from the top of basering. If you have a top ring, this would usually be the distance to the top ring.

Dist. from Skirt to Hole

Enter the distance from the OD of the skirt to the pin hole.

Analysis Results

The tail lug design consists of a three-part analysis: the base ring assembly (base ring, skirt and top ring), the strength of the weld, and the tail lug itself. The analysis assumes no bending in the tail lug. In the absent of the top ring, only the base ring and decay length (*e*) are considered for the section modulus calculation.

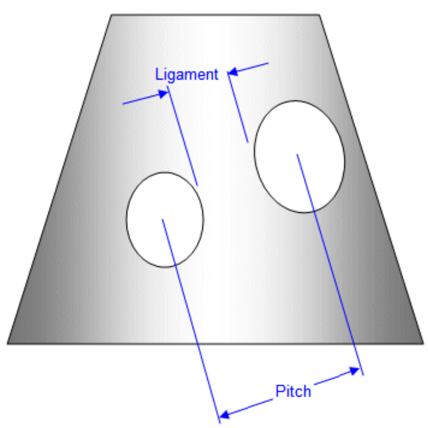
The following allowable stresses are used to check the design strength.

- Shear at the pin hole: 0.4 S_y
- Bearing stress: 0.75 S_y
- Weld stress: 0.49 S_{allow}

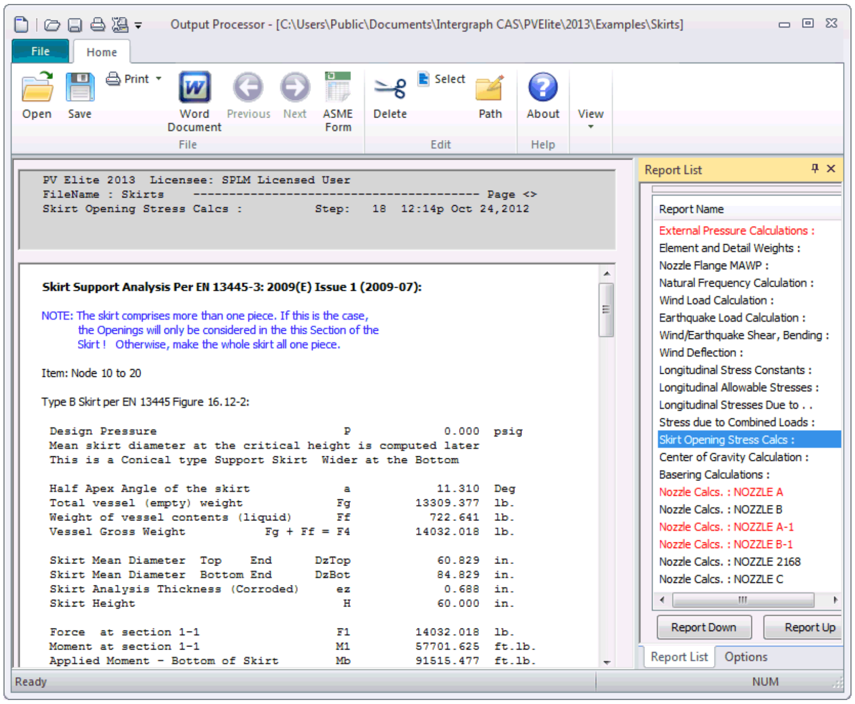
Skirt Opening Analysis Considerations

Pitch and Ligament

- **Pitch** is the center-to-center distance between adjacent holes.
- **Ligament** is the edge-to-edge distance between holes.



If the ligament is too small there is a possibility of collapse and buckling of the skirt. EN 13445, the code chosen as the basis for the analysis, is silent on this issue. This is because EN 13445 only gives consideration to one hole not the multiple holes that may be analyzed by PV Elite. The output processor includes a **Skirt Opening Stress Calcs** report. That report contains the full analysis results for all the openings that were included in the **Skirt Access Openings** dialog box:



IMPORTANT The results give a recommended minimum ligament distance, but you must evaluate whether this meets project requirements. See the following example:

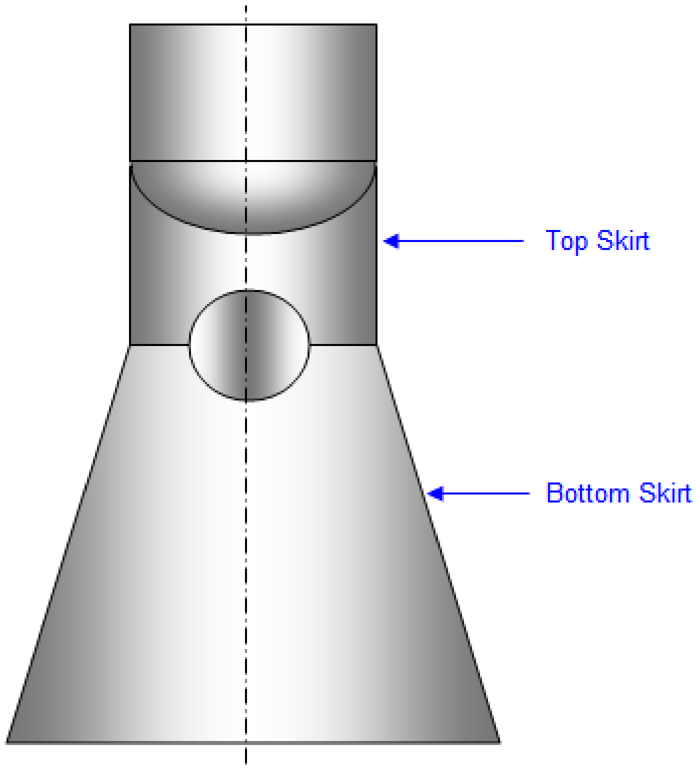
```
Summary of Stresses:
Stress              Actual         Allowed      Result    Equation
                     psi             psi                  Eqn 16.12
-----------------------------------------------------------------
Stot1pi            7808.51        42120.57       Pass        46
Stot1po            2251.14        42120.57       Pass        47
Stot1qi            9645.18        42442.72       Pass        48
Stot1qo           -1219.34        42442.72       Pass        49
Stot2pi           10717.20        42335.75       Pass        50
Stot2po           -1715.91        42335.75       Pass        51
Stot2qi            8063.63        42335.75       Pass        52
Stot2qo             937.67        42335.75       Pass        53
Stot3pi           -1338.29        43186.85       Pass        54
Stot3po            2448.57        43186.85       Pass        55
Stot3qi             727.67        43196.11       Pass        56
Stot3qo           -1331.37        43196.11       Pass        57

Skirt stress at the openings / critical section:
Sm4p                101.72        15000.00       Pass        70
Sm4q               -617.04        15000.00       Pass        71

The stresses are satisfactory

Analysis according to EN 13445 Section 8.7 allowable compressive stress [fc]:
 Note: This is a supplementary calculation not required by the code
 Py    = 2 * fz * ez / R = 2 * 15000.000 * 0.833 / 25.585 = 977.12 psig
 Pm    = 1.21*E*(ez/R)² = 1.21*29 *10^6*(0.833 /25.585 )² = 37225.29 psi
 Pm/Py = 37225.293 / 977.122 = 38.097
```

Vessels with Multiple Skirt Elements

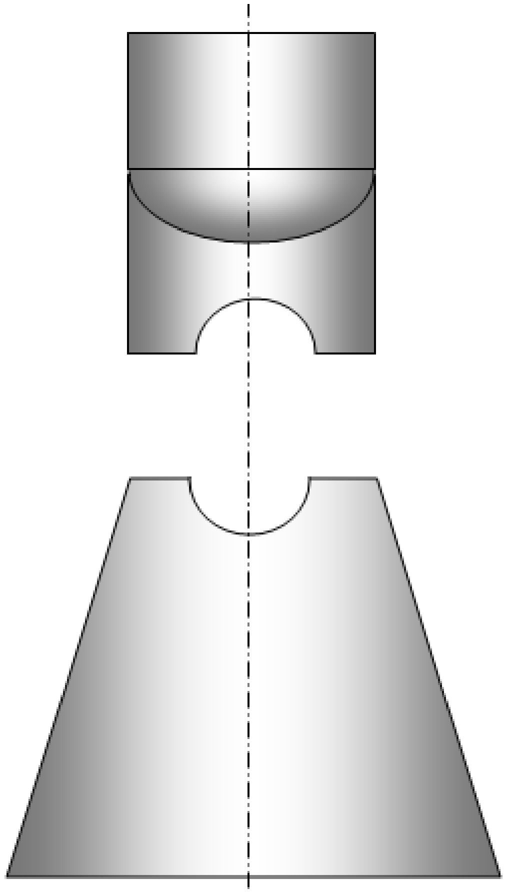
Complications arise when:

- A skirt is made from two pieces. That is, two skirts support the vessel.
- The two skirts have different thicknesses.
- The two skirts have different materials.
- The two skirts have different taper (conical) angles.

- An access opening or hole spans the skirt pieces. The opening cannot be considered as a single opening.



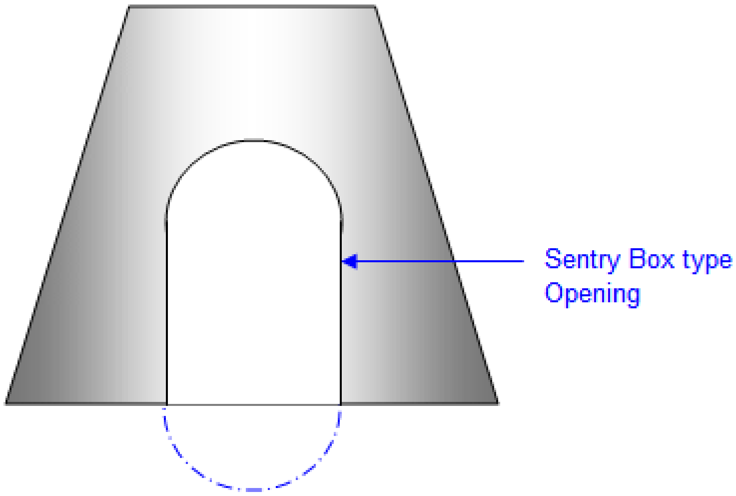
To properly analyze these situations, each skirt is created as a separate element:



1. In the bottom skirt, enter the hole geometry. The height of the center of the hole is located at or near to the top of the skirt and the top half of the hole is outside of the top node of the skirt. The software adjusts the analysis for this, but warns you that the opening is beyond the skirt.
2. In the top skirt, enter the hole geometry. In this case the height of the center of the hole is at or near zero at the bottom of the skirt. Again, the software adjusts the analysis for this, but warns you that the opening is beyond the skirt.

Sentry Box Openings

The software allows entry of oblong holes. With an oblong hole, sentry box geometry is possible:



The bottom of the oblong opening breaks the bottom of the skirt. The software adjusts the analysis for this, but warns you that the opening is beyond the skirt.

Basic Skirt Thickness

The required thickness of the skirt under tension and compression loads is determined using the same formula used for the compressive stress in the concrete, except using the thickness of the skirt rather than the width of the basering:

$$s = -\left(\frac{W}{A} + \frac{Mc}{I}\right)$$

Where:

W = Weight of the vessel during operation or testing
M = Maximum bending moment on the vessel
A = Cross-sectional area of the skirt
c = Distance from the center of the basering to the skirt (radius of the skirt)
I = Moment of inertia of the skirt cross section

In tension, this actual stress is simply compared to the allowable stress, and the required thickness can be calculated directly by solving the formula for t. In compression, the allowable stress must be calculated from the ASME Code, per paragraph UG-23, where the geometry factor is calculated from the skirt thickness and radius, and the materials factor is found in the Code external pressure charts. As with all external pressure chart calculations, this is an iterative procedure. A thickness is selected, the actual stress is calculated, the allowable stress is determined, and the original thickness is adjusted so that the allowable stress approaches the actual stress.

Stress in Skirt Due to Gussets or Top Ring

If there are gussets or gussets and a top ring included in the base plate geometry, there is an additional load in the skirt. Jawad and Farr (Structural Analysis and Design of Process Equipment, pg 434) have analyzed this load and determined that the stress in the skirt, s, due to the bolt load on the base plate is calculated as follows:

$$s = \frac{1.5F\,b}{\pi t^2 h}$$

Where:

F = Bolt load
b = Width of the gusset at its base
t = Thickness of the skirt
h = Height of the gusset

Jawad and Farr note that this stress should be combined with the axial stress due to weight and bending moment and should then be less than three times the allowable stress. They thus categorize this stress as secondary bending. The software performs the calculation of this stress, and then repeats the iterative procedure described above to determine the required thickness of the skirt at the top of the basering.

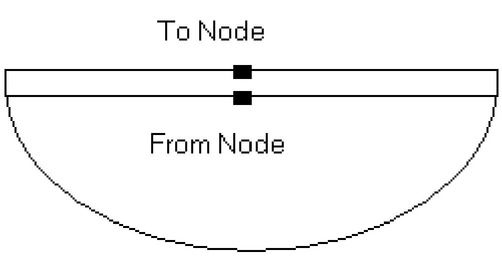
Skirt Access Openings Dialog Box

Defines data for access openings on a skirt element.

From Node

Displays the software-generated node number describing the starting location of the element. The **From Node** value for this element is also used to define starting locations for details such as nozzles, insulation, and packing that are associated with this element.

The software defines a vertical vessel from bottom to top. If the vertical vessel is on skirt, the first element is the skirt. If it is on legs or lugs, the first element is a head and the legs or lugs are defined as details on the appropriate head or shell elements.



The software defines a horizontal vessel from left to right. The first element in a horizontal vessel is usually a head, and the support saddles are defined as details on the appropriate shell elements.

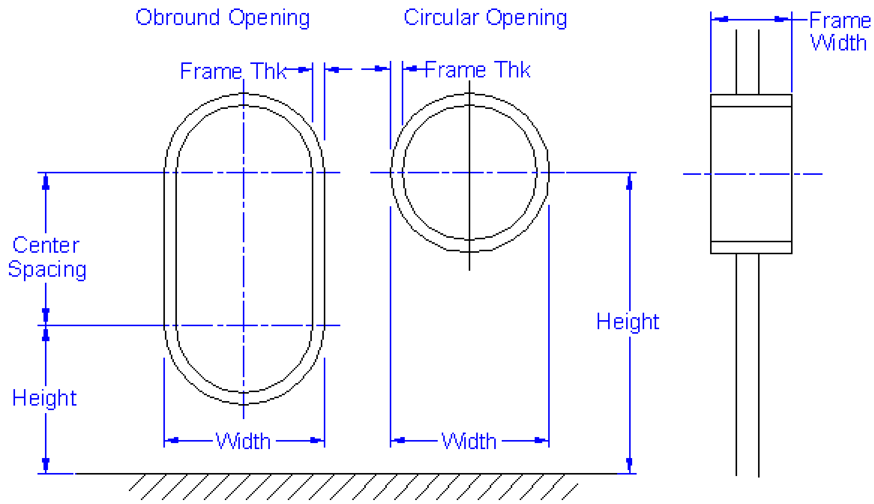
NOTE This value is not a function of the selected vessel code (such as, PD 5500, EN or ASME).

Number of Skirt Openings

Select the number of holes in the skirt. You can define up to ten holes.

Width

Enter the width of the opening.



Height

Enter the distance from the base of the skirt to the centerline of the opening.

Center Spacing

Enter the distance between centerlines on a rounded oblong opening.

Frame Thk

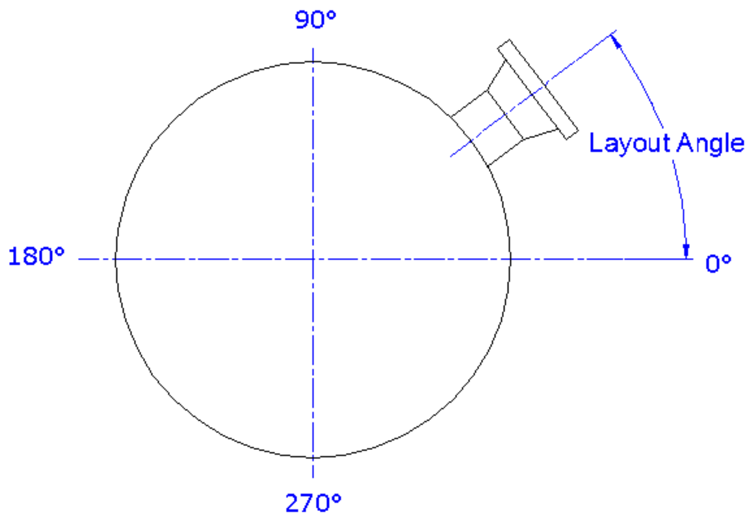
Enter the thickness of the opening reinforcement frame.

Frame Width

Enter the depth of the opening reinforcement frame.

Layout Angle

Enter the location of the opening around the skirt, when looking at a plan view of the skirt.



References

1. *Boiler & Pressure Vessel Code: Section VIII.* [S.I.]: ASME Publications, 2015. Print. Div. 1.
2. *Boiler & Pressure Vessel Code: Section VIII.* [S.I.]: ASME Publications, 2015. Print. Div. 2.
3. Brownell, Lloyd Earl, and Edwin H. Young. *Process Equipment Design: Vessel Design*. New York: Wiley, 1959. Print.
4. Farr, James R., and Maan H. Jawad. *Guidebook for the Design of ASME Section VIII Pressure Vessels*. New York: ASME, 1998. Print.
5. Jawad, Mann H., and James R. Farr. *Structural Analysis and Design of Process Equipment.* Second ed. New York: Wiley, 1989. Print.
6. Megyesy, Eugene F. *Pressure Vessel Handbook*. Eighth ed. Tulsa, OK: Pressure Vessel Handbook Pub., 1986. Print.
7. Moss, Dennis R. *Pressure Vessel Design Manual: Illustrated Procedures for Solving Major Pressure Vessel Design Problems*. Amsterdam: Gulf Professional Publ, 2004. Print.
8. Singh, Krishna P., and Alan I. Soler. *Mechanical Design of Heat Exchangers: and Pressure Vessel Components*. Cherry Hill: Arcuturus, 1984. Print.

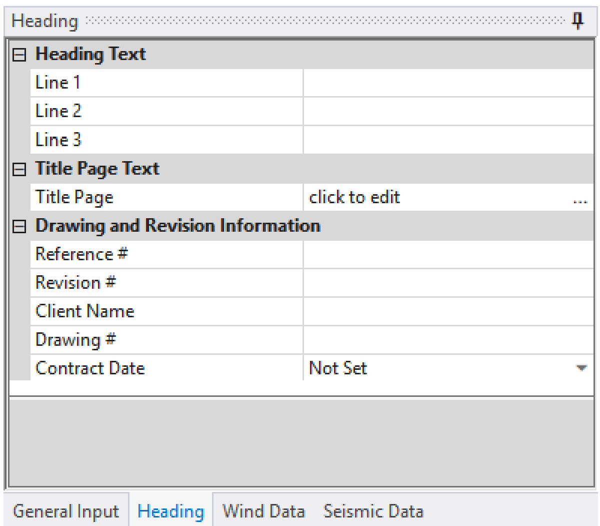
SECTION 12

Report Headings (Heading Tab)

Use the **Heading** tab to enter page heading text, title page text, and drawing and revision information to display on output reports. You can enter up to three lines of heading text to appear at the top of each page of the output report. The title page text displays on the title page of the output report, and the drawing and revision information displays on the output report cover sheet.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > Heading** .

   *The **Heading** tab appears at the bottom-left of the PV Elite window.*



Heading Text

Enter up to three lines of text to appear at the top of each printed page of the output report.

Title Page Text

Click ... to open the **Title Page Text** dialog box, and then enter text to display on the title page of the output report.

Set Title

Click to insert a default title page in the **Title Page Text** dialog box. The default title page is stored in the file *Title.Hed*, in the software subdirectory. You can edit the file.

Reference #

Enter the reference number of the model.

Revision #

Enter the revision number of the model.

Client Name

Enter the name of the client for whom the model is created.

Drawing #

Enter the drawing number of the model.

Contract Date

Enter the contract date in mm/dd/yyyy format, or click the drop-down arrow and select the date from the calendar that displays.

NOTE A header containing the Intergraph and PV Elite trademarks displays when you print a report or a model of the vessel to PDF. For more information about this header and how to customize it, see *PDF Header* (page 351).

PDF Header

PV Elite contains a header that displays when you generate output reports or print a model of the vessel to PDF. By default, this header contains the Intergraph and PV Elite trademarks, but you can customize it to include your company name and logo, or you can create your own custom header from scratch.

You can create your own custom header using a program like Microsoft Word® to make modifications to the OutputHeader.docx file located in the System folder. To access the OutputHeader.docx file, click **Edit Header** in the **Preview/Print** pane of the **Output Processor** menu. After you have modified your header, save the document in PDF format and replace the current OutputHeader.pdf file in the System folder.

NOTE If the text of an output report or the vessel image displays on top of the header, you can start the text lower on the page by adjusting the top margin using the **Page Setup** dialog. You can access the dialog by clicking **File > Page Setup** from the Output Processor.

Use the **Heading** tab in the PV Elite Input Processor to customize a text-based header and cover sheet for output reports. For more information, see *Customize report header* (page 545).

PDF Project Page

PV Elite includes a **Project Page** when you generate output reports to PDF. The **Project Page** is displayed at the beginning of the output reports and can be used as a graphical cover page. The software contains a default layout of the **Project Page**, but you can customize this page to include your own text and images.

To customize the **Project Page**, use a program like Microsoft Word® to make modifications to the Title_Page.docx file located in the System folder. After you have customized the document, save it in PDF format and replace the current Title_Page.pdf file in the System folder.

You can also include a custom **Project Page** for individual PV Elite files by placing a copy of the Title_Page.pdf in the local directory folder where your PV Elite file is saved. The software uses the Title_Page.pdf file located in the local directory first, and if it does not locate a copy of the file, it uses the file located in the System folder.

NOTE To remove the **Project Page** from displaying in PDF output reports, remove the Title_Page.pdf file from the System folder.

SECTION 13

Design Constraints Tab

Enter data such as pressures and temperatures, hydrotest data, and wall thicknesses in the **Design Constraints** tab. If no data is entered, the software uses the system defaults.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > Design Constraints**.

   *The **Design Constraints** tab appears at the bottom-left of the PV Elite window.*

In This Section

Design Data (Design Constraints Tab)

Vessel analysis uses the following set of design parameters.

Design Internal Press

Enter the design internal pressure for the vessel. This value is used as general design data and also with the UG-99b note 36 type hydrotest.

Design External Press

Enter the design external pressure for the vessel if the vessel is required to be rated for vacuum conditions.

Design Internal Temp

Enter the design internal temperature for the vessel. This value is only used by the input echo to help insure the correct design data was entered. It is not used by the analysis portion of the software.

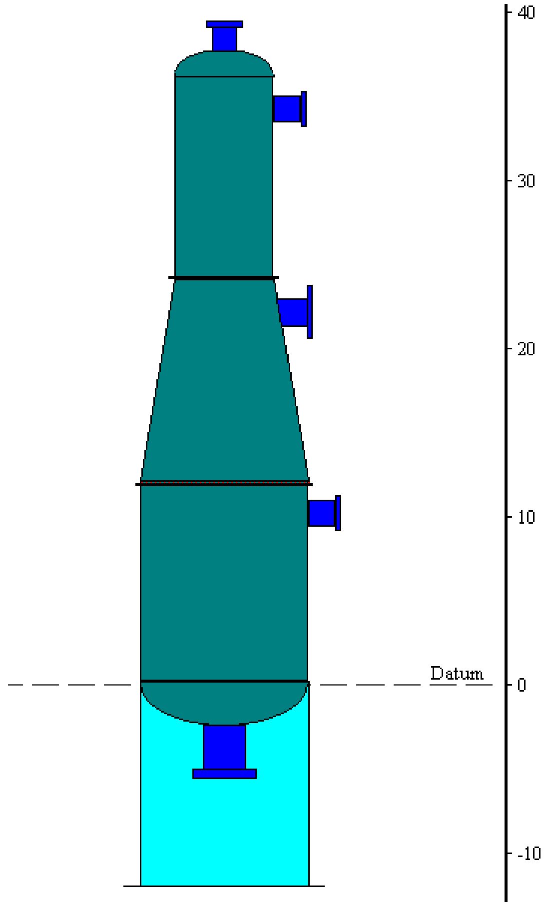
Design External Temp

Enter the design external temperature for the vessel if the vessel is required to be rated for vacuum conditions.

Datum Line Options

Click ... to display the **Datum Line Options** dialog box. For more information, see *Datum Line Options Dialog Box* (page 362).

Enter the location of the datum line for the **From Node** of the first element. You can then use **List Dialog**  on the **Auxiliary** toolbar to enter the distance of nozzles and platforms from the datum line.



Hydrotest Type

Select the type of hydrotest. For Division 1, the software provides the following methods to determine hydrotest pressure. Select one of the following:

- **UG-99b** - ASME UG-99 (b), Division 1. The hydrotest pressure is 1.3 times (1.5 for pre-99 addenda) the maximum allowable working pressure for the vessel multiplied by the lowest ratio of the stress value S for the test temperature to the stress value S for the design temperature. This type of hydrotest is normally used for non-carbon steel vessels for which the allowable stress changes with temperature, starting even at a somewhat low temperature.
- **UG-99c** - ASME UG-99(c), Division 1. The hydrotest pressure is determined by multiplying the minimum MAP by 1.3 (1.5 for pre-99 addenda) and reducing this value by the hydrostatic head on that part. The hydrostatic head is calculated based on the dimensions of the vessel and by values for **Projection from Top, Projection from Bottom, and Projection from Bottom Oper**. In addition, **Hydrotest Position** is used to determine the head pressure.
- **UG-99b(36)** - ASME UG-99(b), footnote 36, Division 1. The hydrotest pressure is 1.3 times (1.5 for pre-99 addenda) the design pressure for the entire vessel, multiplied by

the lowest ratio of the stress value Sa for the test temperature to the stress value S for the design temperature.

- **UG-100** - ASME UG-100 pneumatic test. The test pressure is 1.1 times (1.25 for pre-99 addenda) the maximum allowable working pressure for the entire vessel, multiplied by the lowest ratio of the stress value Sa for the test temperature to the stress value S for the design temperature.

  NOTE The stress ratio mentioned above includes bolt allowable stresses for flanges that are designed according to Appendix 2. This allowance usually results in a ratio of 1. See ASME Interpretation VIII-1-83-260 for more information. Click **Tools** > **Configuration** to turn off this option, resulting in a ratio greater than one in cases in which the operating and ambient stresses for the vessel parts are not the same.

- **No Hydro** - No hydrotest pressure.
- **User Entered Pressure**
- **1.43 * MAWP (PED)**
- **App. 27-4 -** ASME Appendix 27-4, Division 1. The hydrotest pressure is 1.3 times the maximum allowable working pressure for the vessel multiplied by the lowest ratio of the stress value S for the test temperature to the stress value S for the design temperature. This type of hydrotest is normally used for glass-lined vessels.

For Division 2, the software provides the following methods to determine hydrotest pressure. Select one of the following:

- **AT-300** - ASME AT-300, Division 2, based on vessel design pressure. The hydrotest pressure is 1.25 times the design pressure marked on the vessel, multiplied by the lowest ratio of the stress intensity value Sm for the test temperature to the stress intensity value Sm for the design temperature. This type of hydrotest is normally used for non-carbon steel vessels for which the allowable stress changes with temperature starting even at a somewhat low temperature.
- **AT-301** - ASME AT-301, Division 2, based on calculated pressure. A hydrostatic test based on a calculated pressure is allowed by agreement between the user and the manufacturer. The hydrostatic test pressure at the top of the vessel is the minimum of the test pressures calculated by multiplying the basis for calculated test pressure for each element by 1.25 and then reducing this value by the hydrostatic head on that element.
- **AT-410** - ASME AT-410, Division 2, based on vessel design pressure. The pneumatic test pressure shall be no less than 1.15 times the design pressure multiplied by the lowest ratio of the stress value S for the test temperature to the stress value S for the design temperature.
- **Hydrostatic**
- **Pneumatic**
- **No Hydro** - No hydrotest pressure.
- **User Entered Pressure**

Hydrotest Position

Select a hydrotest position. This input is required so that the total static head can be determined and subtracted when **UG-99c** is selected for **Hydrotest Type**. This value is used in conjunction with **Projection from Top**, **Projection from Bottom**, and **Flange**

Distance to Top to determine the total static head. Select from the following:

- **Vertical** - The vessel is tested in the upright or vertical position. This is not common.
- **Horizontal** - The vessel is tested in the horizontal position. This is common for most vessels. The vessel is on its side (in the case of a vertical vessel) or in its normal position (for a horizontal vessel).

Projection from Top

Enter the projected distance of the nozzle from the outer surface of the vessel *in its test position* to the face of the highest flange. This distance is added to the height (for vertical test positions) or to the maximum diameter of the vessel (for horizontal test positions) to determine the static head when **UG-99c** is selected for **Hydrotest Type**.

Projection from Bottom

Enter the projected distance of the nozzle from the outer surface of the vessel *in its test position* to the face of the lowest flange. This distance is added to the height (for vertical test positions) or to the maximum vessel diameter (for horizontal test positions) to determine the static head when **UG-99c** is selected for **Hydrotest Type**. This distance is essential when a nozzle flange projecting from the bottom of the vessel is governing the analysis.

Projection from Bottom Ope

Enter the projected distance of the nozzle from the outer surface of the vessel to the face of the flange. This distance is used to calculate the MAWP of an ANSI flange when it governs the MAWP. If the ANSI flange governs, then the static operating liquid head is subtracted from the basic rating of the flange.

Min. Des Metal Temp

Enter the specified minimum design metal temperature for the vessel. The software does not design any components based on this value. It is merely to help document the calculations and to be used for reference. This value is listed in the Internal Pressure Calculations report for comparison with the calculated UCS-66 minimum temperature.

No UG-20(f) Exemptions

Select this option if the vessel cannot take exemptions according to UG-20(f), for ASME VIII-1.

Flange Distance to Top

Enter the distance from the centerline/face of the nozzle flange to the top of the vessel. This value is used when the flange governs the MAP of the vessel. If the vessel is in the vertical position and **UG-99c** is selected for **Hydrotest Type**, this value is used in conjunction with **Projection from Top** to determine the total static head to subtract for the C type hydrostatic test.

Construction Type

Select the type of construction to be included on the name plate. Select **Welded**, **Press. Welded** (Pressure welded), **Brazed**, or **Resist. Welded** (Resistance welded).

Service Type

Select a type of special service in which the vessel is used. Select **None**, **Lethal** (Lethal service), **Unfired Steam** (Unfired steam boiler), **Direct Firing**, **Nonstationary**, **Air/Water/Steam**, **Sour**, **Severe Sour**, or **Amine.** This value is for information only; it is reported in the input echo.

Degree of Radiography

Select the symbolic representation of the degree of radiography. Select one of the following:

- **RT 1** - The complete vessel satisfies the full radiography requirements of UW-11(a) and the spot radiography provisions of UW-11(a)(5)(b) have not been applied.
- **RT 2** - The complete vessel satisfies the full radiography requirements of UW-11(a)(5) and the spot radiography provisions of UW-11(a)(5)(b) have been applied.
- **RT 3** - The complete vessel satisfies the spot radiography requirements of UW-11(b).
- **RT 4** - Only part of the vessel has met the other category requirements, or none of the other requirements are applied.
- **None**

This value is for information only; it is reported in the input echo.

NOTES

- **PD 5500 Non-destructive Testing Category 1**: 100% non-destructive testing (NDT) per section is required according to 5.6.4.1. There are no restrictions for vessels built under this category. The NDT is either (generally) radiography, or ultrasonic testing. The code must be consulted in all matters pertaining to this category. This is the strictest category.
- **PD 5500 Non-destructive Testing Category 2**: Limited random spot NDT according to section 5.6.4.2 is required. Generally about 10% of the welds must be examined by NDT (radiography or ultrasonic). There are restrictions when employing this category. Only certain groups of materials may be used (for example, groups 1, 2 and 4). The maximum thickness of the main components must not exceed 40 mm or 30 mm depending on the material group. It is essential that Table 3.4-1 be consulted when employing category 2. The code must be consulted in all matters pertaining to this category.

- **PD 5500 Non-destructive Testing Category 3**: Only visual inspection of the weld is required. However, steels are restricted to C and CMn types and materials in group 8. The maximum thickness is restricted to 13 mm for C and CMn steels, and 25 mm for materials in group 8. The design stress is much lower (see 3.4.2.2 for details). This category is restricted to temperatures no lower than 0°C. The code must be consulted in all matters pertaining to this category.

Table 6.6.1-1 — Testing groups for steel pressure vessels

Requirements	Testing group [a]						
	1		2		3		4 [b,j]
	1a	1b	2a	2b	3a	3b	
Permitted materials [g]	1 to 10	1.1, 1.2, 8.1	8.2, 9.1, 9.2, 9.3, 10	1.1, 1.2, 8.1	8.2, 9.1, 9.2, 10	1.1, 1.2, 8.1	1.1, 8.1
Extent of NDT for governing welded joints [e,h]	100 %	100 %	100 % - 10% [d]	100 % - 10 % [d]	25 %	10 %	0 % [k]
NDT of other welds	Defined for each type of weld in Table 6.6.2-1						
Joint coefficient	1	1	1	1	0,85	0,85	0,7
Maximum thickness for which specific materials are permitted	Unlimited [f]	Unlimited [f]	30 mm for groups 9.1, 9.2 16 mm for groups 9.3, 8.2 [i], 10	50 mm for groups 1.1, 8.1 35 mm for group 1.2	30 mm for groups 9.2, 9.1 16 mm for groups 8.2, 10	50 mm for groups 1.1, 8.1 30 mm for group 1.2	16 mm for groups 1.1, 8.1
Welding process	Unlimited [f]	Unlimited [f]	Fully mechanised welding only [c]		Unlimited [f]	Unlimited [f]	Unlimited [f]
Service temperature range	Unlimited [f]	Unlimited [f]	Unlimited [f]	Unlimited [f]	Unlimited [f]		Limited to (-10 to +300) °C for group 1.1, (-105 to +300) °C for group 8.1

Miscellaneous Weight %

Click ... to open the **Miscellaneous Weight Percent Inclusion** dialog box. This dialog box lets you specify to increase the mass of various elements and their details. There are three methods for applying the percentage of extra weight to use:

1. **Use a single percentage for all components** - Increases all metallic elements of the vessel by a certain percentage. In the **Miscellaneous Weight Percent for all Items** box, type a percentage value to include additional weight which accounts for vessel attachments and internal items not otherwise included in the vessel. Typical values are **3.0** or **5.0**. The software multiplies the total weight of the vessel by 1.0 plus this value converted to a decimal value (such as 1.03 or 1.05). Type **0** if no additional weight is needed.

2. **Use individual percentages for the different components** - Increases the weight of an item by the percentage specified for the item in the **Miscellaneous Weight Percentages** section.

3. **Add percentages to increase weights of shells and heads** - Increases the weights of the shells and heads by the sum of the percentage increase for nozzles, clips, and piping.

   NOTE If you use this option and model the nozzles or clips, the weight will be excessive.

   When using this option, the software calculates the weight of platforms automatically. You only need to specify the following information:

- **Is There a Top Head Platform? -** Enables the additional weight fields.
- **Top Head Platform Uniform Weight** - Specifies the grating weight from which the top head platform is constructed.
- **Circular Platform Uniform Weight** - Specifies the grating weight from which the circular platforms are constructed.
- **Ladder Uniform Weight -** Indicates the weight of the ladder per unit length.

NOTES

The top head platform, if specified, is square. The width the software uses to calculate the area is the larger of 4.5 feet (1.3716 meters) or the vessel inside diameter at the top head. Ultimately, the software adds the weight of the platform to the weight of the top head.

Circular platforms are placed every 20 feet (6.096 meters) along the height of the vessel. The platforms are 3.5 feet (1.0668 meters) wide and sweep 180 degrees. PV Elite calculates the weights of the platforms. The software then adds the value to the mass of the element on which the platforms would theoretically exist, based on the dimension from the base of the skirt.

The software also calculates the total weight of ladders along the vessel. In addition, PV Elite proportions the total ladder weight to each element based on the element's length.

Design Code

Displays the material design code for the project. This value is selected when you create a new project with **File** > **New** and can be changed by selecting a new value for **Design/Code** on the **Units/Code** toolbar. Available codes are **ASME SectionVIII-Divison 1**, **ASME SectionVIII-Divison 2**, **British Standard PD 5500**, or **EN-13445**.

User Defined MAWP

Enter a manually-defined value for maximum allowable working pressure to override the software-generated value. The software-generated MAWP is based on pressure ratings for the elements and ANSI flanges. If this value is zero, the software-generated MAWP is used. This is the default behavior.

User Defined MAPnc

Enter a manually-defined value for MAPnc to override the software-generated value. The software-generated MAPnc is based on pressure ratings for the elements and ANSI flanges. If this value is zero, the software-generated MAPnc is used. This is the default behavior.

User Defined Hydro. Press

Enter a manually-defined value for hydrostatic test pressure to override the software-generated value. The value is then used to calculate the stresses on elements subjected to this pressure. If this value is greater than 0, the software uses this pressure plus the applicable hydrostatic head that is computed based on the hydrotest position. If this value is zero, the software-generated hydrostatic test pressure is used. This is the default behavior.

Additional Ope. Static Press

Enter the additional static pressure at the top of the vessel.

Use Higher Long. Stress

Select to use higher allowable stresses for longitudinal stress calculations for wind and earthquake loadings. Loads are increased by an occasional load factor of 1.2.

ASME Section VIII, Division 1, Paragraph UG-23(d) permits the allowable stress for the combination of earthquake loading, or wind loading with other loadings to be increased by a factor of 1.2.

ASME Section VIII Division 2 A08 does not explicitly allow for an increase in allowable stresses, but this is subject to change.

Some wind and earthquake codes such as ASCE-7 98 and IBC 2000 and later have explicitly defined load combinations embedded in the standard. This set of codes essentially uses the same methodology to define earthquake loads. The load combinations either divide the earthquake load by a factor of 1.4 or multiply by 0.7 to convert from Limit State to Allowable Stress design. ASCE-7 2005, Chapter 13, paragraph 13.1.7 states that when a reference document "ASME Section VIII Division 1 in this case" provides a basis for earthquake design (UG-22), the reference document is used. This paragraph goes on to state that the loads shall be multiplied by 0.7. Please note that the IBC codes point directly to ASCE for wind and seismic load calculations.

NOTES

- In PV Elite versions prior to 2009, the allowables used for longitudinal stress calculations did not allow the 1.2 increase in tensile and compressive allowables for earthquake load cases, even if this option was selected. This restriction was removed in the 2009 version.
- For PD 5500 and EN-13445 the occasional load factor is not applied.

Consider Vortex Shedding

Select to perform vortex shedding calculations on tall, slender vertical vessels susceptible to wind-induced oscillations. This method is documented in the National Building Code of Canada and in texts on wind engineering. The software calculates fatigue stresses based on loads generated by wind vibration and the number of hours of safe operation remaining under vibration conditions. The software calculates the likelihood of this occurrence based on the research of Zorilla and Mahajan. If there is no possibility of wind vibration and you have this option selected, the software warns you that unrealistically high stresses will result and gives you the option to turn this calculation off.

For vertical vessels in which the height to diameter ration is less than 15, select the **No Vortex Shedding for H/D <= 15** field to not perform vortex shedding calculations.

Is This a Heat Exchanger?

If the Dimensional Solutions 3D file interface button is selected, also select this option to write geometry and loading information for this vessel design to the *<jobname>*.ini file created in the current working directory. See *Dimensional Solutions http://www.Dimsoln.com* for more information about the Dimensional Solutions product line. This entry is optional.

NOTE To completely define an exchanger it is necessary to enter in the required information regarding the tubes, tubesheets and the floating head (if any). With the exchanger data, PV Elite can then compute the weights and required thicknesses of the exchanger components. For more information, see **Tubesheet** (page 133).

Corroded Hydrotest

Select to use the corroded wall thickness when calculating stresses on the elements during the hydrotest, when it is necessary to hydrotest the vessel after it has corroded. If cleared, the software uses the uncorroded wall thickness

NOTE Longitudinal stresses due to hydrostatic test pressure are also calculated using the corroded wall thickness when this option is selected.

Hyd. Allowable is 90% Yield

Select to use 90 percent of the ambient yield stress as the hydrotest allowable stress. Clear to use the ASME Division 1 value, which is 1.3 times the ambient allowable stress *Sa* for the material. When the vessel is tested, the largest circumferential stress should not exceed this value. The software recalculates the hydrotest allowable each time this option is selected or cleared.

ASME Steel Stack

Select to perform an ASME steel stack analysis, based on the ASME recommended guidelines for Steel Stacks STS-2000 with addenda. This analysis is for circular stacks that meet the design requirements in the steel stack guidelines. The results are shown in the *ASME STS Stack Calculations* report. If **Design Code** is not set to **Division 1** (ASME VIII-1), the stack analysis is not performed.

Also select this option if you are analyzing a steel stack and want to check it against ANSI/ASME STS-2000/STS-1a-2003. After the software completes the calculation, the program generates the *Stress Due to Combined Loads* report with a listing of the stack calculations. Compressive allowables in the report are calculated based on Section 4.4.

When selected, expand **ASME Steel Stack** and enter values for **ASCE Wind Exposure**, **Factor of Safety**, **Mean Hourly Wind Speed**, **Is the Stack Lined?**, and **Importance Factor**.

IMPORTANT Read and understand the ASME stack guidelines. This is not a code like ASME Division 1 or 2, but a set of design guidelines for designers and engineers.

The following paragraphs from the stack guidelines are addressed:

- 4.4 Allowable Stresses
- 4.4.1 Longitudinal Compression, equations 4.7,4.8 and 4.9
- 4.4.2 Longitudinal Compression and Bending
- 4.4.3 Circumferential Stresses
- 4.4.4 Combined Longitudinal and Circumferential Compressive Stresses
- 4.4.5 Circumferential Compression in Stiffeners, equations 4.14, 4.15, 4.16
- 4.4.7 Minimum Structural Plate Thickness
- 5.2.2 Wind Responses, equations 5.3, 5.4 and (1),(2) and (3), (b) equations 5.5, 5.6 and 5.7

ASCE Wind Exposure

Select a value for wind exposure, taken from the ASCE #7 Wind Design Code. It is permissible to use another wind code or pressure versus elevation. However, in order to determine the Wind Responses, the Exposure Category (A,B,C or D) based on ASCE must be entered:

Exposure	Exposure Description
A	Large City Centers

B	Urban and Suburban Areas
C	Open Terrain with scattered obstructions
D	Flat, unobstructed areas exposed to wind flowing over open water for a distance of at least 1 mile.

The default value is **C**.

Factor of Safety

Enter a value for the factor of safety. Factors of safety from ASME STS, Table 4.1 are shown below:

F.S.	**Load Combination**
1.50	Dead + Live + Other + Thermal + Along or Cross Wind
1.50	Dead + Live + Other + Thermal + Seismic
1.33	Dead + Live + Other + Abnormal Thermal + Along Wind/4
1.33	Construction

The default value is **1.5**.

Mean Hourly Wind Speed

Enter the mean hourly wind speed at the stack location.

Is the Stack Lined?

Select this option to indicate that the stack lined with at least a 2 inch thick 100 pcf liner.

Importance Factor

Enter the ASCE-7/STS importance factor from Table I-3.

Category	**Nonhurricane-Prone Regions and Hurricane-Prone Regions with V = 85-100 mph and Alaska**	**Hurricane-Prone Regions with V > 100 mph**
I	**0.87**	**0.77**
II	**1.00**	**1.00**
III	**1.15**	**1.15**
IV	**1.15**	**1.15**

Datum Line Options Dialog Box

Specifies options for datum lines.

Datum is taken from bottom/left tangent or bottom of vessel

Select this option and type any offset from that point if the datum is to be taken from the bottom of the vessel or the left tangent.

For Vertical Vessels

Specifies where the datum line is to be positioned if the datum is not taken from the bottom/left tangent or bottom of the vessel. The available options include:

- Top Shell Seam or Top of Vessel
- Bottom Shell Seam or Bottom of the Vessel
- Base of Skirt

Offset (+ or -) distance from seam

Modifies the position of the datum line vertically from the seam based on the value in the **For Vertical Vessels** box.

For Horizontal Vessels

Specifies where the datum line is to be positioned if the datum is not taken from the bottom/left tangent or bottom of the vessel. The available options include:

- Left Shell Seam or Left End of Vessel
- Right Shell Seam or Right End of Vessel

Offset (+ or -) distance from seam

Modifies the position of the datum line horizontally from the seam based on the value in the **For Horizontal Vessels** box.

Design Modification (Design Constraints Tab)

Select Wall Thickness for Internal Pressure

Select **Yes** to automatically set the wall thickness for internal pressure. If the required element thickness for internal pressure exceeds the value of **Finished Thickness** defined for **Element Data** on the **General Input** tab, the software increases **Finished Thickness** to meet or exceed the thickness required for internal pressure. The software exceeds the required thickness only if you select **Round Thickness to Nearest Nominal Size** on the **Job Specific Setup Parameters Tab (Configuration Dialog)** (page 224) dialog box (**Tools** tab > **Set Configuration Parameters**). The software automatically calculates this wall thickness as model data is entered, so select this option before any part of the vessel is modeled. If the original value for **Finished Thickness** is greater than the required thickness, then no change is made.

NOTE During data entry, the software does not check the required thickness for flanges. That check is performed during analysis.

Select Wall Thickness for External Pressure

Select **Yes** to automatically set the wall thickness for external pressure. The **Equipment**

Installation and Miscellaneous Options Dialog Box (page 368) opens. If the required element thickness for external pressure exceeds the value of **Finished Thickness** defined for **Element Data** on the **General Input** tab, the software calculates the required thickness of each element (or group of elements) and increases the value of **Finished Thickness**. The software notifies you when the thickness is changed.

NOTE **Select Stiffening Rings for External Pressure** is set to **No** when this value is **Yes**.

Select Stiffening Rings for External Pressure

Select **Yes** to automatically set the wall thickness for external pressure. The **Equipment Installation and Miscellaneous Options Dialog Box** (page 368) opens. If the required element thickness for external pressure exceeds the value of **Finished Thickness** defined for **Element Data** on the **General Input** tab, the software calculates the required thickness of each element (or group of elements) and increases the value of **Finished Thickness**. The software notifies you when the thickness is changed.

NOTE **Select Stiffening Rings for External Pressure** is set to **No** when this value is **Yes**.

Select Wall Thickness for Axial Stress

Select **Yes** to automatically increase the thickness for axial stress in vertical vessels. The software calculates the axial stress and required thickness of each element (or group of elements) for longitudinal loadings (such as wind, earthquake, and weight of vertical vessels) and increases the value of **Finished Thickness**. The software exceeds the required thickness only if you select **Round Thickness to Nearest Nominal Size** on the **Job Specific Setup Parameters Tab (Configuration Dialog)** (page 224) dialog box.

SECTION 14

Load Cases Tab

Enter stress combination and nozzle pressure load cases. If no data is entered, the software uses the system defaults.

1. Select the **Home** tab.
2. In the **Input/Output** panel, select **Input > Stress Combination Load Cases**.

   *The **Load Cases** tab appears at the bottom-left of the PV Elite window.*

Reset Cases

Click to reset **Stress Combination Load Cases** to the default set of cases provided with the software.

Global Scalar for WI Loads

Enter a value for the wind scalar multiplier. After wind loads are generated, they are multiplied by this value. A value that is greater than one increases the loads. A value less than one decreases the loads. Enter zero if there are no wind loads. The > button to the right sets the value back to the default value.

In many building codes, load cases for wind and seismic are discussed. When the code uses allowable stress design, wind and seismic loads are usually scaled by a factor. For example, in the IBC Code, earthquake loads are divided by a factor of 1.4. Other codes multiply by the earthquake load by 0.7.

Global Scalar for EQ Loads

Enter a value for the seismic scalar multiplier. After seismic loads are generated, they are multiplied by this value. A value that is greater than one increases the loads. A value less than one decreases the loads. Enter zero if there are no seismic loads. The > button to the right sets the value back to the default value.

In many building codes, load cases for wind and seismic are discussed. When the code uses allowable stress design, wind and seismic loads are usually scaled by a factor. For example, in the IBC Code, earthquake loads are divided by a factor of 1.4. Other codes multiply by the earthquake load by 0.7.

Use and Allow editing of Local Scalars in the Load Cases

Select to use load case scalars in place of global scalars. Also select this option if you selected **Determine the Basering design bolt load accounting for Load Case Factors?** in the **Basering Dialog Box** (page 326) on the **General Input** tab. If this option is not selected, then values of 1.0 are used for scalar multipliers.

Set Load Cases to show WI and EQ Scalars

Click to show the scalar multipliers for **Global Scalar for WI Loads** and **Global Scalar for EQ Loads** in the load combination equations. These values change internally when the wind or seismic design code is changed. When version 2005 or earlier files are used, this value is automatically calculated.

In This Section

Stress Combination Load Cases (Load Cases Tab)

Enter the loads to combine for each load case. The software performs calculations for various combinations of internal pressure, external pressure, hydrotest pressure, wind load, and seismic load. You can define up to twenty combinations of these loadings for evaluation. Load cases are defined by a string of abbreviations showing the loads to be added. For example, "IP+OW+WI" is the sum of internal pressure plus operating weight plus wind. The software provides the following set of default load cases:

Load Case	**Load Combinations**
1	NP+EW+WI+FW+BW
2	NP+EW+EQ+FS+BS
3	NP+OW+WI+FW+BW
4	NP+OW+EQ+FS+BS
5	NP+HW+HI
6	NP+HW+HE
7	IP+OW+WI+FW+BW
8	IP+OW+EQ+FS+BS
9	EP+OW+WI+FW+BW
10	EP+OW+EQ+FS+BS
11	HP+HW+HI
12	HP+HW+HE
13	IP+WE+EW
14	IP+WF+CW
15	IP+VO+OW
16	IP+VE+EW

17	NP+VO+OW
18	FS+BS+IP+OW
19	FS+BS+EP+OW
20	

The following abbreviations are used for loads:

NP No Pressure

IP Internal Pressure

EP External Pressure

HP Hydrotest Pressure

EW Empty Weight

OW Operating Weight

HW Hydrotest Weight

WI Wind Load

EQ Earthquake Load

HE Hydrotest Earthquake

HI Hydrotest Wind

BW Bending Stress due to Lat. Forces for the Wind Case, Corroded

BS Bending Stress due to Lat. Forces for the Seismic Case, Corroded

BN Bending Stress due to Lat. Forces for the Wind Case, Uncorroded

BU Bending Stress due to Lat. Forces for the Seismic Case, Uncorroded

CW Axial Weight Stress, New and Cold (no corrosion allowance, CA)

WE Wind Bending Moment, New and Cold (Empty) (no CA)

WF Wind Bending Moment, New and Cold (Filled) (no CA)

FS Axial Stress due to Applied Axial Forces in Seismic Cases

FW Axial Stress due to applied forces in Wind Cases

Live loads (wind and earthquake) are calculated for the operating and hydrotest conditions. In both cases, the basic loads calculated are identical but the hydrotest live loads are usually a fraction of the operating live load. These hydrostatic fractions (percents) are entered in the live load definitions.

If **Consider Vortex Shedding** is selected on the **Design Constraints Tab (page 352),** the following loads may also be used:

VO Bending Stress due to Vortex Shedding Loads (Ope)

VE Bending Stress due to Vortex Shedding Loads (Emp)

VF Bending Stress due to Vortex Shedding Loads (Test, no CA.)

PV Elite allows individual load case descriptors to have their own scale factors. These factors scale the stresses produced by the corresponding load case component. For example 1.25EQ would produce an earthquake stress 1.25 times higher than the design earthquake stress. An example of a complete load case would be:

```
IP+OW+0.7143EQ+FS+BS
```

This facility allows designers to comply with a variety of loading scenarios. Another application of this may be that fractions of wind and seismic loads can be added together in the same load case. ASME states that doing this is not required; however, some design institutions mandate this practice. Here is another example:

```
0.7EQ+0.25WI+OW
```

Notice that there is no need to put a star (*) in front of each descriptor. If this box is not checked then values of 1.0 will be used for scalar multipliers. However, if there is a global scalar specified for wind or seismic, that value will be used. Please note that this is for vertical vessels only. During the stress calculations, the maximum stress is saved at the location of the support (skirt base, lug,or leg). Knowing the section properties, the moment needed to create that stress can be computed and used in the skirt, lug or leg calculation as required.

Any load case component can have a specified scalar. It is not meaningful to have a value in front of the NP component. It is important to specify NP for any case that does not have pressure.

For vertical vessels, the maximum stress is saved at the location of the support (skirt base, lug, and leg). Using the section properties, the moment needed to create that stress can be calculated and used in the skirt, lug or leg calculation as required.

NOTE It is often stated that the required thickness of the skirt is needed. It is not valid to directly calculate this value based on bending stress and axial stress because the section modulus is needed and the element OD or ID is still unknown. While it is possible to make an assumption, this will not generate an accurate mathematical result. A small change in the thickness can change the allowable compressive stress (factor A and factor B) in a very non-linear fashion. For more information, see British Code PD 5500, Annex B, paragraph B.1.5.

Vary Compressive Allowable for Internal/External Cases

Select to use the external design temperature to calculate the stress factor "B" for load combination cases that involve external pressure and dead weight. The software uses the design internal temperature to calculate the allowable compressive stress from the External Pressure Chart. By default, the software uses the maximum of the internal and external design temperatures to calculate the allowable compressive stress for operating-type cases. This is also true for cases involving internal pressure.

For example, a load case of "IP + OW + WI" uses the design temperature for internal pressure to calculate the allowable compressive stress. The load case "EP + OW + WI" uses the external design temperature to calculate the allowable compressive stress.

CAUTION When using this option, a disruption in process may leave the column at design internal temperature and a vacuum. If the design external temperature was much lower, this could lead to non-conservative results.

Corrode Case Components WE, WF and CW

Select to use loads **WE** (Wind Bending Moment Empty, no CA), **WF** (Wind Bending Moment Filled, no CA), and **CW** (Axial Weight Stress, no CA) in the corroded condition.

Installation | Misc. Options

Click ... to open the **Equipment Installation and Miscellaneous Options Dialog Box** (page 368) and specify options as to where items such as platforms, packing, and insulation are to be installed. These options are used to compute the center of gravity of the vessel in both the shop and field positions. The options are also used to compute weights such as operating weight and field test weight.

Fatigue Analysis

Click ... to open the **Fatigue Pressure/Cycle and UTS-Yield Data Dialog Box** and set options used to perform fatigue analysis on nozzles. In the dialog box, you enter values for **Number of Fatigue Cases to Process**, and then values for **Low Pressure**, **High Pressure**, and **# of Cycles** for each case. You must also select **Fatigue Calc?** in the **Nozzle Input Analysis** dialog box for each nozzle.

Equipment Installation and Miscellaneous Options Dialog Box

Define data for stiffening rings, saddles, legs, and lugs.

Platform Wind Area Calculation Method

Select the method for determining the surface area that a platform provides for wind load calculations. The selection is used to compute the wind area for all platforms specified in this project. Information on the force coefficient is also provided. Select one of the following:

- **Height * Width * Force Coefficient [Cf]**
- **1/2 Floor Plate Area * Force Coefficient [Cf]**
- **1/3 Height * Width * Force Coefficient [Cf]**
- **1/3 Projected Area * Force Coefficient [Cf]**

The software uses the area of the platforms to calculate forces applied to the vessel during wind loading analysis. Unfortunately, there is no standard method for computing the amount of area that a platform provides for wind load calculations.

NOTE If the selection is changed after one or more platforms are installed, click **List Dialog** to open the **Detail Listing Dialog Box** and then click **Platform Wind Area** to update the areas.

List Dialog

Click to open the **Detail Listing Dialog Box** and add details for platforms, nozzles, weights, packing, forces/moments, and pressure rings.

Adjust detail elevations by

Enter a value to shift the position of all details by the specified elevation distance. A negative value will move details down or left. A positive value moves the details up or right. This option is useful when all of the details such as rings, nozzles, and trays need to be adjusted by a specified amount. This may happen if an element is inserted into the model after it has been completed and the detail elevations need to be kept constant.

CAUTION If the adjustment moves a detail (such as a tray) into an element (such as a body flange), the software does not allow this, and the detail is lost and cannot be recovered.

But only for "From" nodes > or =

Enter the **From Node** number where you would like the change in position to start. All details on this element and the following elements are affected. A value of zero affects all elements.

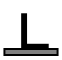
External Pressure Ring Requirement Data

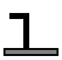
Structural Database

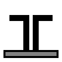
Select the structural database to use for external pressure ring stiffeners. Select **AISC**, **Korean/Japanese**, **German 1991**, **UK**, **Australian**, **South African**, or **India ISI**.

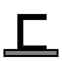
Stiffener Type to Meet Inertia Requirements

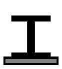
Select the stiffener type to use for external pressure rings on the vessel. For ASME VIII-1 and VIII-2, the software determines maximum stiffener spacing and adds rings to the vessel. Select one of the following:

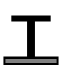

Equal Angle (flange in either direction)

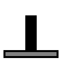

Unequal Angle (flange in either direction, "hard way" shown)


Double Large Angle
Double Small Angle
(with long or short sides back-to-back)


Channel


I - Beam


WT Section
MT Section
ST Section

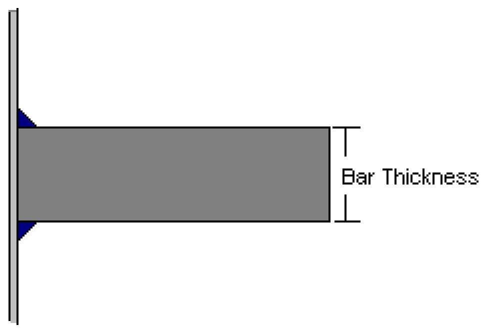

Bar
The software designs a ring with an aspect ratio of 10 to 1.00. The height of the ring is 10 times its thickness. The minimum starting ring width is 0.5 inches (12mm).

For angle sections, are they rolled the "Hard Way"?

Select if the stiffener selected in **Stiffener Type to Meet Inertia Requirements** is rolled the "hard way" to have the strong axis of the ring perpendicular to the vessel wall, with the flange away from the vessel wall.

Bar thickness to use when designing new rings

Enter a thickness when **Bar** is selected for **Stiffener Type to Meet Inertia Requirements.** The software uses the thickness to calculate the bar depth for the pressure ring. The software creates a ring with a 10 to 1 aspect ratio. In other words, the width of the ring is ten times the thickness. If no value is entered, the software uses a default thickness of 0.375 inches. (9.5 mm).



Select From Standard Bar Ring List

When **Bar** is selected for **Stiffener Type to Meet Inertia Requirements**, select this option to calculate the minimum bar pressure ring size that meets moment of inertia requirements of UG-29(a) (or Appendix 1-5 or 1-8 for a cone-cylinder junction ring). The bar ring is selected from the list below:

Ring Thickness (in.)	**Ring Widths (in.)**				
1/4	1.5	1.75	2.00	2.50	---
3/16	2.5	---	---	---	---
3/8	2.0	2.5	3.0	3.5	4.0
1/2	3.5	4.0	4.5	5.0	5.5
5/8	5.0	6.0	---	---	---
3/4	5.0	5.5	6.0	8.0	---
7/8	6.0	8.0	---	---	---
1	6.0	8.0	10.0	11.0	---

1 1/4	8.0	10.0	12.0	---	---
1 1/2	8.0	10.0	12.0	---	---
2	12.0	18.0	---	---	---
4	30.0	---	---	---	---

Rigging Data

Impact Factor (<1 NoCalc)

Enter a rigging impact factor. Typical values range from 1.5 to 2.0, but a maximum value of 3.0 is possible. The software performs a rigging analysis (combined shear plus bending stress) to account for sudden loads when the vessel is lifted from the ground. The impact factor effectively increases the overall weight of the vessel, and accounts for a skirt support and **Lug Distances from Base**. If the value is **0**, no rigging analysis is performed.

Lug Distances from Base

Enter the lifting lug elevation for the vessel. Enter an elevation distance, one in each text box, to account for two lifting lugs. The order of the elevations in the text boxes does not matter. These distances are measured from the bottom of the vertical vessel or from the left end of the horizontal vessel. If both values are **0**, no rigging analysis is performed.

Saddle Calc Options

Calc per which method

Select a saddle calculation method: **ASME Div 2** or **PD 5500**.

NOTE The software automatically sets the saddle calculation method to the appropriate option when you create a new job by selecting **Home** > **New** and choosing a code specification from the drop-down box. For example, when you select to create a new **British Standard PD 5500** job, the software sets **Calc per which Method** to **PD 5500**.

Use New Metal Weight for Saddle Calcs

Select to use the uncorroded new metal weight to determine saddle loads for the operating condition. By default, the software uses the corroded metal weight.

Number of intermediate supports to be used during the hydrotest

Enter a value from **0** to **20**. When some larger vessels are hydrotested after construction, a number of intermediate supports may be placed under the vessel to keep the saddle stresses below their allowables.

Installation Location

The location of installation of various components, the platform, packing tray insulation or lining, can be specified here, whether in the shop or the field.

Combine Wind and Earthquake Loads

This option allows for wind and earthquake loads to be combined for saddles, legs, and lugs.

Nozzle Design Options (Load Cases Tab)

Define additional load case parameters for nozzle design.

Nozzle/Clip Design Pressure Options

Select one of the following design pressure methods for calculating the pressure at the nozzle:

- **1. MAWP + Static Head to Element Bottom** - Calculates the internal pressure on the nozzle on the bottom of the element where the nozzle is located. This pressure is the maximum allowable working pressure (MAWP) of the vessel plus the static liquid pressure head to the bottom of that element. Thus, the design pressure can vary for nozzles located on different elements. This option is appropriate if you are certain that your nozzle locations will not vary during the design process. If you use this option and a nozzle is lowered in the vessel and under additional pressure due to liquid head, you must rerun the analysis in order to determine if the nozzle geometry is satisfactory.
- **2. Design P + Static Head** - Calculates the exact internal pressure at the nozzle location. The pressure is the design internal pressure plus the additional static liquid pressure at the nozzle location. This option is appropriate for re-rating vessels or for the design of new vessels where there are no MAWP considerations. If the overall MAWP of the vessel is to be determined, it is strongly recommended that the model be rerun with the computed MAWP to be sure that all components pass at this higher pressure, which will be stamped on the nameplate. The pressure for all elements can be changed at **Design Internal Press** on the **Design Constraints Tab** (page 352).
- **3. Overall MAWP + Static Head (governing element)** - Calculates one design internal pressure for all of the nozzles located on the vessel, based on the static liquid pressure to the bottom of the element that is governing the MAWP. If the nozzle location on a vessel changes due to a client request, there is no need to rerun nozzle calculations because the pressure used in the calculations does not change. This method is ideal for designing new vessels and is generally the most conservative option.
- **4. MAWP + Static Head to Nozzle** - Calculates the MAWP of the vessel and then adds the static liquid pressure from the liquid surface to the nozzle location. For nozzles at different elevations, the design pressure will vary.

  NOTE If the bottom head is governing the MAWP, this will be the most conservative option.

NOTE If the resulting nozzle reinforcement MAWP does not need to govern the MAWP of the vessel, options 1, 3 or 4 should be used. This is a common requirement for vessels that are used in the chemical and petro-chemical industries.

Consider MAPnc

Select to require that nozzle reinforcement calculations are performed for the MAP new and cold condition. The software checks to see if the nozzle is reinforced adequately using the MAPnc generated during internal pressure calculations. When the area-of-replacement calculations are made for this case, cold allowable stresses are used and the corrosion allowance is set to 0. Designing nozzles for this case helps the vessel to comply with UG99 or appropriate (hydrotest) requirements. Check your design requirements to see if this case is required by your client.

Consider External Loads for Nozzle Tr

Select to calculate the nozzle area of replacement requirements using the required thickness of the shell. This value, *tr*, is critical in the ASME code. The software determines the maximum thickness based on the highest stress ratio and uses that value if it governs over the required thickness based on internal or external pressure. There are cases where pressure requirements do not govern the value of tr. This can occur when a nozzle is located near the bottom of a tall vertical vessel. If there is a high wind load or seismic load on the structure, bending stress can govern the required thickness of the shell section. If this is the case, then the value of tr (per UG-22 Div. 1) should be based on the controlling factor.

NOTE Optionally, if tr needs to be specified for a specific nozzle, the value can be entered directly to **User Tr** on the **Nozzle Input/Analysis Dialog Box** of **Nozzle (page 60)**.

Use Appendix 1-9 (Div. 1)

Select to use ASME Code Case 2168. On February 14, 1994 ASME approved case 2168, providing an alternative method for reinforcing radial nozzles in cylindrical shells. The nozzle must be connected to the cylindrical shell by a full penetration groove weld.

Nozzle Sort Options

Select a sort order of nozzles for the Nozzle Schedule report. Select **By Name Ascending**, **By Name Descending**, **By Diameter, Ascending**, **By Diameter, Descending**, or **No Sorting**.

ASME Large Nozzle Calc Options

Select the load cases to use for evaluation of large openings. Select **Use 1-7** or **Use 1-10**.

ANSI Flange Pressure Reduction Options

Select a method for ANSI flange pressure reduction. Several methods are available to de-rate the flange MAWP based on external loads. If flanges are externally loaded they have the potential to leak. To keep this from occurring, it might be necessary to choose a heavier class of flange than one that is good for the design pressure per the B16.5/47 standard.

NOTE At the time of this writing (November 2017), the ASME Code has no rules on a particular method to use. They do however state (in a Code Case) that the external loadings must considered.

- **Kellogg Method** - The Kellogg method is well known and conservative. The axial load and moment are used to compute an equivalent pressure that is then deducted from the flange rating from the B16/47 table.
- **PVP Method** - The PVP Method is based on the paper PVP 2013-97814 with some modification. PV Elite uses the Sustained load category with a factor of 32 on Me instead of 16 that is based on operating loads. Subsequent unpublished work based on this method uses the value of 32. Sustained forces and moments must be entered for those results to be meaningful. Otherwise, the computed flange rating is zero.
- **50% Stress Method** - If the computed stress/allowable stress is < 0.5 on the pipe wall, then the allowable pressure is the full rating from the ANSI/ASME standard. If the stress ratio is >= 0.5, then the full equivalent pressure based on the Kellogg method is subtracted from the flange rating.
- **DNV Method** - The DNV method is considered to be a bit unconservative. It is essentially 1.3 times the flange rating minus the equivalent pressure based on the

Kellogg method. The idea is that because the flanges will be hydrotest at an elevated pressure and because there will loading applied (flanges in the piping system), then their rating can be elevated using the above equation. Most piping is tested to 1.5 times the design pressure, but we use a factor of 1.3 for conservatism and because 1.3 is the factor used in Division 1 for hydrotesting pressure vessels.

SECTION 15

Wind Loads (Wind Data Tab)

Click **Input** > **Wind Loads** to enter wind code data. The **Wind Data** tab appears on the left. Select a wind design code and enter data required by that code. If no data is entered, the software uses the system defaults.

Wind Design Code

Select the design code to use for wind calculations:

Code	Description
As/Nz 1170:2002 **An/Nz 1170.2:2011**	Design Wind Code of Australia and New Zealand, 2002 and 2011 editions For more information, see *As/Nz 1170:2002 & As/Nz 1170.2:2011 Wind Data* (page 378).
ASCE-93 **ASCE-95** **ASCE-98/02/05/IBC-03** **ASCE-2010** **ASCE-2016**	American Society of Civil Engineers Standard 7 (formerly ANSI A58.1). For more information, see: *ASCE-93 Wind Data* (page 385) *ASCE-95 and ASCE-98/02/05/IBC-03 Wind Data* (page 387) *ASCE-2010 Wind Data* (page 388)
BS6399-97	Britain's Wind Design Code, 1997 Edition (Replaces CP-3). For more information, see *BS6399-97 Wind Data* (page 394).
Brazil NBR 6123	Design Wind Code for Brazil. For more information, see *Brazil NBR 6123 Wind Data* (page 390).
China GB 50009	Design Wind Code for China. For more information, see *China GB 50009 Wind Data* (page 397).
EN-2005 **EN-2010**	Design Wind Codes for several European Countries with a 2005 and 2010 editions. For more information, see *EN-2005 and EN-2010 Wind Data* (page 399).
Euro Code	Design Wind Code for several European countries, including France. For more information, see *Euro Code Wind Data* (page 400).
IBC 2006 **IBC 2009** **IBC 2012** **IBC 2015** **IBC 2018**	International Building Codes. For more information, see *IBC 2006, 2009, 2012, 2015, and 2018 Wind Data* (page 404)

IS-875	Wind Design Code of India, 1987 edition Amd. 1 and 2 (2003). For more information, see *IS-875 Wind Data* (page 406).
JPI-7R-35-2004	Japanese Wind Code. For more information, see *JPI-7R-35-2004 Wind Data* (page 408).
KBC 2016	Korean Wind Code. For more information, see *KBC 2016 Wind Data* (page 409).
Mexico	Official Design Wind Code of Mexico, 1993 and 2008. For more information, see Mexico Wind Data.
NBC-95 **NBC-2005** **NBC-2010** **NBC-2015**	National Building Code of Canada. For more information, see: *NBC-95 and NBC-2005 Wind Data* (page 425) *NBC-2010 Wind Data* (page 421) *NBC-2015 Wind Data* (page 423)
SANS 10160-3:2010	South African National Standard 10160, Section 3, 2010 edition. For more information, see *SANS 10160-3:2010 Wind Data* (page 427).
UBC	Uniform Building Code. For more information, see *UBC Wind Data* (page 429).
User Defined	Enter your own wind pressure versus elevation in the Wind Profile Table.
No Wind Loads	

Special Effective Wind Diameter

To open the *Special Effective Wind Diameter Dialog Box* (page 377) dialog, press the 

button next to the **Wind Design Code** field on the Wind Design tab.

Percent Wind for Hydrotest

Enter the percentage of the wind load (not wind speed) that is applied during the hydrotest. This is typically as low as 33 percent of the design wind load, because it can be assumed that the vessel will not be hydrotested during a hurricane or severe storm.

Beta: Operating/Empty/Full

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

In This Section

Special Effective Wind Diameter Dialog Box

In many cases it is necessary to design a vertical column for the purpose of determining foundation loads without knowing much of anything about the vessel other than its diameter, height, location, and a few other parameters. The **Special Effective Wind Diameter** dialog box allows you to effectively and conservatively design the column for wind loads based on many years of EPC knowledge.

To open the **Special Effective Wind Diameter** dialog box press ... next to the **Wind Design Code** field on the **Wind Design** tab. By providing the values for the inputs on the dialog box, the effective diameter over which the wind acts is computed by a proprietary formula.

NOTE When using this feature, the software assumes there are no platforms modeled in PV Elite. You should either model the platforms, use an appropriate wind diameter multiplier, or use this dialog box, but not all three. Additionally, you should set the wind diameter multiplier on each element to **1** if you are using this method for a typical model, although there may be exceptions.

This design method works in conjunction with the specifications made in the **Miscellaneous Weight %** field. In addition to computing an acceptable effective wind diameter, it is also necessary to compute an appropriate vessel weight, by accounting for items like nozzles, clips, and platforms.

Calculate Windload Diameter for Models with no Platforms

Select this option to enable the additional fields on the **Special Effective Wind Diameter** dialog box and indicate to the software that this requirement should be considered during analysis.

Calculate Effective Diameter for Wind Diameter Multipliers that are not 1

Select this option to allow the software to calculate the effective diameter when the wind diameter multiplier is not 1. For example, select this option if there are helical strakes on a section of the column that require their wind load diameter calculations be made using the traditional method.

Factor to Account for Miscellaneous Attachments

This value is typically 1.1 and directly increases the vessel diameter. This accounts for miscellaneous attachments. It is very similar to the **Wind Diameter Multiplier** on the **Element Data** input tab.

Wind Projection Factor

This value is typically 36 inches (915 mm) and linearly increases the effective diameter result.

Insulated Diameter of Overhead Line

Enter the outside diameter of the overhead line pipe plus two times the insulation thickness on the pipe (if there is any). Most process columns have piping that cause wind drag. Overhead lines run down the side of column and are typically restrained to the vessel by various support mechanisms and also by the nozzle connection. The wind load on the piping is transferred back to the vessel and then to the foundation.

As/Nz 1170:2002 & As/Nz 1170.2:2011 Wind Data

The principles of the Australian/New Zealand code are basically the same as other wind codes. A moving air mass causes forces to be exerted on anything that obstructs its path. The starting point of any wind action analysis starts with the Bernoulli equation that converts the kinetic energy of the wind into a static pressure. This pressure simply attempts to move anything in its path. In the case of a vertical tower, the wind deflects the tower in the direction of the flowing wind. Because of the friction, the wind (which is a fluid) passes around the vessel, creating small eddie currents called vortices on either side of the vessel. The spinning air loses some of it pressure as its angular kinetic energy increases. This is again an effect predicted by the Bernoulli equation. This spinning eddie, or vortex, causes a lowering of the pressure on one side of the vessel. As these vortices occur alternatively on the sides of the vessel, the tower tends to sway from side to side perpendicular to the direction of the wind, vibrating like an upside-down pendulum.

The wind pressure causes a static deflection in the direction of the wind stream, and the vortices cause the vessel to vibrate at right angles to the wind stream (vector) as mentioned above.

Static deflection and dynamic vibration cause the vessel to experience longitudinal, or axial bending stresses. Both the stresses and deflection must be limited to provide for good design.

As stated above, the wind kinetic energy is converted to pressure, but there are other factors influencing the final wind pressure. Among these factors are (but not limited to) the following:

- The height, or altitude, of the tower or vessel in the wind stream.
- The elevation of the site on which the vessel is installed.
- The base elevation of the vessel if is supported on an elevated platform or structure.
- The effect of the surround geography, such as hills and escarpments.
- The effect of upstream buildings.

To perform a reasonable analysis, you must enter sufficiently accurate information for the software to apply the correct factors and special calculations for specific cases.

Design Wind Speed

Enter the design wind speed V_r . This is the regional wind speed, as described in section 3.2, table 3.1 of the code:

Regional wind speed (m/s)	**Region**				
	Non-cyclonic			**Cyclonic**	
	A (1 to 7)	**W**	**B**	**C**	**D**
V_5	32	39	28	F_C33	F_D35
V_{10}	34	41	33	F_C39	F_D43
V_{20}	37	43	38	F_C45	F_D51
V_{25}	37	43	39	F_C47	F_D53
V_{50}	39	45	44	F_C2	F_D60
V_{100}	41	47	48	F_C56	F_D66
V_{200}	43	49	52	F_C61	F_D72
V_{500}	45	51	57	F_C66	F_D80
V_{1000}	46	53	60	F_C70	F_D58
V_{2000}	48	54	63	F_C73	F_D90
V_R (see NOTE)	$67-41R^{-0.1}$	$104-70R^{-0.045}$	$106-92R^{-0.1}$	F_C x ($122-104R^{-0.1}$)	F_D x ($156-142R^{-0.1}$)

NOTE Round the calculated value to the nearest 1 m/s.

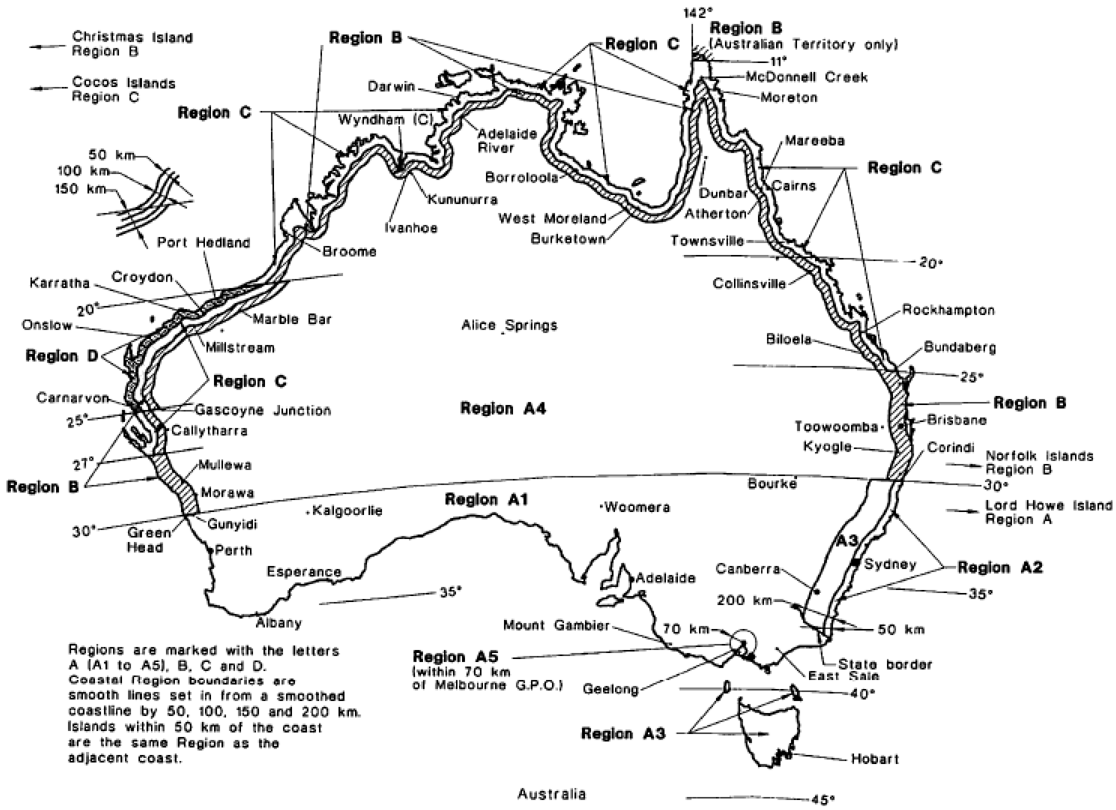
If you are entering the wind speed in other units, remember to convert the above wind speeds from m/s to your specific units:

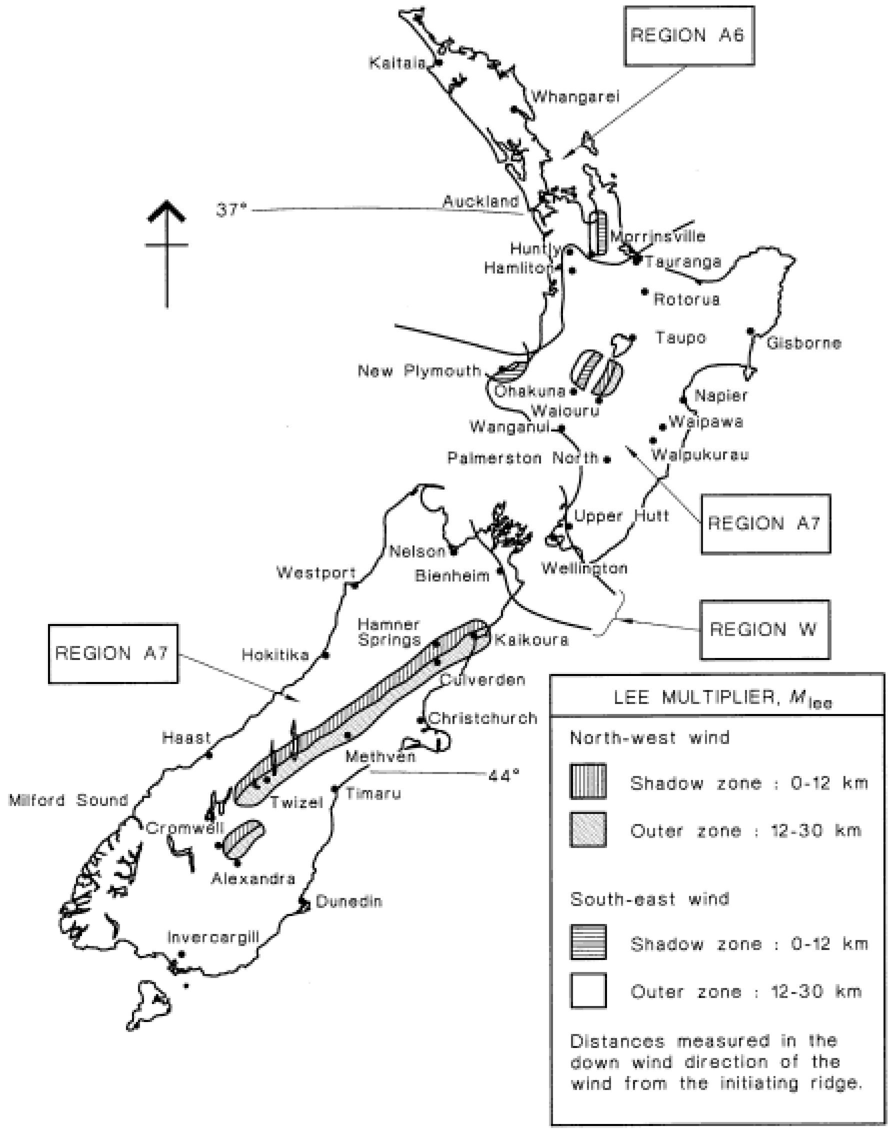
Convert m/s to	**Multiply by**
kph	3.600

mph	2.237
ft/s	3.281

Wind Region

Select the wind region. The wind region is determined from the geographic locations for Australia and New Zealand. The maps of these locations are found in Figure 3.1 of the code:





New Zealand

Terrain Category

Select the terrain category, as defined in section 4.2.1 of the code:

Category 1

Exposed open terrain, with few or no obstructions and water surfaces at serviceable wind speeds.

Category 2

Water surfaces, open terrain, grassland with few, well-scattered obstructions having heights generally from 1.5 m to 10 m.

Category 3

Terrain with numerous closely spaced obstructions 3 m to 5 m high, such as areas of suburban housing.

Category 4

Terrain with numerous large, high (10 m to 30 m high) and closely spaced obstructions, such as large city centres and well-developed industrial complexes.

NOTE Select the terrain category with due regard to the permanence of the obstructions that constitute the surface roughness. In particular, vegetation in tropical cyclonic regions shall not be relied upon to maintain surface roughness during wind events.

Lee Effect Multiplier (Mlee)

Enter the lee effect multiplier. The default value is **1.0**.

Paragraph 4.4.3 discusses the issue of the lee effect multiplier. In the case of New Zealand, reference is made to the New Zealand site map shown in **Wind Region**. For all other sites, it shall be taken as 1.0. No lee zones have been defined in Australia.

The full paragraph from the code reads as follows:

> 4.4.3: The lee (effect) multiplier (Mlee) shall be evaluated from New Zealand sites in the lee zones as shown in Figure 3.1(b). For all other sites, the lee multiplier shall be 1.0. Within the lee zones, the lee multiplier shall apply only to wind from the cardinal directions nominated in Figure 3.1(b). Each lee zone shall be 30 km in width, measured from the leeward crest of the initiating range, downwind in the direction of the wind nominated. The lee zone comprises a 'shadow lee zone', which extends 12 km from the upwind boundary of the lee zone (crest of the initiating range), and an 'outer lee zone' over the remaining 18 km. The lee multiplier shall be 1.35 for sites within the shadow lee zone (i.e., within 12 km of the crest of the range). Within the outer lee zone, the lee multiplier shall be determined by linear interpolation with the horizontal distance, from the shadow/outer zone boundary (where Mlee - 1.35), to the downwind lee zone boundary (where Mlee = 1.0).

Hill Shape Factor (Mh)

Enter the hill shape factor M_h, taken from Table 4.4:

Upwind Slope (H/2Lu)	**M_h**
< 0.05	1.0
0.05	1.08
0.10	1.16
0.20	1.32
0.30	1.48

>= 0.45	1.71

Paragraph 4.4.2 gives precise details for the derivation of the hill shape factor.

Wind Direction Multiplier

Enter the wind direction multiplier. The default value is 1.0.

The wind direction multiplier is detailed in paragraph 3.4 of the code, specifically Table 3.2. As the wind multiplier is determined from the cardinal wind directions (N, NE, E, SE, S SW, W and NW), the value for any direction is specified in the table as **1.0**. This value is recommended for all cases.

Convert to Permissible Stress Gust Wind Speed

Select to convert the given wind speed to a permissible stress basis. This lowers the wind loads on the vessel. AS/NZS 1170.2 Supp 1:2002, Section C3 discusses the division of the wind speed given in the standard by the square root of 1.5.

Surface Roughness Height hr

Enter the surface roughness value h_r in mm. This value is used to calculate the ratio h_r/d which is then used to calculate the drag force coefficient (C_d) for rounded cylindrical shapes, according to Table E3. For pressure vessels, typical values range from **0.003** mm for painted metal surfaces to **15** mm for heavily rusted surfaces. Light rust has a value of **2.5** mm, while galvanized steel has a value of **0.15** mm.

Site Elevation (E)

Enter the height of the site above mean sea level *E*.

Base Elevation from Site (Eb)

Enter the height of the base of the vessel above the site level E_b. This is relevant in cases where the vessel is supported on a structure or an elevated foundation base.

Avg Spacing of Shielding Bldgs

Enter the average spacing of the buildings that shield the vessel. This is discussed in paragraph 4.3.3 of the code:

The shielding parameter (s) in Table 4.3 shall be determined as follows:

$$s := \frac{l_s}{\sqrt{h_s \cdot b_s}}$$

where

l_s = average spacing of shielding buildings, given by:

$$= h \cdot \left(\frac{10}{n_s} + 5\right)$$

h_s = average roof height of shielding buildings

b_s = average breadth of shielding buildings, normal to the wind stream

h = average roof height, above ground, of the structure being shielded

n_s = number of upwind shielding buildings within a 45 deg. sector of radius 20h and with $h_s \geq z$

Avg Breadth of Shielding Bldgs

Enter the average breadth of the buildings that shield the vessel. For more information, see **Avg Spacing of Shielding Bldgs.**

Avg Height of Shielded Bldgs (hs)

Enter the average height of the buildings that shield the vessel. For more information, see **Avg Spacing of Shielding Bldgs.**

of Upwind Bldgs @ 45 degrees

Enter the number of upwind buildings within a 45° arc. The upwind buildings are the ones shielding the vessel. For more information, see **Avg Spacing of Shielding Bldgs.**

ASCE-93 Wind Data

Design Wind Speed

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant

Select an ASCE-7 exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor

Enter a value for the importance factor. ASCE-7 generally requires a value between 0.95 and 1.11. The software uses this value directly without modification. Values are taken from table 5 of the ASCE standard:

	Importance Factor		
Category	**100 mi from Hurricane Oceanline**	**At Oceanline**	**Classification**
I	**1.00**	**1.05**	Buildings and structures not listed below. **NOTE** Most petrochemical structures use this category.
II	**1.07**	**1.11**	Buildings and structures where more than 300 people congregate in one area.
III	**1.07**	**1.11**	Buildings designed as essential facilities, hospitals, and so on.
IV	**0.95**	**1.00**	Buildings and structures that represent a low hazard in the event of a failure.

Roughness Factor

Enter the roughness factor (from ASCE-7, Table 12):

- **1 -** Round, moderately smooth
- **2** - Round, rough (D'/D = 0.02)
- **3** - Round, very rough (D'/D = 0.08)

NOTES

- Most petrochemical sites use a value of 1 (moderately smooth). Some designers use a value of 3 (very rough) to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.

- If your specification calls out for a defined value of the wind shape factor, click **Tools** tab > **Set Configuration Parameters** to enter it in directly on the *Job Specific Setup Parameters Tab (Configuration Dialog)* (page 224) tab.

ASCE-95 and ASCE-98/02/05/IBC-03 Wind Data

Basic Wind Speed [V]

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness

Select an ASCE-7 exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor [I]

Enter a value for the importance factor. ASCE-7 generally requires a value between 0.77 and 1.15. The software uses this value directly without modification. Values are taken from table 6-2 of the ASCE 95 standard or table 6-1 from the ASCE 98 standard:

Category	Factor	Classification
I	**0.87**	Buildings and other structures that represent a low hazard to human life in the event of failure. **NOTE** Most petrochemical structures use this category.
II	**1.00**	Buildings and structures except those listed in categories I, III and IV.
III	**1.15**	Buildings and structures that represent a substantial hazard in the event of a failure.

IV	**1.15**	Buildings designed as essential facilities, such as hospitals.

NOTE In the 98 standard, the importance factor can be **0.77** for category I when wind speeds are greater than 100 miles per hour.

Type of Surface

Enter the roughness factor (from ASCE-7, Table 12):

- **Smooth**
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

NOTES

- Most petrochemical sites use smooth. Some designers use very rough to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab, click **Set Configuration Parameters** to enter it in directly on the **Job Specific Setup Parameters** tab.

Hill Height [H]

Enter the height of a hill or escarpment relative to the upwind terrain. See ASCE 7-95/98, Fig. 6-2 for details.

Distance to Site [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. See ASCE 7-95/98 Fig. 6-2 for details.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. See ASCE 7-95/98, Fig. 6-2 for more details.

Type of Hill

Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. See ASCE 7-95/98, Fig. 6-2 for details.

ASCE-2010 Wind Data

Enter data for the ASCE 7-2010 wind code.

Basic Wind Speed [V]

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness

Select an exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor [I]

This value is no longer used.

Type of Surface

Enter the roughness factor (from ASCE 7-10, Figure 29.5-1):

- **Smooth** (Moderately smooth)
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Smooth**.
- You can use very rough to account for items such as platforms, piping, and ladders, instead of entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters** to enter it directly on the **Job Specific Setup Parameters** tab.

Hill Height [H]

Enter the height of a hill or escarpment relative to the upwind terrain. For more information, see ASCE 7-10, Figure 26.8-1.

Distance to Site [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. For more information, see ASCE 7-10, Figure 26.8-1.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. For more information, see ASCE 7-10, Figure 26.8-1.

Type of Hill

Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. For more information, see ASCE 7-10, Figure 26.8-1.

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Brazil NBR 6123 Wind Data

NBR 6123 provides two types of wind analysis: static and dynamic.

Static wind analysis is performed when the fundamental period of vibration is less than one second. The goal of the analysis is to determine the wind pressure at each elevation of interest (z). The basic equation is:

$$q(z) = V_k^2/1.63$$

Where:

- $V_k = V_o * S_1 * S_2 * S_3$
- V_o, S_1 and S_3 are constants
- S_2 changes as a function of the height
- Wind velocities are specified in m/s and pressures are specified in N/mm^2

After the pressure is calculated, the force on the element is the pressure times the element area times the shape factor:

<table>
<tr><th rowspan="2">Surface Condition</th><th colspan="7">Shape Factor
for Height / Diameter of:</th></tr>
<tr><th>0.5</th><th>1.0</th><th>2.0</th><th>5.0</th><th>10</th><th>20</th><th>∞</th></tr>
<tr><td>Rough</td><td>0.7</td><td>0.7</td><td>0.7</td><td>0.8</td><td>0.9</td><td>1.0</td><td>1.2</td></tr>
<tr><td>Smooth</td><td colspan="5">0.5</td><td>0.6</td><td>0.6</td></tr>
</table>

Dynamic wind analysis is performed when the fundamental period of vibration is greater than 1 second. You must first calculate the project wind velocity V_p:

$$V_p = 0.69 * V_o * S_1 * S_3$$

Next, calculate d/H and $V_p * T_1/1800$. Using these two terms and knowing the height of the vessel, the value of Xi can be determined from figures 14-18 in NBR 6123.

b and p, functions of **Roughness Category**, can now be determined:

Value	**Ground Category**				
	I	**II**	**III**	**IV**	**V**
p	0.095	0.150	0.185	0.230	0.310
b	1.23	1.00	0.86	0.71	0.50

Finally, calculate the wind pressure:

$$q(z) = 0.613 * b_2 * V_p^2\{(z/10)^{2p} + [(H/10)^p * (z/H)^{1.7} * (4.4/(2.7 + p) * Xi]\}$$

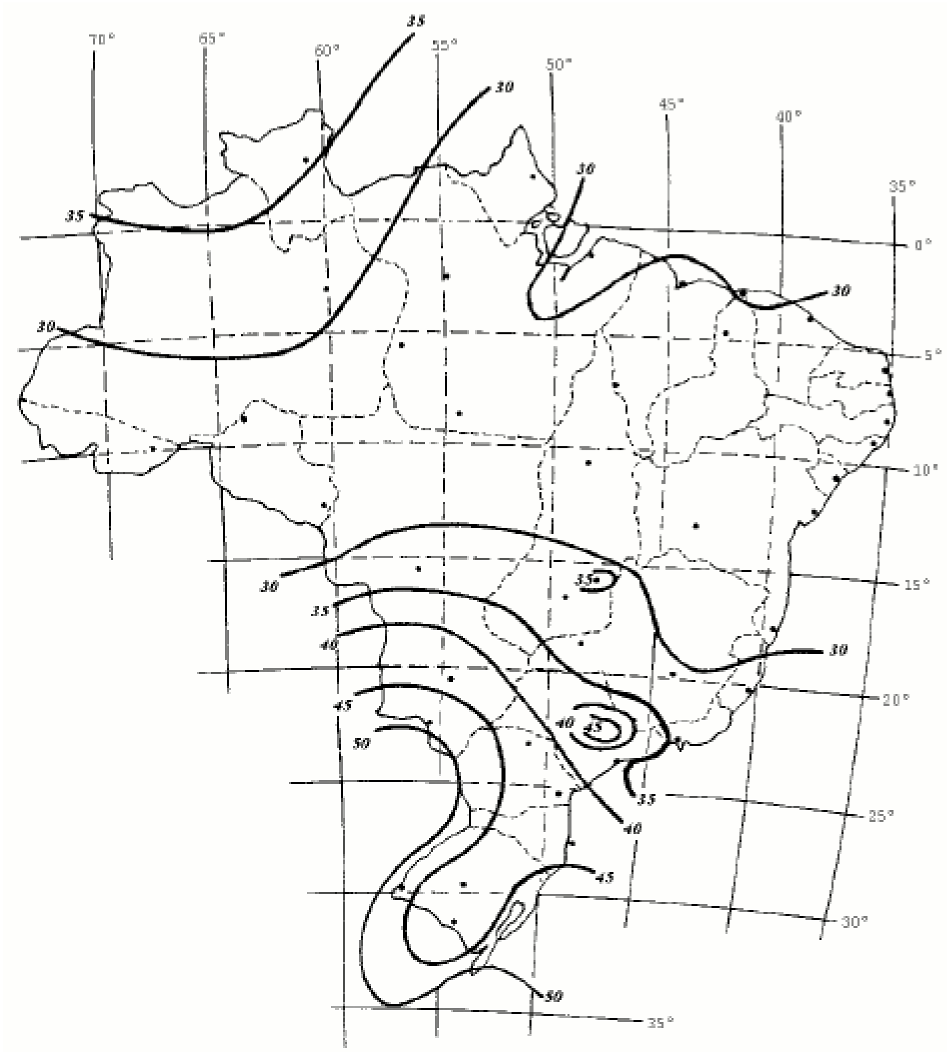
After the pressure at the needed elevation is known, the force is determined as stated above.

NOTE If you have a shape factor value that you want to enter yourself, go to the **Tools** tab, click **Set Configuration Parameters**, and then enter the value on **Job Specific Setup Parameters** tab in the **Wind Shape Factor** box.

Basic Wind Velocity (Vo)

Enter the wind velocity V_o from a three second gust, exceeded only once in 50 years. The velocity is measured at 10 meters over smooth open ground and depends on the plant location. As a general rule, the wind can blow in any horizontal direction. This velocity is taken from Figure 1 and item 8 showing the iso-velocities over Brazil.

The following are found in Petrobras document BPE-500-P4-19i and the Brazilian Wind Code NBR 6123:



Refinery Name	Wind Velocity (in m/s)
LUBNOR	30.0
RECAP	40.0
REDUC	35.0
REFAP	45.0
REGAP	30.0
REPAR	40.0

REPLAN	45.0
REMAN	30.0
REVAP	40.0
RPBC	50.0
RLAM	30.0

Topographic Factor (S1)

Enter the topographic factor S_1. This factor accounts for the variations and profile of the land. For plain, or slightly uneven ground, use a value of **1**. The larger this value is, the greater the final calculated wind pressure will be. If the vessel is on a hill top, this value should be calculated according to section 5.2 of NBR 6123.

Roughness Category (S2)

Select the roughness category S_2:

Category	**Description**
I	Applies to plain ground with large dimensions (more than 5 km of extension)
II	Applies to plain (or slightly uneven) ground with few, and separated, obstacles
III	Applies to plain or uneven ground obstructed by obstacles (walls or separated low buildings)
IV	Applies to ground with many grouped obstacles in industrial or urban areas
V	Applies to ground with many grouped and tall obstacles (such as developed industrial areas)

NOTE The *lower* the category, the *higher* the wind load. For example, Category I produces a higher wind load than Category II.

Dimension Class

Select the Dimension class. This parameter accounts for the greatest horizontal or vertical dimension of the vessel.

Class	**Description**
A	The greatest dimension is less than or equal to 20 meters
B	The greatest dimension is greater than 20m and less than 50 meters
C	The greatest dimension is greater than or equal to 50 meters

Statistical Factor (S3)

Enter the statistical factor S_3.This factor accounts for security and the expected life of the equipment. For industrial plants, use **1.0**.

Base Elevation

Enter the distance the base of the equipment is from grade.

Vessel Surface Condition

Select the vessel surface condition, **Smooth** or **Rough**. **Rough** results in an increased value of the shape factor and generates a higher wind load on the vessel because there is more drag. The shape factor is based on the height to diameter ratio of the vessel:

Surface Condition	**Shape Factor for Height / Diameter of:**						
	0.5	1.0	2.0	5.0	10	20	∞
Rough	0.7	0.7	0.7	0.8	0.9	1.0	1.2
Smooth	0.5					0.6	0.6

BS6399-97 Wind Data

BS 6399-97 - The British Wind Code - Loadings for buildings - Part 2: Code of practice for wind loads. The year of issuance of this code is 1997, replacing CP3.

Design Wind Speed

Enter the design value of the wind speed V_b. This varies according to geographical location and according to company or vendor standards. Typical wind speeds are shown in Figure 6 of BS 6399 and vary from 20 to 31 meters per second (44.7 to 69.3 miles per hour). The wind speeds are only relevant to the United Kingdom.

NOTE Enter the lowest value reasonably allowed by the standards you are following, because the wind design pressure, and thus force, increases as the square of the speed.

Site Elevation (Delta s)

Enter the site altitude above mean sea level (paragraph 2.2.2.2 of the code). This value plus **Base Elevation** is used to calculate the height of each point in the vessel above mean sea level. For example, if the vessel is installed on a site that is 100 m (328 ft) above sea level, it is exposed to a higher wind pressure *P* than if installed on the beach (at mean sea level).

Upwind Building Height Ho

Enter the average height H_o of buildings upwind of the vessel, when **Town** is selected for **Vessel Location**. The buildings tend to shield the vessel from the wind. Conservatively, this value can be zero, so that the vessel takes the full force of the wind. H_o is used to modify the effective vessel wind height H_e for any vessel element. See paragraph 1.7.3.3 of BS6399.

NOTE The corrected final wind pressure V_e acting on any element of the vessel is determined by **Vessel Location**, **Distance to Coastline**, and the effective height H_e (modified by **Upwind Building Height Ho)**. Factors in table 4 of BS6399 modify the wind velocity. This table derives S_b, which is calculated internally by the software.

Upwind Building Spacing X

Enter the average spacing *X* of buildings upwind of the vessel, when **Town** is selected for **Vessel Location**. The buildings tend to shield the vessel from the wind. If the buildings are closer together, they provide greater protection from the wind. See paragraph 1.7.3.3 of BS6399.

Base Elevation

Enter the elevation at the base of the vessel. This value plus the value of **Site Elevation (delta s)** is used to calculate the height of each point in the vessel above mean sea level.

Vessel Location

Select the type of location where the vessel is installed, **Town**, or **Country**.

NOTE The corrected final wind pressure V_e acting on any element of the vessel is determined by **Vessel Location**, **Distance to Coastline**, and the effective height H_e (modified by **Upwind Building Height Ho)**. Factors in table 4 of BS6399 modify the wind velocity. This table derives S_b, which is calculated internally by the software.

Distance to Coast Line

Enter the distance the vessel is located from the coast.

NOTE The corrected final wind pressure V_e acting on any element of the vessel is determined by **Vessel Location**, **Distance to Coastline**, and the effective height H_e (modified by **Upwind Building Height Ho)**. Factors in table 4 of BS6399 modify the wind velocity. This table derives S_b, which is calculated internally by the software.

Size Effect Factor Ca

Enter the size effect factor C_a. This factor generally ranges from 0.53 to a maximum value of 1.0. The size effect factor is a function of the diagonal dimension *a*, the effective height H_e,**Vessel Location**, and **Distance to Coastline**. This value is taken from figure 4 of BS-6399-2.

Factor Kb from Table 1

Select the building-type factor K_b, taken from Table 1 of BS6399. The default value is 2. Select 8, 4, 2, 1, or 0.5 based on the following vessel height limitations:

K_b	Building Type	Maximum Total Height of Vessel
8	Welded steel unclad frames.	23 m (75.4 ft)
4	Bolted steel and reinforced concrete unclad frames.	75m (246 ft)
2	Portal sheds and similar light structures with few internal walls.	240m (787 ft)
1	Framed buildings with structural walls around lifts and stairs only (such as, office buildings of open plan or with partitioning).	300m (984 ft)
0.5	Framed buildings with structural walls around lifts and stairs with	300m (984 ft)

	additional masonry subdivision walls (such as apartment buildings), building of masonry construction and timber-framed housing.	

NOTE Towers over 75 meters in height are not likely, and require other design considerations in addition to wind loading.

Annual Probability Factor Q

Enter the probability factor *Q*, used to calculate the final probability factor S_p associated with the likelihood of high velocity gusts occurring over specified periods. The default value is 0.02, set by the code as a standard value for a mean recurrence of 50 years. See Annex D of BS6399 for more information.

Q	Explanation
0.632	NOTE 1: The annual mode, corresponding to the most likely annual maximum value. (S_p = 0.749)
0.227	NOTE 2: For the serviceability limit, assuming the partial factor for loads for the ultimate limit is ?f = 1.4 and for the serviceability limit is ?f = 1.0, giving S_p = Sqrt(1 / 1.4) = 0.845. (S_p = 0.845)
0.02	NOTE 3: The standard design value, corresponding to a mean recurrence interval of 50 years. (S_p = 1.000)
0.0083	NOTE 4: The design risk for bridges, corresponding to a mean recurrence interval of 50 years. (S_p = 1.048)
0.00574	NOTE 5: The annual risk corresponding to the standard partial factor for loads, corresponding to a mean recurrence interval 1,754 years. Back-calculated assuming the partial factor load for the ultimate limit is ?f = 1.4 and all risk is ascribed to the recurrence of wind. (S_p = Sqrt(1.4))
0.001	NOTE 6: The design risk for nuclear installations, corresponding to a mean recurrence interval of 10,000 (ten thousand) years. (S_p = 1.263)

Seasonal Factor Ss

Enter a factor S_s for exposure to the weather. The default value is 1.0. BS6399 in paragraph 2.2.2.4 states: "For permanent buildings and buildings exposed for continuous periods of more than six months a value of 1.0 should be used for S_s." A value of less than **1.0** should only be used after solid research.

Directional Factor Sd

Enter a factor S_d for directionality of the tower. The default value is **1.0** because a tower is typically symmetrical about its central axis. A value of less than **1.0** should only be used under exceptional circumstances. For other values, see Table 3. The values in that table range between 0.73 and 1.00.

Vessel Surface Type

Select the surface type of the vessel, **Rough** or **Smooth**.

China GB 50009 Wind Data

The Chinese Wind Code analysis in the software is taken from Chinese specification GB 50009 - 2001, 2002. The wind loading calculation guidelines begin on page 24 of the code.

The basic formulation for determining the wind pressure at an arbitrary elevation is based on equation 7.1.1-1. This equation is for Main Wind Force Resisting Systems. This is the printed equation: *wk* = *ßzμsμzwo*. From the tables in the code, values of *μs*, *μz* and the other values are determined.

This formula includes the shape coefficient. The generated wind pressure is not dependent on the type of structure. However, when the final force is calculated, it is necessary to include the shape factor, such as for a cylinder taken from page 39 for a tower or chimney.

Ref. Wind Pressure (wo)

Enter the reference wind pressure W_o , from table D.4 of the Chinese Wind Design Code. The value should be no less than 0.3 kN/m^2.

Terrain Roughness Category

Enter the terrain roughness category:

Value	Description
A	Flat, unobstructed open terrain (most conservative)
B	Village, hill and less populated and less congested sites
C	Populated sites with low buildings and shorter structures
D	Densely populated areas with many tall structures that provide shielding (least conservative)

The category is used to find elevation μ_z, from Table 7.2.1:

Height (m)	**Terrain Roughness Category**			
	A	**B**	**C**	**D**
5	1.17	1.00	0.74	0.62
10	1.38	1.00	0.74	0.62
15	1.52	1.14	0.74	0.62
20	1.63	1.25	0.84	0.62
30	1.80	1.42	1.00	0.62
40	1.92	1.56	1.13	0.73
50	2.03	1.67	1.25	0.84

60	2.12	1.77	1.35	0.93
70	2.20	1.86	1.45	1.02
80	2.27	1.95	1.54	1.11
90	2.34	2.02	1.62	1.19
100	2.40	2.09	1.70	1.27
150	2.64	2.38	2.03	1.61
200	2.83	2.61	2.30	1.92
250	2.99	2.80	2.54	2.19
300	3.12	2.97	2.75	2.45
350	3.12	3.12	2.94	2.68
400	3.12	3.12	3.12	2.91
>= 450	3.12	3.12	3.12	3.12

Vessel Surface Condition

Select the vessel surface condition, taken from page 39 of the Chinese Wind Code. Select **Smooth**, **Rough**, or **Very Rough**. You can override this option in **Tools** > **Configuration**.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Is the Vessel Building Supported?

Select to use the factors for β_{gz} from Table 7.5.1:

Height (m)	**Terrain Roughness**			
	A	**B**	**C**	**D**
5	1.69	1.88	2.30	3.21
10	1.63	1.78	2.10	2.76
15	1.60	1.72	1.99	2.54
20	1.58	1.69	1.92	2.39

30	1.54	1.64	1.83	2.21
40	1.52	1.60	1.77	2.09
50	1.51	1.58	1.73	2.01
60	1.49	1.56	1.69	1.94
70	1.48	1.54	1.66	1.89
80	1.47	1.53	1.64	1.85
90	1.47	1.52	1.62	1.81
100	1.46	1.51	1.60	1.78
150	1.43	1.47	1.54	1.67
200	1.42	1.44	1.50	1.60
250	1.40	1.42	1.46	1.55
300	1.39	1.41	1.44	1.51

The value of β_{gz} decreases with elevation so that the wind pressure decreases as the elevation increases. The software uses equation 7.1.1-2 in the code to determine the wind pressure at the needed elevation.

EN-2005 and EN-2010 Wind Data

Ref. Wind Velocity Vb,0

Enter the basic wind velocity $V_{b,0}$ of the area where the equipment is situated. $V_{b,0}$ is used along with **C Dir** and **C Season** to compute V_b.

Terrain Category

Select the terrain category:

Terrain Category	Description
0	Sea or coastal area exposed to the open sea.
I	Lakes or flat and horizontal areas with negligible vegetation and without obstacles.
II	Area with low vegetation such as grass and isolated obstacles (trees, buildings) with separations of at least 20 obstacle heights.
III	Area with regular cover of vegetation or buildings, or with isolated obstacles with

	separations of maximum 20 obstacle heights (such as villages, suburban terrain, permanent forest).
IV	Area in which at least 15% of the surface is covered with buildings and their average height exceeds 15 m.

NOTE The *lower* the category, the *higher* the wind load. Category 0 produces the highest wind loads, while Category 5 produces the lowest.

C Dir

Enter the directional factor, C_{dir}. Values are found in the National Annex. The recommended value is **1.0**.

C Season

Enter the season factor, C_{season}. Values are found in the National Annex. The recommended value is **1.0**.

CsCd

Enter the structural factor C_sC_d, used to determine the force on the vessel. This value is defined in the EN 1991-1-4:2005(E) wind load specification, Annex D. This value normally ranges between **0.90** and **1.10**. The greater the value of the structural factor, the higher the element load.

Force Coefficient Cf

Enter the force coefficient C_f. This value accounts for the fact that the vessel is circular in cross section and is used to modify the area of the vessel that the wind is blowing against. This value is quite often specified in the design specifications or can be calculated based on the methodology given in Section 7.9 for circular cylinders. A typical value for C_f is between **0.7** and **0.8**.

Base Height

Enter in the distance from the bottom of the vessel to base (ground) elevation, in cases where vessels are not fixed to the ground, but are attached to other structures.

Euro Code Wind Data

In common with other wind codes, the principles of the European code are basically the same. A moving air mass causes forces to be exerted on anything that obstructs its path. The starting point of any wind action analysis starts with the Bernoulli equation that converts the kinetic energy of the wind into a static pressure. This pressure simply attempts to move anything in its path. In the case of a vertical tower, the wind deflects the tower in the direction of the flowing wind. Because of the friction, the wind (which is a fluid) passes around the vessel, creating small eddie currents called vortices on either side of the vessel. The spinning air loses some of it pressure as its angular kinetic energy increases. This is again an effect predicted by the Bernoulli equation. This spinning eddie, or vortex, causes a lowering of the pressure on one side of the vessel. As these vortices occur alternatively on the sides of the vessel, the tower tends to sway from side to side perpendicular to the direction of the wind, vibrating like an upside-down pendulum.

The wind pressure causes a static deflection in the direction of the wind stream, and the vortices cause the vessel to vibrate at right angles to the wind stream (vector) as mentioned above.

Static deflection and dynamic vibration cause the vessel to experience longitudinal, or axial bending stresses. Both the stresses and deflection must be limited to provide for good design.

As stated above, the wind kinetic energy is converted to pressure, but there are other factors influencing the final wind pressure. Among these factors are (but not limited to) the following:

- The height, or altitude, of the tower or vessel in the wind stream.
- The elevation of the site on which the vessel is installed.
- The base elevation of the vessel if is supported on an elevated platform or structure.
- The effect of the surround geography, such as hills and escarpments.
- The effect of upstream buildings.

To perform a reasonable analysis, you must enter sufficiently accurate information for the software to apply the correct factors and special calculations for specific cases.

Ref. Wind Velocity Vref,0

Enter the reference wind velocity V_{ref}, taken from section 7.2 of the code. Annex A provides the values:

Country	**Suggested Wind Velocities (SI Metric Unit System)**
Austria	Special consideration must be given, depending on the district.
Belgium	26.2 m/s
Denmark	27.0 m/s
Finland	23.0 m/s
France	24.0 m/s to 34.0 m/s, depending on the district (Departements et Cantons).
Germany	24.3 m/s to 31.5 m/s, depending on the particular Zone.
Greece	30.0 m/s to 36.0 m/s, depending whether coastal or country.
Iceland	The wind speed must be calculated - See Annex A.8.
Ireland	27.0 m/s to 30 m/s, depending on the region and exposure to the Atlantic Ocean.
Italy	25.0 m/s to 31 m/s, depending on the region.
Luxembourg	26.0 m/s
Netherlands	22.5 m/s to 30.0 m/s, depending on return period and area.
Norway	25.0 m/s to 65 m/s. Please refer to Figure A7 in the code.

Terrain Category

Select the terrain category, taken from Section 8.3 of the code:

Terrain Category	**k_T**	**Z_0 [m]**	**Z_{min} [m]**	**ε**	
I - Flat, Unobstructed	Rough open sea, lakes with at least 5 km fetch upwind and smooth flat country without obstacles	0.17	0.01	2	[0.13]
II - Farmland Areas	Farmland with boundary hedges, occasional small farm structures, houses or trees	0.19	0.05	4	[0.26]
III - Suburban/ Industrial	Suburban or industrial areas and permanent forests	0.22	0.3	8	[0.37]
IV - Urban Areas	Urban areas in which at least 15% of the surface is covered with buildings and their average height exceeds 15 m	0.24	1	16	[0.43]

C Dir

Enter the direction factor C_{Dir}. The direction factor is a function of the country where the vessel is installed. From Annex A:

Country	**C_{Dir}**
Austria	1.0
Denmark	1.0
Finland	1.0
France	1.0
Germany	1.0
Greece	1.0
Iceland	Not Specified
Ireland	1.0
Italy	Not Specified

C Tem

Enter the temperature factor C_{tmp}, which affects the air density. From Annex A:

Country	**C_{tmp}**
Austria	Not Specified

Denmark	1.0
Finland	1.0
France	1.0
Germany	Please refer to Table shown in A.6. Values range from .30 to 0.65
Greece	1.0
Iceland	Not Specified
Ireland	1.0
Italy	Not Specified

C Alt

Enter the altitude factor C_{Alt}. From Annex A:

Country	C_{Alt}
Austria	Not Specified
Denmark	1.0
Finland	1.0
France	1.0
Germany	Calculated from the formula in section A.6. 1,0 seems to be a safe value.
Greece	1.0
Iceland	Not Specified
Ireland	1.0
Italy	Not Specified

G Peak

Enter the peak factor. In section 8.5 the value is given as 3.5.

Force Coefficient Cf

Enter the force coefficient $C_{f,0}$. For cylindrical surfaces the force coefficient is 0.70. This assumes the Reynolds number of $2.10^5 < Re < 1,2.10^6$. For more information, see Table 10.7.1 in the code. No other values are suggested, so please refer to the code for any relevant information. Section 10.8.1 goes further into the subject where the Reynolds number can be computed. Figure 10.8.2 can also be consulted, but you will need to calculate the Reynolds number.

Base Height

Enter the base height. This is the height of the base of the vessel above the site (or ground).

Cd (used if > 0.94)

Enter the direction factor C_{dir}. A value of **0** is recommended in the absence of any other information. The software calculates this value, but if a value greater than 0.94 is entered, that value will be used. For more information, see Annex A of the code.

IBC 2006, 2009, 2012, 2015, and 2018 Wind Data

Basic Wind Speed [V]

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness

Select an ASCE-7 exposure factor:

- **Exposure A** - Large city centers.
- **Exposure B** - Urban and suburban areas.
- **Exposure C** - Open terrain.

  NOTE Most petrochemical sites use this value.
- **Exposure D** - Flat unobstructed coastal areas.
- **Not Used**

Importance Factor [I]

Enter a value for the importance factor. ASCE-7 generally requires a value between 0.77 and 1.15. The software uses this value directly without modification. Values are taken from table 6-2 of the ASCE 95 standard or table 6-1 from the ASCE 98 standard:

Category	Factor	Classification
I	**0.87**	Buildings and other structures that represent a low hazard to human life in the event of failure. **NOTE** Most petrochemical structures use this

		category.
II	**1.00**	Buildings and structures except those listed in categories I, III and IV.
III	**1.15**	Buildings and structures that represent a substantial hazard in the event of a failure.
IV	**1.15**	Buildings designed as essential facilities, such as hospitals.

NOTE In the 98 standard, the importance factor can be **0.77** for category I when wind speeds are greater than 100 miles per hour.

Type of Surface

Enter the roughness factor (from ASCE-7, Table 12):

- **Smooth**
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

NOTES

- Most petrochemical sites use smooth. Some designers use very rough to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab, click **Set Configuration Parameters** to enter it in directly on the **Job Specific Setup Parameters** tab.

Hill Height [H]

Enter the height of a hill or escarpment relative to the upwind terrain. See ASCE 7-95/98, Fig. 6-2 for details.

Distance to Site [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. See ASCE 7-95/98 Fig. 6-2 for details.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. See ASCE 7-95/98, Fig. 6-2 for more details.

Type of Hill

Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. See ASCE 7-95/98, Fig. 6-2 for details.

IS-875 Wind Data

Use the IS-875 2015 Standard?

Select to use IS 875: 2015 as the code for analysis. This option is only available when **IS-875** is selected for **Wind Design Code.**

Importance Factor

Enter a value for the importance factor *K4*. The importance factor generally requires a value between 1.00 and 1.30. Values are taken from section 4.4.3.4 of the IS 875 standard.

Factor	Classification
1.30	Structures of post-cyclone importance for emergency services (such as cyclone shelters, hospitals, schools, communication towers, and so forth)
1.15	Industrial structures
1.00	All other structures

NOTE The importance factor is only used when you select **Use the IS-875 2015 Standard?**.

Basic Wind Speed

Enter a value for basic wind speed. Basic wind speed is based on peak gust velocity averaged over a short time interval of about three seconds and corresponds to mean heights above ground level over open terrain. The value varies according to geographical location and according to company or vendor standards. Typical wind speeds range from 85 to 120 miles per hour. Enter the lowest value reasonably allowed by the standards that you are following. This is because the wind design pressure, and thus force, increases as the square of the speed. This entry is optional, and overrides the basic wind speed determined from the zone entered for **Wind Zone Number**.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Wind Zone Number

Enter a value for the wind zone. India is divided into six wind zones. The software provides the basic wind speed based on the zone. Basic wind speed is applicable to a 10 meter height above mean ground level for different zones, as determined from Figure 1 in IS 875.

Zone	Basic Wind Speed
1	33 m/sec 73.82 miles/hour

2	39 m/sec 87.25 miles/hour
3	44 m/sec 98.43 miles/hour
4	47 m/sec 105.15 miles/hour
5	50 m/sec 111.86 miles/hour
6	55 m/sec 123.04 miles/hour

NOTE Optionally, you can enter a value for **Basic Wind Speed**. This value overrides **Wind Zone Number**.

Risk Factor

Enter a value for the risk coefficient *K1*. The value varies based on the zone selected for **Wind Zone Number**, and assumes 100 years as the mean probable design life of a tower.

Wind Zone	**Risk Factor (K1)**
1	**1.05**
2	**1.06**
3	**1.07**
4	**1.07**
5	**1.08**
6	**1.08**

Terrain Category

Select the category for the terrain in which the vessel is located:

- **Category 1 -** Exposed open terrain with few or no obstructions. The average height of any object surrounding the equipment is less than 1.5 m. This category includes open seacoasts and flat, treeless plains.
- **Category 2 -** Open terrain with well-scattered obstructions having a height generally between 1.5 to 10 m. This includes airfields, open parklands, and undeveloped, sparsely built-up outskirts of towns and suburbs. This category is commonly used for design purposes.
- **Category 3 -** Terrain with numerous closely-spaced obstructions with buildings and structures up to 10 m in height. This includes well-wooded areas, towns, and industrial areas that are fully or partially developed.

- **Category 4** - Terrain with numerous tall, closely-spaced obstructions. This includes large city centers, with obstructions above 25 m, and well-developed industrial complexes.

Equipment Class

Select the equipment/component classification:

- **Class A** - Equipment and/or components have a maximum dimension (horizontal or vertical dimension) of less than 20 m.
- **Class B** - Equipment and/or components have a maximum dimension (horizontal or vertical dimension) between 20 and 50 m.
- **Class C** - Equipment and/or components have a maximum dimension (horizontal or vertical dimension) greater than 50 m.

Topography Factor

Enter a topography factor *K3*. This factor takes care of local topographic features such as hills, valleys, cliffs, or ridges, which can significantly affect wind speed in their vicinity. The effect of topography is to accelerate wind near summits of hills or crests of cliffs, and decelerate the wind in valleys or near the foot of a cliff. Topography effect is significant if the upwind slope is greater than about three degrees. If the upwind slope is below three degrees, the value of *K3* is 1.0. For slopes above three degrees, the value of *K3* ranges between 1.0 and 1.36.

Use the Gust Response Factor?

Select to calculate the gust response factor according to IS-875, and use the factor in the appropriate equations. Select only if the design specifications and the customer explicitly requires the gust response factor. This factor increases the wind load three to six times and may lead to a very conservative wind design.

Compute Cf from Table 23 or Table 28 (2015)?

Select to calculate the drag coefficient C_f. The C_f values come from Table 23 of IS 875: 1987 or Table 28 of IS 875: 2015.

JPI-7R-35-2004 Wind Data

Design Wind Speed

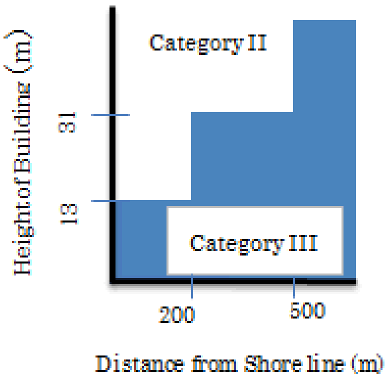
Enter the design value of the wind speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than if it were mounted at grade.

Exposure Constant

Select an exposure factor:

<table>
<tr><th colspan="2">Exposure Category</th><th>Z_b (m)</th><th>Z_G (m)</th><th>α</th></tr>
<tr><td>I</td><td>Outside of the town planning area and flat, unobstructed terrain</td><td>5</td><td>250</td><td>0.10</td></tr>
<tr><td>II</td><td>Outside of the town planning area other than Exposure Category I, except for building height is 13 m or less.
Inside of the town planning area other than Exposure Category IV,

</td><td>5</td><td>350</td><td>0.15</td></tr>
<tr><td>III</td><td>Other than Category I, II, and IV</td><td>5</td><td>450</td><td>0.20</td></tr>
<tr><td>IV</td><td>Inside of the town planning area and the center of city</td><td>10</td><td>550</td><td>0.27</td></tr>
</table>

KBC 2016 Wind Data

Enter data for the KBC 2016 wind code.

Basic Wind Speed [V]

Enter a value for basic wind speed. Basic wind speed is based on peak gust velocity averaged over a short time interval of about three seconds and corresponds to mean heights above ground level over open terrain. The value varies according to geographical location and according to company or vendor standards. Typical wind speeds range from 85 to 120 miles per hour. Enter the lowest value reasonably allowed by the standards that you are following. This is because the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Terrain Surface Roughness

Select an exposure factor:

- **Exposure A** - Large city centers
- **Exposure B** - Small buildings
- **Exposure C** - Open terrain
- **Exposure D** - Flat, unobstructed

Importance Factor [I]

Enter a value for the importance factor. The software uses this value directly without modification.

Type of Surface

Select the surface roughness factor:

- **Smooth** (moderately smooth)
- **Rough** (D'/D = 0.02)
- **Very Rough** (D'/D = 0.08)

Hill Height [Hh]

Enter the height of a hill or escarpment relative to the upwind terrain.

Horizontal Crest Distance [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. For more information, see KBC 2016 Chapter 3 Design Loads.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment, L_h.

Type of Hill

Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**.

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Mexico 1993 Wind Data

Mexico 1993 En Español (page 416)

Design Wind Speed

Enter the velocity of the wind, V_R, the maximum mean velocity likely to occur within a defined recurrence period in a determined zone or region of the country, as defined by Paragraph 4.6.2 (Isotach Maps regional Velocity).

The isotach maps that are included in this clause, with the different periods of return, such velocities, refer to homogeneous conditions that correspond to a height of 10 meters over the surface of the floor in the flat terrain (Category 2 per table I.1). The maps do not consider the local terrain roughness characteristics or the specific topography of the site. Therefore, such velocity is associated with three-second wind gusts and takes into account the possibility that there might be hurricane winds present in the coastal zones.

The regional velocity, V_R, is determined by taking into account the geographic location of the site of the uproot of the building and its destination.

In figures I.1 through I.4, the isotach regional maps are shown, corresponding to the periods of recurrence for 200, 50 and 10 years.

The importance of the structures (Para. 4.3) dictates the periods of recurrence which should be considered for the wind design. From this, the groups A, B and C in **Structure Class** associate themselves with the periods of return of 200, 50 and 10 years, respectively. The uproot site is located in the map with the recurrence period which corresponds to the group to which the building belongs to, in order to obtain the regional velocity. In the Tomo III from Ayudas de diseño, a table is shown with the main cities in the country and their corresponding regional velocities for the different periods of return.

Site Elev Above Sea Level

Enter the site elevation above mean sea level h_m.

Base Elevation (hb)

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Structure Class

Select a value for the structure class, from Paragraph 4.6.1, Table I.2:

- **A** - Horizontal or vertical structures measuring less than 20 meters of length. Every remote structural element is exposed directly to the wind action.
- **B** - Horizontal or vertical structures measuring between 20 and 50 meters of length
- **C** - Horizontal or vertical structures measuring more than 50 meters of length

Factors Affected by Structure Class

Paragraph 4.6.3.1 size factor F_C, α, and δ. The size factor, F_C, takes into consideration the time in which the gust of wind works in an effective manner over a building of given dimensions. Considering the classification of the structures size (table I.2), this factor can be determined according to table I.3.

Structure Class	F_C
A	1.0
B	0.95
C	0.90

The coefficients *α* and *δ* perform with the **Terrain Category**. Table I.4 below assigns the values for these coefficients. In Figure III.1 from *Tomo de Ayudas de diseño*, it is shown the factor's variation Fα with the height, with the terrain category and the structure class.

Terrain Category	α			δ
	Structure Class			
	A	B	C	
1	0.099	0.101	0.105	245
2	0.128	0.131	0.138	315
3	0.156	0.160	0.171	390
4	0.170	0.177	0.193	455

δ - The average height from the level of the land of uprooted above, by which the variation of the velocity of the wind is not important and can be supposed constant. This height is known as the gradient height; δ and Z are given in meters.

α - The exponent that determines the variation form of the wind velocity with the height. This value is unitless.

Terrain Category

Select the category for the terrain in which the vessel is located, from Paragraph 4.6.1, Table I.1:

Cat	Description	Examples	Limitations
1	Open terrain, practically smooth and without obstructions.	Coastal flat stripes, swamp zones, aerial fields, pasture fields and crop lands with no hedges or fences. Flat snow-covered surfaces.	The minimum length for this type of terrain in the direction of the wind must be of 2000 m or 10 times the height of the structure to be designed.
2	Flat terrain or undulated, with few obstructions.	Crop lands or farms with few obstructions around such as hedges of fences, trees and scattered buildings.	The obstructions have a height of 1.5 to 10 m, in a minimum length of 1500 m.

3	Terrain covered by many obstructions narrowly spaced out.	Urban, suburban areas and forests, or any other terrain with many obstructions widely scattered. The sizes of the buildings are like the houses and living spaces.	The obstructions have a 3 to 5 m height. The minimum length for this type of terrain in the direction of the wind must be 500 m or 10 times the height of the structure.
4	Terrain with many big, tall, and narrowly spaced-out obstructions.	Cities with downtown areas and well developed industrial complex areas.	At least 50% of the buildings have a height of more than 20 m. The obstructions measure up from 10 to 30 m in height. The minimum length for this type of land in the direction of the wind should be the biggest between 400 m and 10 times the height of the construction.

Factors Affected by Terrain Category

The factors κ', η, and δ, are related to the terrain category, as defined in Paragraph 4.9.3.2, Table I.29. κ' η are unitless, while δ is the height gradient in m.

	Terrain Category			
Factor	**1**	**2**	**3**	**4**
κ'	1.224	1.288	1.369	1.457
η	-0.032	-0.054	-0.096	-0.151
δ	245	315	390	455

The factors α' and k_r are also related to the terrain category, k_r is a factor related to the roughness of the land.

	Terrain Category			
Factor	**1**	**2**	**3**	**4**
k_r	0.06	0.08	0.10	0.14
α'	0.13	0.18	0.245	0.31

Year of Analysis

Select the analysis year to use.

Topographic Factor (Ft)

Enter a value for the topography local factor F_t, from Paragraph 4.5.4, Table I.5. This factor takes into account the local topographic effect from the place in which the structure will uproot. For example, for buildings found on hillsides, on top of hills, or on mountains at important heights with respect to the general level of the surrounding terrain, it is probable that wind accelerations will generate and the local velocity should be increased. Experts in local wind velocities should justify and validate the results of any of these procedures.

Places	**Topography**	**F_t**
Protected	Promontories base and skirts of mountain ranges from the leeward side.	**0.8**
	Closed valleys.	**0.9**
Normal	Flat terrain, Open field, absence of important topographical changes, with smaller slopes than 5%.	**1.0**
Exposed	Lands inclined with slopes between 5 and 10%, flat coastal and open valleys.	**1.1**
	Top of promontories, hills or mountains, terrains with greater slopes than 10%, glens closed and valleys that form a funnel or islands.	**1.2**

Damping Factor (Zeta)

Enter a value for the damping factor ζ. Enter **0.01** for steel framework construction. Enter **0.02** for concrete framework construction.

Drag Coefficent (Ca)

Enter a value for the drag coefficient C_a, from Paragraph 4.8.2.12 (Chimneys and Towers), Table I.28:

Cross Section	**Type of Surface**	**Drag Coefficient C_a for H/b of:**			
		1	7	25	≥ 40
Circular ($bV_D \geq 6$ m²/s)	Smooth or little rough (d'/b ≈ 0.0)	**0.5**	**0.6**	**0.7**	**0.7**
	Rough (d'/b ≈ 0.02)	**0.7**	**0.8**	**0.9**	**1.2**
	Very rough (d'/b ≈ 0.08)	**0.8**	**1.0**	**1.2**	**1.2**
Circular ($bV_D \geq 6$ m²/s)	Any	**0.7**	**0.8**	**1.2**	**1.2**
Hexagonal or octagonal	Any	**1.0**	**1.2**	**1.4**	**1.4**
Square (wind normal to a face)	Any	**1.3**	**1.4**	**2.0**	**2.2**
Square (wind on a corner)	Any	**1.0**	**1.1**	**1.5**	**1.6**

NOTES

- **b** - The diameter or the horizontal dimension of the structure, including the roughness of the wall. To determine the product bV_D, this diameter is located at two-thirds of the total height from the level of the land, in m.
- **d'** - The dimension that extends from the roughness, such as ribs or "spoilers", in m.
- **V_D** - The velocity of the wind of design (4.6), in m/s at two thirds of the total height
- For intermediate values of *H/b* and *d'/b*, lineal interpolation is permitted.

Strouhal Number (St)

Enter a value for the Strouhal number S_t. Enter **0.2** for circular sections or **0.14** for rectangular sections.

Barometric Height (Omega)

Enter the barometric pressure Ω. This value is related to **Site Elev Above Sea Level**, as described in Paragraph 4.6.5, Table I.7:

Height (meters above sea level)	Barometric Pressure (mm Hg)
0	760
500	720
1000	675
1500	635
2000	600
2500	565
3000	530
3500	495

NOTE Interpolate for intermediate values.

Ambient Temperature (T)

Enter a value for the ambient temperature *T*.

Mexico 1993 En Español

See Mexico 1993 Wind Data for the English version of this text.

Párrafo 4.6.1

Tabla I.1 CATEGORIA DEL TERRENO SEGUN SU RUGOSIDAD

Cat.	Descripción	Ejemplos	Limitaciones
1	Terreno abierto, prácticamente plano y sin obstrucciones	Franjas costeras planas, zonas de pantanos, campos aéreos, pastizales y tierras de cultivo sin setos o bardas alrededor. Superficies nevadas planas	La longitud mínima de este tipo de terreno en la dirección del viento debe ser de 2000 m o 10 veces la altura de la construcción por diseñar, la que sea mayor.
2	Terreno plano u ondulado con pocas obstrucciones	Campos de cultivo o granjas con pocas obstrucciones tales como setos o bardas alrededor, árboles y construcciones dispersas	Las obstrucciones tienen alturas de 1.5 a 10 m, en una longitud mínima de 1500 m
3	Terreno cubierto por numerosas obstrucciones estrechamente espaciadas	Áreas urbanas, suburbanas y de bosques, o cualquier terreno con numerosas obstrucciones estrechamente espaciadas. El tamaño de las construcciones corresponde al de las casas y viviendas	Las obstrucciones presentan alturas de 3 a 5 m. La longitud mínima de este tipo de terreno en la dirección del viento debe ser de 500 m o 10 veces la altura de la construcción, la que sea mayor
4	Terreno con numerosas obstrucciones largas, altas y estrechamente espaciadas	Centros de grandes ciudades y complejos industriales bien desarrollados.	Por lo menos el 50% de los edificios tiene una altura mayor que 20 m. Las obstrucciones miden de 10 a 30 m de altura. La longitud mínima de este tipo de terreno en la dirección del viento debe ser la mayor entre 400 m y 10 veces la altura de la construcción.

Tabla I.2 CLASE DE ESTRUCTURA

Clase	Descripcion
A	Todo elemento estructural aislado, expuesto directamente a la accion del viento; Construcciones horizontales o verticales cuya mayor dimension, sea menor que 20 metros.
B	Construcciones horizontales o verticales cuya mayor dimension, varie entre 20 y 50 metros.
C	Construcciones horizontales o verticales cuya mayor dimension, sea mayor que 50

	metros.

Párrafo 4.6.2 MAPAS DE ISOTACAS. VELOCIDAD REGIONAL, V_R

La velocidad regional del viento, V_R, es la máxima velocidad media probable de presentarse con un cierto periodo de recurrencia en una zona o región determinada del país.

En los mapas de isotacas que se incluyen en este inciso con diferentes periodos de retorno, dicha velocidad se refiere a condiciones homogéneas que corresponden a una altura de 10 metros sobre la superficie del suelo en terreno plano (Categoría 2 según la tabla I.1); es decir, no considera las características de rugosidad locales del terreno ni la topografía específica del sitio. Asimismo, dicha velocidad se asocia con ráfagas de 3 segundos y toma en cuenta la posibilidad de que se presenten vientos debidos a huracanes en las zonas costeras.

La velocidad regional, V_R, se determina tomando en consideración tanto la localización geográfica del sitio de desplante de la estructura como su destino.

En las figures I.1 a I.4 se muestran los mapas de isotacas regionales correspondientes a periodos de recurrencia de 200, 50 y 10 anos, respectivamente.

La importancia de las estructuras (véase el inciso 4.3) dictamina los periodos de recurrencia que deberán considerarse para el diseño por viento; de esta manera, los Grupos A, B y C se asocian con los periodos de retorno de 200, 50 y 10 años, respectivamente. El sitio de desplante se localizará en el mapa con el periodo de recurrencia que corresponde al grupo al que pertenece la estructura a fin de obtener la velocidad regional. En el Tomo III de Ayudas de diseño se presenta una tabla con las principales ciudades del país y sus correspondientes velocidades regionales para diferentes periodos de retorno.

Párrafo 4.6.3 FACTOR DE EXPOSICION, Fα

El coeficiente Fα refleja la variación de la velocidad del viento con respecto a la altura Z. Asimismo, considera el tamaño de la construcción o de los elementos de recubrimiento y las características de exposición.

El factor de exposición se calcula con la siguiente expresión:

$$F\alpha = F_C \; F_{RZ}$$

en donde:

F_C : es el factor que determina la influencia del tamaño de la construcción, adimensional, y

F_{RZ} : el factor que establece la variación de la velocidad del viento con la altura Z en función de la rugosidad del terreno de los alrededores, adimensional.

Los coeficientes F_C y F_{RZ} se definen en los incisos 4.6.3.1 y 4.6.3.2, respectivamente.

Párrafo 4.6.3.1 FACTOR DE TAMAÑO, F_C α y δ

El factor de tamaño, F_C, es el que toma en cuenta el tiempo en el que la ráfaga del viento actúa de manera efectiva sobre una construcción de dimensiones dadas. Considerando la clasificación de las estructuras según su tamaño (véase la tabla I.2), este factor puede determinarse de acuerdo con la tabla I.3.

Tabla I.3 FACTOR DE TAMAÑO, F_C

Clase de estructura	**F_C**
A	1.0
B	0.95
C	0.90

δ : es la altura, media a partir del nivel del terreno de desplante, por encima de la cual la variación de la velocidad del viento no es importante y se puede suponer constante; a esta altura se le conoce como altura gradiente; δ y Z están dadas en metros, y

α : el exponente que determina la forma del la variación de la velocidad del viento con la altura y es adimensional.

Los coeficientes α y δ están en función de la rugosidad del terreno (tabla I.1). En la tabla I.4 se consignan los valores que se aconsejan para estos coeficientes. En la figura III.1 del Tomo de Ayudas de diseño se muestra la variación del factor Fα con la altura, con la categoría del terreno y con la clase de estructura.

Tabla I.4 VALORES DE α y δ

Categoría de terreno	α			δ
	Clase de estructura			
	A	**B**	**C**	
1	0.099	0.101	0.105	245
2	0.128	0.131	0.138	315
3	0.156	0.160	0.171	390
4	0.170	0.177	0.193	455

Párrafo 4.5.4 FACTOR DE TOPOGRAFIA, F_T

Este factor toma en cuenta el efecto topográfico local del sitio en donde se desplantara la estructura. Así, por ejemplo, si la construcción se localiza en las laderas o cimas de colinas o montañas de altura importante con respecto al nivel general del terreno de los alrededores, es muy probable que se generen aceleraciones del flujo del viento y, por consiguiente, deberá incrementarse la velocidad regional.

Tabla I.5 FACTOR DE TOPOGRAFIA LOCAL, F_T

Sitios	Topografía	F_T
Protegidos	Base de promontorios y faldas de serranías del lado de sotavento.	0.8
	Valles cerrados.	0.9
Normales	Terreno prácticamente plano, campo abierto, ausencia de cambios topográficos importantes, con pendientes menores que 5%.	1.0
Expuestos	Terrenos inclinados con pendientes entre 5 y 10%, valles abiertos y litorales planos.	1.1
	Cimas de promontorios, colinas o montañas, terrenos con pendientes mayores que 10%, cañadas cerradas y valles que formen un embudo o cañon, islas.	1.2

Expertos en la materia deberán justificar y validar ampliamente los resultados de cualquiera de estos procedimientos.

Párrafo 4.6.5 VALORES DE LA ALTITUD, h_m

Tabla I.7 RELACION ENTRE LOS VALORES DE LA ALTITUD, h_m

Altitud (msnm)	Presión barométrica (mm de H_g)
0	760
500	720
1000	675
1500	635
2000	600
2500	65
3000	530
3500	495

Nota: Puede interpolarse para valores intermedios de la altitud, h_m

Mexico 1993 Wind Data (page 411)

Tabla I.28 COEFICIENTE DE ARRASTRE, C_a

Sección transversal	Tipo de superficie	Relación H/b			
		1	7	25	≥ 40
Circular ($bV_D \geq 6\ m^2/s$)	Lisa o poco rugosa ($d'/b \approx 0.0$)	0.5	0.6	0.7	0.7
	Rugosa ($d'/b \approx 0.02$)	0.7	0.8	0.9	1.2
	Muy rugosa ($d'/b \approx 0.08$)	0.8	1.0	1.2	1.2
Circular ($bV_D \geq 6\ m^2/s$)	Cualquiera	0.7	0.8	1.2	1.2
Hexagonal u octagonal	Cualquiera	1.0	1.2	1.4	1.4
Cuadrada (viento normal a una cara)	Cualquiera	1.3	1.4	2.0	2.2
Cuadrada (viento sobre una esquina)	Cualquiera	1.0	1.1	1.5	1.6

b : es el diámetro o la dimensión horizontal de la estructura, incluyendo la rugosidad de la pared; para determinar el producto **bV_D**, este diámetro será el que se localiza a dos tercios de la altura total, a partir del nivel del terreno, en m.

d' : es la dimensión que sobresale de las rugosidades, tales como costillas o "spoilers", en m.

V_D : es la velocidad del viento de diseño (inciso 4.6), convertida a m/s, y valuada para los dos tercios de la altura total.

Para valores intermedios de **H/b** y **d'/b** se permite la interpolación lineal.

Párrafo 4.9.3.2 LAS VARIABLES κ' η δ:

Tabla I.29 FACTORES κ' η δ

Categoría	1	2	3	4
κ'	1.224	1.288	1.369	1.457
η	-0.032	-0.054	-0.096	-0.151
δ	245	315	390	455

Las variables κ' η δ, adimensionales, dependen de la rugosidad del sitio de desplante, y δ es la altura gradiente en m. Estas variables se definen en la tabla I.29.

k_r : es un factor relacionado con la rugosidad del terreno:

ζ : es el coeficiente de amortiguamiento critico:
para construcciones formadas por marcos de acero = 0.01
para construcciones formadas por marcos de concreto = 0.02

VALORES DE α' k_r

Categoría de terreno	**α'**	**k_r**
1	0.13	0.06
2	0.18	0.08
3	0.245	0.10
4	0.31	0.14

VALORES DE ζ

Nota:	**ζ**
Para construcciones formadas por marcos de acero	0.01
Para aquellas formadas por marcos de concreto	0.02

NUMERO DE STROUHAL

S_t : es el numero de Strouhal, adimensional; 0.2 para secciones circulares y 0.14 para las rectangulares.

NBC-2010 Wind Data

Enter data for the NBC 2010 wind code.

Reference Velocity Pressure (q)

Enter the reference value of the wind pressure, q. This varies according to geographical location and according to company or vendor standards.

Design Wind Speed

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are **85.0**, **100.0**, **110.0**, and **120.0**.

NOTE Use the lowest value reasonably allowed because the wind design pressure and force increase as the square of the speed.

Importance Factor (Iw)

Enter a value for the NBC wind importance factor, I_w. The software uses this value directly without modification. Values are taken from Table 4.1.7.1 of Division B.

Category	**Importance Factor, I_w**	
	ULS	**SLS**
Low	**0.8**	**0.75**
Normal	**1**	**0.75**
High	**1.15**	**0.75**
Post Disaster	**1.25**	**0.75**

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant

Select an NBC exposure factor, as defined in UBC-91, Section 2312:

- **Not Used**
- **Exposure A** - **Open/Standard** exposure.

  NOTE Most petrochemical sites use exposure A.
- **Exposure B** - **Rough** areas similar to urban and suburban areas.
- **Exposure C** - **Very Rough** area similar to centers of large cities.

NOTE These exposure factors are in reverse order from the exposure factors of ASCE-7 or UBC.

Surface Roughness

Select the roughness factor:

- **Moderately smooth**
- **Rough surface** (D'/D = 0.02)
- **Very rough surface** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Moderately smooth**.
- You can use **Very rough surface** to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.
- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters**. On the **Job Specific Setup Parameters** tab, enter that value in the **Wind Shape Factor** dialog box.

Hill Height [Hh]

Enter the height of a hill or escarpment relative to the upwind terrain, H_h. For more information, see ASCE 7-10, Figure 29.8-1.

Distance to Site [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. For more information, see ASCE 7-10, Figure 29.8-1.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment, L_h. For more information, see ASCE 7-10, Figure 29.8-1.

Type of Hill

Select the type of hill: **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. For more information, see ASCE 7-10, Figure 29.8-1.

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

NBC-2015 Wind Data

Enter data for the NBC 2015 wind code.

Reference Velocity Pressure [q]

Enter the reference value of the wind pressure, q. This varies according to geographical location and according to company or vendor standards.

Design Wind Speed

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are **85.0**, **100.0**, **110.0**, and **120.0**.

NOTE Use the lowest value reasonably allowed because the wind design pressure and force increase as the square of the speed.

Importance Factor [Iw]

Enter a value for the NBC wind importance factor, I_w. The software uses this value directly without modification. Values are taken from Table 4.1.7.3 of Division B.

Category	Importance Factor, I_w	
	ULS	**SLS**
Low	**0.8**	**0.75**
Normal	**1**	**0.75**
High	**1.15**	**0.75**
Post Disaster	**1.25**	**0.75**

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant

Select an NBC exposure factor, as defined in UBC-91, Section 2312:

- **Not Used**
- **Exposure A** - **Open/Standard** exposure.

  NOTE Most petrochemical sites use exposure A.

- **Exposure B** - **Rough** areas similar to urban and suburban areas.
- **Exposure C** - **Very Rough** area similar to centers of large cities.

NOTE These exposure factors are in reverse order from the exposure factors of ASCE-7 or UBC.

Surface Roughness

Select the roughness factor:

- **Moderately smooth**
- **Rough surface** (D'/D = 0.02)
- **Very rough surface** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Moderately smooth**.
- You can use **Very rough surface** to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.

- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters**. On the **Job Specific Setup Parameters** tab, enter that value in the **Wind Shape Factor** dialog box.

Hill Height [Hh]

Enter the height of a hill or escarpment relative to the upwind terrain, H_h. For more information, see ASCE 7-10, Figure 29.8-1.

Distance to Site [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. For more information, see ASCE 7-10, Figure 29.8-1.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment, L_h. For more information, see ASCE 7-10, Figure 29.8-1.

Type of Hill

Select the type of hill: **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. For more information, see ASCE 7-10, Figure 29.8-1.

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

NBC-95 and NBC-2005 Wind Data

Reference Wind Pressure

Enter the reference value of the wind pressure, q. This varies according to geographical location and according to company or vendor standards.

Design Wind Speed

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are **85.0**, **100.0**, **110.0**, and **120.0**.

NOTE Use the lowest value reasonably allowed because the wind design pressure and force increase as the square of the speed.

Importance Factor (Iw)

Enter a value for the NBC wind importance factor, I_w. The software uses this value directly without modification. Values are taken from Table 4.1.7.1 of Division B.

Category	**Importance Factor, I_w**	
	ULS	**SLS**
Low	**0.8**	**0.75**
Normal	**1**	**0.75**
High	**1.15**	**0.75**
Post Disaster	**1.25**	**0.75**

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant

Select an NBC exposure factor, as defined in UBC-91, Section 2312:

- **Not Used**
- **Exposure A** - **Open/Standard** exposure.

  NOTE Most petrochemical sites use exposure A.

- **Exposure B** - **Rough** areas similar to urban and suburban areas.
- **Exposure C** - **Very Rough** area similar to centers of large cities.

NOTE These exposure factors are in reverse order from the exposure factors of ASCE-7 or UBC.

Roughness Factor

Select the roughness factor:

- **Moderately smooth**
- **Rough surface** (D'/D = 0.02)
- **Very rough surface** (D'/D = 0.08)

NOTES

- Most petrochemical sites use **Moderately smooth**.
- You can use **Very rough surface** to account for items such as platforms, piping, and ladders, instead of either entering them explicitly as a tributary wind area or implicitly as an increased wind diameter.

- If your specification calls out for a defined value of the wind shape factor, go to the **Tools** tab and click **Set Configuration Parameters**. On the **Job Specific Setup Parameters** tab, enter that value in the **Wind Shape Factor** dialog box.

Hill Height [Hh]

Enter the height of a hill or escarpment relative to the upwind terrain. See ASCE 7-95/98, Fig. 6-2 for details.

Distance to site [x]

Enter the distance (upwind or downwind) from the crest of the hill to the building site. See ASCE 7-95/98 Fig. 6-2 for details.

Crest Distance [Lh]

Enter the distance upwind of the crest of the hill to where the difference in ground elevation is half the height of the hill or escarpment. See ASCE 7-95/98, Fig. 6-2 for more details.

Type of Hill

Select the type of hill. Select **None**, **2-D Ridge**, **2-D Escarpment**, or **3-D Axisym Hill**. See ASCE 7-95/98, Fig. 6-2 for details.

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

SANS 10160-3:2010 Wind Data

South African National Standard (SANS) wind data code guidelines detail standards for determining natural wind actions for the structural design of buildings and industrial structures. The standard focuses on predicting the characteristics of wind actions on land-based structures with an overall height of 100 m or less, elements of buildings and structures having a natural frequencies greater than 5 Hz, and chimneys with circular cross-sections with heights of less than 60 m and a height-to-diameter ratio of less than 6, 5.

NOTE This part of SANS 10160 does not apply to structures and buildings higher than 100 m, dynamic effects and design of dynamically-sensitive structures (such as slender chimneys), or off-shore structures.

Percent Wind for Hydrotest

Enter the percentage of the wind load (not wind speed) that is applied during the hydrotest. This is typically as low as 33 percent of the design wind load, because it can be assumed that the vessel will not be hydrotested during a hurricane or severe storm.

Basic Wind Speed

Enter the basic wind speed for the vessel location, which you can find from the wind speed map of South Africa in SANS 10160-3 or as needed by end user requirements.

Terrain Category

Enter the terrain roughness category:

Category	Description
A	Flat, horizontal terrain with negligible vegetation and without any obstacles (for example, coastal areas exposed to open sea or large lakes).
B	Area with low vegetation, such as grass and isolated obstacles (for example, trees and buildings) with separations of at least 20 obstacle heights.
C	Area with regular cover of vegetation or buildings or with isolated obstacles with separations of maximum 20 obstacle heights (such as villages, suburban terrain, and permanent forest).
D	Area in which at least 15 percent of the surface is covered with buildings and their average height exceeds 15 m.

For more information on the roughness factors and height parameters that apply to the various terrain categories, see Section 7.3.2, *Terrain roughness,* in SANS 10160-3:2010, Edition 1.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than if it were mounted at grade.

Site Altitude Above Sea Level

Enter the site altitude above mean sea level. Refer to Table 4, *Air density as a function of site altitude*, in Section 7.4 of SANS 10160-3:2010 for the recommended values of the air density as a function of the altitude above sea level.

Topography Factor (Co)

Enter the topography factor (Co), which determines the largest increase in the window speed near the top of the slope. The topography factor accounts for the increase of mean wind speed over isolated hills and escarpments only, and is also determined by the wind speed at the base of the hill or escarpment. For more information on when to consider the effects of topography, see SANS 10160-3:2010, *Section A.3 Numerical Calculations of Topography Coefficients*.

Force Coefficient (Cf)

Enter the force coefficient value (Cf), which is the overall effect of the wind on a structure, structural element, or component as a whole. The force coefficient includes friction, if not specifically excluded.

Beta: Operating

In the first text box, enter the value of the structural damping coefficient (percentage of critical damping) operating-case beta for vortex shedding. The default value is 0.01. This value is used to compute the dynamic gust effect factor, G, as outlined in commentary section 6.6, page 158, of ASCE 7-95; section 6.5.8, pages 29-30, of ASCE 7-98/02/05; or section 26.5, page 255, of ASCE 7-10. If your design code is not ASCE, then the software uses the damping coefficient according to that wind design code. If your design code does not call out for a specific value of beta, then use the default value of **0.01**.

Beta: Empty

In the second text box, enter a value for the empty-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

Beta: Full

In the third text box, enter a value for the filled-case beta. The software calculates a gust response factor based on these values of beta and generates additional wind loads on the vessels. If the code does not supply this value, then leave the text box empty.

UBC Wind Data

Design Wind Speed

Enter the design value of the wind speed. This varies according to geographical location and according to company or vendor standards. Typical wind speeds in miles per hour are 85.0, 100.0, 110.0, and 120.0.

NOTE Enter the lowest value reasonably allowed by the standards that you are following. The reason for this is that the wind design pressure, and thus force, increases as the square of the speed.

Base Elevation

Enter the elevation at the base of the vessel. This value is used to calculate the height of each point in the vessel above grade. For example, if the vessel is mounted on a pedestal foundation or on top of another vessel, it is exposed to higher wind pressures than when it is mounted at grade.

Exposure Constant

Select an UBC exposure factor, as defined in UBC-91 Section 2312:

- **Exposure B** - Terrain with building, forest, or surface irregularities 20 feet or more in height, covering at least 20 percent or the area extending one mile or more from the site.
- **Exposure C** - Terrain that is flat and generally open, extending one-half mile or more from the site in any full quadrant.

- **NOTE** Most petrochemical sites use this value. It is used to set the gust factor coefficient (Ce) found in Table 23-G.
- **Exposure D** - The most severe exposure with basic wind speeds of 80 miles per hour or higher. Terrain is flat and unobstructed, facing large bodies of water over one mile or more in width relative to any quadrant of the building site. This exposure extends inland from the shoreline one quarter mile or zero times the building (vessel) height, whichever is greater.
- **Not Used**

Importance Factor

Enter a value for the UBC importance factor. The software uses this value directly without modification. Values are taken from Table 23-L of the UBC standard:

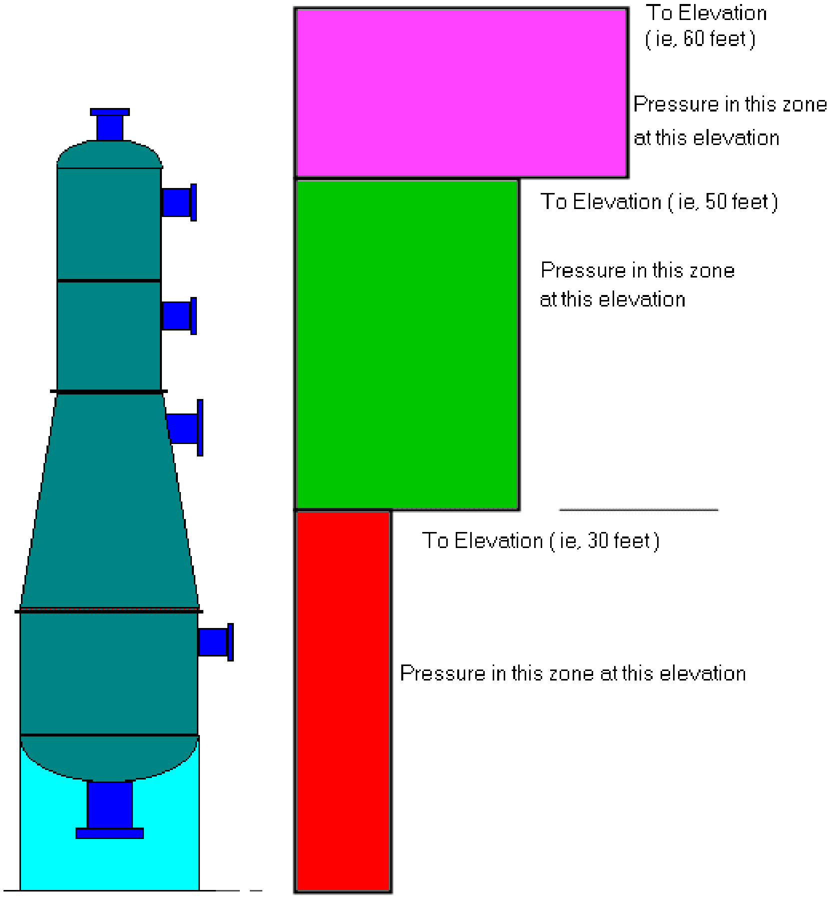
Category	**Factor**	**Classification**
I	**1.15**	Essential facilities
II	**1.15**	Hazardous facilities
III	**1.0**	Special occupancy structures
IV	**1.0**	Standard occupancy structures

User-Defined Wind Data

When **User Defined** is selected for **Wind Design Code**, enter values for wind pressure versus elevation in the table. The software uses only the values from the table, forgoing all code calculations. Enter **"To" Height**, the height above grade, in the left text boxes, and **Pressure**, the wind pressure at that height, in the right text boxes. If you have more rows available than you need to describe the wind profile, enter zeros in the remaining cells.

NOTE Multiply the wind pressure at each elevation by the shape factor you wish to use. If you do not your wind loads will be higher (and more conservative) than needed.

The first **"To" Height** value cannot be zero. If it is zero the software will not calculate the wind loads on the following elements. Values should follow the convention below



SECTION 16

Seismic Loads (Seismic Data Tab)

Click **Input** > **Seismic Loads** to enter seismic data. The **Seismic Data** tab appears on the left. Select a seismic design code and enter data required by that code. If no data is entered, the software uses the system defaults.

Seismic Design Code

Select the design code to use for seismic calculations:

Code	Description
As/Nz 1170.4	Australian Standard Part 4: Earthquake Loads, 1993 edition. For more information, see *AS/NZ 1170.4 Seismic Data* (page 434).
ASCE-88 **ASCE-93** **ASCE-95** **ASCE-7-98** **ASCE 7-02/05** **ASCE-2010**	American Society of Civil Engineers Standard 7 (formerly ANSI A58.1). For more information, see: *ASCE-88 Seismic Data* (page 437) *ASCE-93 Seismic Data* (page 439) *ASCE-95 Seismic Data* (page 440) *ASCE 7-98 Seismic Data* (page 442) *ASCE 7-02/05 Seismic Data* (page 445) *ASCE-2010 Seismic Data* (page 448)
Chile NCh2369	NCh2369 - Chilean Code for Seismic Analysis of Vessels - 2003. For more information, see *Chile NCh2369 Seismic Data* (page 451).
China GB 50011	Chinese Seismic Code. For more information, see *China GB 50011 Seismic Data* (page 461).
Costa Rica 2002	Costa Rica Seismic Code 2002. For more information, see *Costa Rica 2002 Seismic Data* (page 461).
G Loading	Enter an appropriate G loading. The software calculates the mass of the element and multiplies by the G loading in the appropriate direction. For vertical vessels, the maximum of the X and Z acceleration values are chosen for the analysis. For horizontal vessels, all three values are used. For more information, see *G Loading Seismic Data* (page 467).
IBC 2000 **IBC 2003** **IBC 2006** **IBC 2009** **IBC 2012** **IBC 2015** **IBC 2018**	International Building Code 2000, 2003, 2006, 2009, 2012, 2015, and 2018 editions. For more information, see: *IBC 2000 Seismic Data* (page 467) *IBC 2003 Seismic Data* (page 470) *IBC 2006 Seismic Data* (page 473) *IBC 2009 Seismic Data* (page 476) *IBC 2012 Seismic Data* (page 480) *IBC 2015 Seismic Data* (page 483) *IBC 2018 Seismic Data* (page 486)

IS-1893 SCM **IS-1893 RSM**	India's seismic code: Seismic Coefficient Method (SCM), 1984 edition and Response Spectrum Method (RSM), 1984 and 2002 editions. For more information, see: *IS-1893 RSM Seismic Data* (page 489) *IS-1893 SCM Seismic Data* (page 491)
KBC 2016	Korean Seismic Code. For more information, see *KBC 2016 Seismic Data* (page 493).
KHK 1997	Japan Seismic Code. For more information, see *KHK 2012 Seismic Data* (page 496).
Mexico Sismo	Mexican Seismic Code. For more information, see *Mexico Sismo Seismic Data* (page 501).
NBC-1995 **NBC-2005** **NBC-2010** **NBC-2015**	National Building Code of Canada, 1995, 2005, 2010, and 2015 editions. For more information, see: *NBC 1995 Seismic Data* (page 504) *NBC 2005 Seismic Data* (page 506) *NBC 2010 Seismic Data* (page 507) *NBC 2015 Seismic Data* (page 509)
PDVSA	Venezuelan Code for Seismic Analysis of Vessels - 1999. For more information, see *PDVSA Seismic Data* (page 510).
Res. Spectrum	Response Spectrum Analysis allows the use of modal time history analysis. The general design guidelines for this analysis are taken from the ASCE 7-98 or IBC 2000 codes. Other predefined spectra are built into the program, such as the 1940 Earthquake El Centro and various spectra form the US NRC guide 1.60. If the spectrum analysis type is user defined, the table of points that define the response spectra must be entered in the table. For tall structures, this analysis gives much more accurate results than the typical static equivalent method. Usually the calculated loads are lower in magnitude than those computed using conventional building code techniques. For more information, see *Res. Spectrum Seismic Data* (page 522).
SANS 10160-4:2010	South African National Standard 10160, Section 2, 2010 edition. For more information, see *SANS 10160-4:2010 Seismic Data* (page 527).
UBC 1994 **UBC 1997**	Uniform Building Code, 1994 and 1997 editions. For more information, see: *UBC 1994 Seismic Data* (page 528) *UBC 1997 Seismic Data* (page 529)
No Seismic Loads	

Percent Seismic for Hydrotest

Enter the percentage of the total seismic horizontal force that is applied during hydrotest. Although you cannot predict an earthquake (as you can high winds) some designers use a reduced seismic load for hydrotest on the theory that the odds of an earthquake during the test are very low, and the hazards of a water release small.

In This Section

AS/NZ 1170.4 Seismic Data

Enter data for the Australian/New Zealand AS/NZ-1170.4 seismic code, 1993 or 2007.

Code Year

Select the revision year of the code: **1993** or **2007**.

Importance Factor (I)

Enter the value of the importance factor *I*. The software uses this value directly without modification. Values are taken from Table 16-K of the UBC 1997 standard:

Structure Type	Description	Importance Factor
I	Structures include buildings not of type II or type III.	**1.0**
II	Structures include buildings that are designed to contain a large number of people, or people of restricted or impaired mobility.	**1.0**
III	Structures include buildings that are essential to post-earthquake recovery or associated with hazardous facilities.	**1.25**

Structural Response Factor (Rf)

Enter the structural response factor R_f, taken from table 6.2.6(b) of the code. For vessels on legs, use **2.1**. For towers, stacks and chimney type structures, use **2.8**.

Site Factor (S)

Enter the site factor *S*. taken from table 2.4(a) or 2.4(b). The factor is a function of the type of soil on which the vessel sits. This value can range between 0.67 and 2.0. A value of 2 is the most conservative and represents a vessel sitting on a foundation of loose sand or clay, while 0.67 represents a vessel sitting on a rock bed.

Soil Profile Table 2.4(a) **General Structures**	**Site factor (S)**
A profile of rock materials with rock strength Class L (low) or better.	**0.67**
A soil profile with either: ▪ Rock materials Class EL (extreme low) or VL (very low). ▪ Not more than 30m of medium dense to very dense coarse sands and gravels; firm, stiff or hard clays; or controlled fill.	**1.0**
A soil profile with more than 30m of: medium dense to very dense coarse sands and gravels; firm, stiff or hard clays; or controlled fill.	**1.25**
A soil profile with a total depth of 20m or more and containing 6 to 12m of: very soft to soft clays; very loose or loose sands; silts; or uncontrolled fill.	**1.5**
A soil profile with more than 12m of: very soft to soft clays; very loose or loose sands; silts; or uncontrolled fill characterized by shear wave velocities less than 150m/s.	**2.0**

Acceleration Coefficient (ax)

Enter the acceleration coefficient a_x, taken from Table 2.3 or Figures 2.3(b) to 2.3(g). This value ranges from 0.04 to 0.22. The higher the acceleration coefficient, the higher the load on the vessel.

Design Category

Select the design category **A**, **B**, **C**, **D**, or **E**. The software uses this value to determine if it is necessary to apply vertical accelerations. If the selected category is **D** or **E**, vertical accelerations are applied. The vertical acceleration is taken to be 0.5 times **Acceleration Coefficient (ax)** in the horizontal direction, according to paragraph 6.8.

Probability Factor (kp)

Enter a value for the probability factor k_p.

Hazard Factor (Z)

Enter the hazard factor.

Structural Performance Factor (Sp)

Enter the structural performance factor.

Structural Ductility Factor (mu)

Enter the structural ductility factor.

Soil Type

Enter the soil type.

Verify base shear meets 80% criteria, per 6.2.3?

Select this option to verify that the value for the base shear meets the required standards as defined in AS/NZ 1170.4 6.2.3. This section states that the base shear must be no less than 80% of the fundamental period of a structure.

When you select this option, a value for **Kt** must be entered.

When you select this option, calculations for the **Natural Period of the structure per...6.2(7)[T1]** and **Design Base Shear per...6.2(3)[VT1]** display on the **Earthquake Load Calculation** output report.

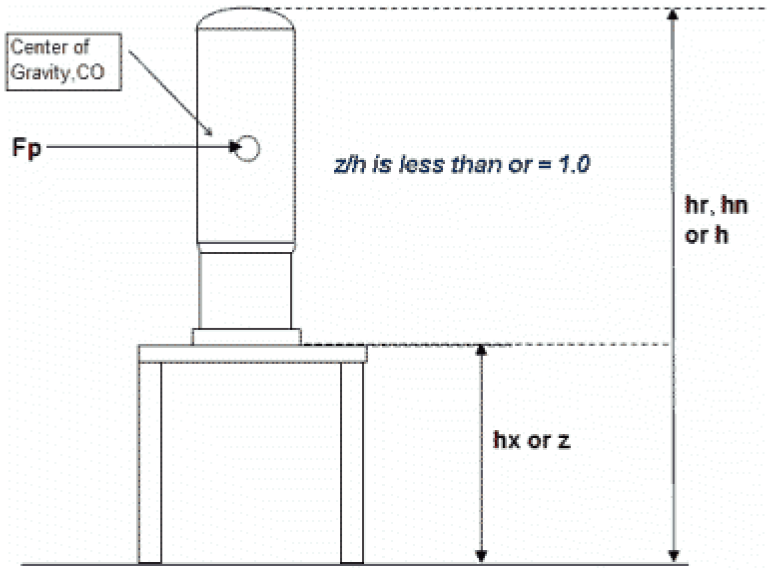
Kt

Enter the factor for determining the building period k_t, taken from section 6.2.3 of the code.

Typical Value	Description
0.11	For moment-resisting steel frames
0.075	For moment-resisting concrete frames
0.06	For eccentrically-braced steel frames
0.05	For all other structures

Component Elevation Ratio (hx/hn)

Enter a value for the elevation ratio if the vessel is supported by a building. The ratio, needed for proper analysis, is the height from the bottom of the vessel *hx* to the building height *hn*. A value for **Attachment Factor (ac)** must also be entered. The base shear is calculated according to equation 5.2.1. After the base shear F_p (V) is calculated, it is applied according to the equations in section 6 of the code.



Attachment Amplification Factor (ac)

Enter the attachment amplification factor a_c. This value is typically **1.0** unless unusually flexible connections are provided (which is not for vessels). Because the load is in linear proportion with a_c, the higher the value is, the higher the load becomes. You must also enter a value for **Component Elevation Ratio (hx/hn)**.

ASCE-88 Seismic Data

Enter data for the ASCE 7-88 seismic code.

Importance Factor

Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from Table 22 of the ASCE 7-88 standard:

Category	Classification	Importance Factor
I	Buildings not listed below. **NOTE** Most petrochemical structures use this category.	**1.0**
II	High occupancy buildings.	**1.25**
III	Essential facilities.	**1.5**
IV	Low hazard buildings.	not applicable

Soil Type

Select a soil type:

- **Soil 1** - Soil Profile S1. Rock or stiff soil conditions (S Factor = 1.0).
- **Soil 2** - Soil Profile S2. Deep cohesionless deposits or stiff clay conditions (S Factor = 1.2).
- **Soil 3** - Soil Profile S3. Soft to medium-stiff clays and sands (S Factor = 1.5).

The soil type indicates a soil profile coefficient *S*, found Table 24 of the code. Soil profiles are defined in Section 9.4.2 of the code. When soil properties are not known, select **Soil 1** or **Soil 2**, based on the value producing the larger value of CS (C is defined in Eq. 8 of the code).

Horizontal Force Factor

Enter the seismic force factor according to ANSI A58.1, Table 24:

Typical Value	Description
1.33	Buildings with bearing walls
1.00	Buildings with frame systems
2.50	Elevated tanks
2.00	Other structures

NOTE The value most often used is 2.0, though 2.5 is sometimes chosen for tanks supported by structural steel or legs.

Seismic Zone

Select the seismic zone, according to ASCE 7-93, Figure 14:

Zone	Description
0	Gulf and other coastal areas
1	Inland areas
2	Rockies, other mountain areas
3	Central Rockies, other mountains
4	California fault areas

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 4 indicates the greatest chance of an earthquake.

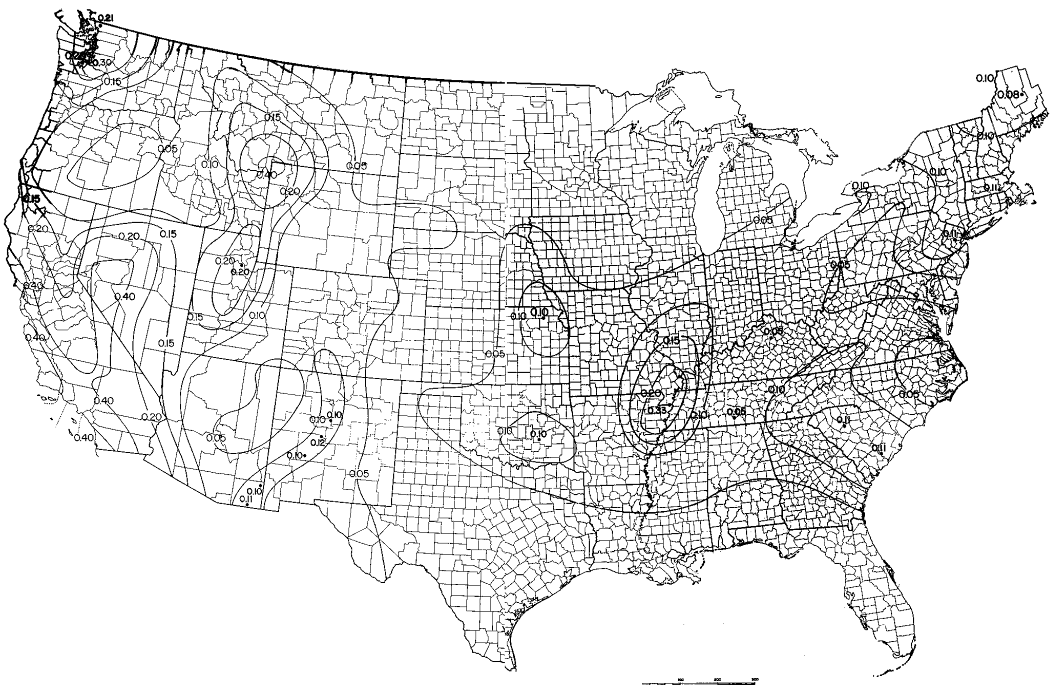
ASCE-93 Seismic Data

Enter data for the ASCE 7-93 seismic code.

NOTE The ASCE 7 earthquake standards released in 1993 are significantly more involved than the previous standards, more strictly limited to buildings, and thus not as easily applied to vessels. The software does not implement the dynamic analysis according to this standard. However, it does address the computation of the element mass multiplier as outlined on page 62 of the standard. In effect, the factors Av, Cc, P, and ac are first multiplied together and then by the weight of the element to obtain the lateral force on the element. The software then calculates the moments on the tower based on these results. You should have a good understanding of this code before using it.

Seismic Coefficient Av

Enter the seismic coefficient *Av*, representing the effective peak velocity-related acceleration according to the ASCE 7-93 standard, *Minimum Design Loads for Buildings and Other Structures*. This value is obtained from the map on pages 36 and 37 of this standard. According to section 9.1.4.1 of the ASCE Code, this value generally ranges from .05 (low incidence of earthquake) to .4 (high incidence of earthquake).



Seismic Coefficient Cc

Enter the seismic coefficient *Cc* for mechanical and electrical components from table 9.8-2, page 63 of the ASCE 7-93 Code. For tanks, vessels and heat exchangers this value is normally taken as **2**.

Performance Factor

Enter the performance criteria factor *P,* from table 9.8-2, page 63 of the ASCE 7-93 code. This factor depends on the Seismic Hazard Exposure Group., defined in Section 9.1.4.2:

P	Definition
1.5	Seismic Hazard Exposure Group III: Essential facilities required for post-earthquake recovery
1.0	Seismic Hazard Exposure Group II: Buildings that have a substantial public hazard due to occupancy or use
0.5	Seismic Hazard Exposure Group I: All other buildings

Amplification Factor

Enter the amplification factor ac, according to ASCE 7-93, Table 9.8-3:

Component Supporting Mechanism	Attachment Amplification Factor (a_c)
Fixed or direct connection seismic-activated restraining device	1.0
Resilient support system where: Tc/T < 0.6 or Tc/T > 1.4 Tc/T D 0.6 or Tc/T W 1.4	 1.0 2.0

NOTE **T** is the fundamental period of the building in seconds. **Tc** is the fundamental period of the component in seconds.

ASCE-95 Seismic Data

Enter data for the ASCE 7-95 seismic code.

Importance Factor

Enter the importance factor, as given in paragraph 9.3.1.5 of ASCE 95. The value is **1.5** for the following situations:

- Life-safety component required to function after an earthquake (such as, fire protection sprinkler system).
- Component contains material that would be significantly hazardous if released.
- Component poses a significant life safety hazard if separated from primary structure.
- Component can block a means of egress or exit way if damaged (such as, exit stairs).

Otherwise the value is **1.0**. The value may also be specified by the customer, but should always be greater than or equal to 1.0.

Force Factor R

Enter the seismic force factor according to ANSI A58.1, Table 24:

Typical Value	Description
1.33	Buildings with bearing walls
1.00	Buildings with frame systems
2.50	Elevated tanks
2.00	Other structures

NOTE The value most often used is 2.0, though 2.5 is sometimes chosen for tanks supported by structural steel or legs.

Seismic Coefficient Ca

Enter the value of seismic coefficient *Ca* (for shaking intensity) according to table 9.1.4.2.4A on page 55 of ASCE7-95. This factor is a function of the soil profile type and the value of Aa.

Soil Profile Type	Seismic Coefficient *Ca* for:						
	Aa<0.05g	**Aa=0.05g**	**Aa=0.10g**	**Aa=0.20g**	**Aa=0.30g**	**Aa=0.40g**	**Aa>=0.5g** [b]
A	**Aa**	**0.04**	**0.08**	**0.16**	**0.24**	**0.32**	**0.40**
B	**Aa**	**0.05**	**0.10**	**0.20**	**0.30**	**0.40**	**0.50**
C	**Aa**	**0.06**	**0.12**	**0.24**	**0.33**	**0.40**	**0.50**
D	**Aa**	**0.08**	**0.16**	**0.28**	**0.36**	**0.44**	**0.50**
E	**Aa**	**0.13**	**0.25**	**0.34**	**0.36**	**0.36**	**a**

a - Site specific geotechnical information and dynamic site response analyses shall be performed.

b - Site specific studies required per Section 9.2.2.4.3 may result in higher values of *Av* than included on hazard maps, as may the provisions of Section 9.2.6.

NOTE For intermediate values, the higher value of straight-line interpolation shall be used to determine the value of *Ca*.

Seismic Coefficient Cv

Enter the value of seismic coefficient *Cv* (for shaking intensity) according to table 9.1.4.2.4B on page 55 of ASCE7-95. This factor is a function of the soil profile type and the value of Aa.

Soil Profile Type	Seismic Coefficient *Cv* for:						
	Aa<0.05g	**Aa=0.05g**	**Aa=0.10g**	**Aa=0.20g**	**Aa=0.30g**	**Aa=0.40g**	**Aa>=0.5g**

							b
A	**Av**	**0.04**	**0.08**	**0.16**	**0.24**	**0.32**	**0.40**
B	**Av**	**0.05**	**0.10**	**0.20**	**0.30**	**0.40**	**0.50**
C	**Av**	**0.09**	**0.17**	**0.32**	**0.45**	**0.56**	**0.65**
D	**Av**	**0.12**	**0.24**	**0.40**	**0.54**	**0.64**	**0.75**
E	**Av**	**0.18**	**0.35**	**0.64**	**0.84**	**0.96**	**a**
	a - Site specific geotechnical information and dynamic site response analyses shall be performed. *b* - Site specific studies required per Section 9.2.2.4.3 may result in higher values of *Av* than included on hazard maps, as may the provisions of Section 9.2.6.						

NOTE For intermediate values, the higher value of straight-line interpolation shall be used to determine the value of *Cv*.

ASCE 7-98 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	**F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:**				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**

D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
Site Class	**S_l <= 0.1**	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5**[2]
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Factor tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

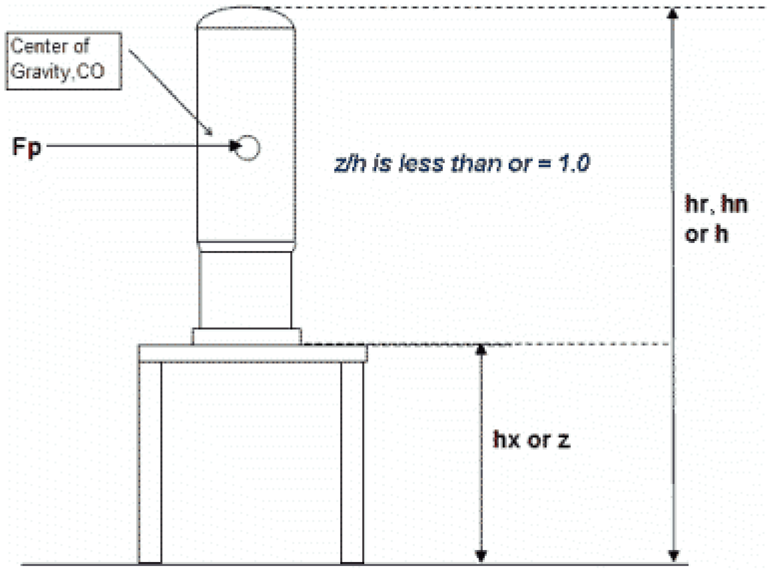
Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code?

Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

ASCE 7-02/05 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
	S_s <= 0.25	S_s = 0.5	S_s = 0.75	S_s = 1.0	S_s >= 1.25[2]
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.2	1.2	1.1	1.0	1.0
D	1.6	1.4	1.2	1.1	1.0
E	2.5	1.7	1.2	0.9	[0.9][1]
F	See note 1				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
	S_l <= 0.1	S_l = 0.2	S_l = 0.3	S_l = 0.4	S_l >= 0.5[2]
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.7	1.6	1.5	1.4	1.3
D	2.4	2.0	1.8	1.6	1.5
E	3.5	3.2	2.8	2.4	[2.4][1]
F	See note 1				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Factor tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

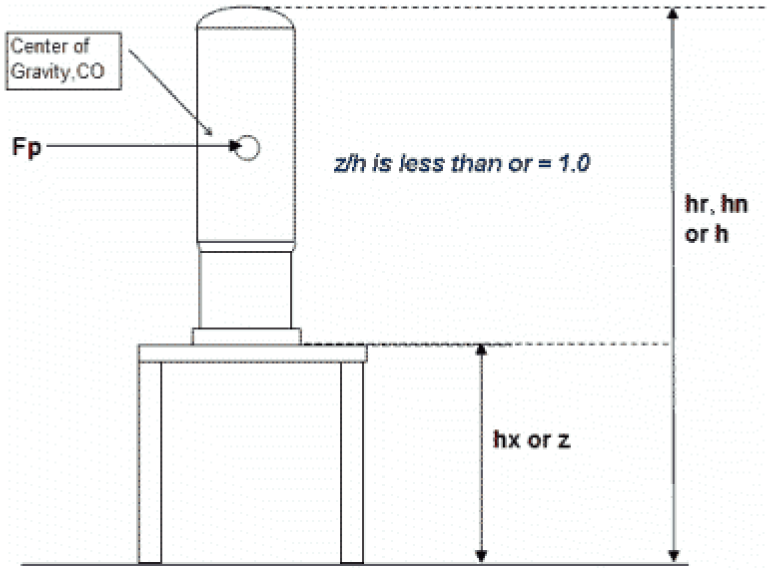
Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code?

Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

ASCE-2010 Seismic Data

Enter data for the ASCE 7-2010 seismic code.

Importance Factor I

Enter the occupancy importance factor, I, according to Sections 11.5.1 and 15.4.1.1, based on Table 1.5-2. The importance factor accounts for loss of life and property. This value typically ranges between **1.0** and **1.5**.

Response Modification Factor R

Enter the seismic force factor, R, from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2, as required. R is usually equal to **2.5** for inverted pendulum systems and cantilevered column systems. For elevated tanks use a value of **4**. For horizontal vessels, leg supported vessels and others use a value of **3.0**.

Acc. based Factor Fa

Enter the short-period site coefficient (at 0.2 s-period), F_a, from Table 11.4-1 as required. For more information, see Section 11.4.3.

Site Class	**S_S<=0.25**	**S_S=0.5**	**S_S=0.75**	**S_S=1.0**	**S_S>=1.25[b]**
A	0.8	0.8	0.8	0.8	0.8

B	1.0	1.0	1.0	1.0	1.0
C	1.2	1.2	1.1	1.0	1.0
D	1.6	1.4	1.2	1.1	1.0
E	2.5	1.7	1.2	0.9	[0.9][a]
F	See Section 11.4.7	a	a	a	a

Acc based Factor Fv

Enter the long-period site coefficient (at 1.0 s-period), F_v, from Table 11.4-2 as required. For more information, see Section 11.4.3.

Site Class	S_1<=0.1	S_1=0.2	S_1=0.3	S_1=0.4	S_1>=0.5[b]
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.7	1.6	1.5	1.4	1.3
D	2.4	2.0	1.8	1.6	1.5
E	3.5	3.2	2.8	2.4	[2.4][a]
F	See Section 11.4.7				

[a] Site specific geotechnical information and dynamic site response analyzes shall be performed.

[b] Site specific studies required per Section 9.4.1.2.4 might result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.

NOTE For intermediate values, the higher value of straight-line interpolation shall be used to determine the value of S_s or S_l.

Max. Mapped Res. Acc. Ss

Enter the value for S_s, the risk-adjusted maximum considered earthquake ground motion parameter for short periods (0.2 seconds), shown in Figures 22-1, 22-3, 22-5, and 22-6. When S_1 is less than or equal to 0.04, and S_s is less than or equal to 0.15, the structure is permitted to be assigned to Seismic Design Category A and is only required to comply with Section 11.7. Do not use percentage values in these fields; use a multiplier of g. For example, if S_s is 25%, use **0.25**.

Max Mapped Res. Acc. S1

Enter the value for S_1, the risk-adjusted maximum considered earthquake ground motion parameter for long periods (1.0 seconds), shown in Figures 22-2, 22-4, 22-5, and 22-6. When S_1 is less than or equal to 0.04, and S_s is less than or equal to 0.15, the structure is permitted to be assigned to Seismic Design Category A and is only required to comply with Section 11.7. Do not use percentage values in these fields; use a multiplier of g. For example, if S_s is 25%, type **0.25**.

Moment Reduction Value tau

This value is no longer used.

Site Class

Select the site class, as defined by Table 20.3-1.

Class	Description
A	Hard Rock
B	Rock
C	Very dense soil and soft rock
D	Stiff soil
E	Soft clay soil
F	Soils requiring site response analysis in accordance with Section 21.1

Component Elevation ratio z/h

Enter the elevation ratio, z/h. If the vessel is attached to another structure, such as a building, the value of the elevation ratio needs to be used for proper analysis. This ratio is the height in the structure where the vessel is attached to the average height of the roof with respect to the base. This value is generally less than or equal to **1**. For more information, see Section 13.3.1.

Component Amplification Factor ap

Enter a value between **1.0** to **2.5**. For vessels, a value of **2.5** is typical. For more information, see Section 13.3.1.

Consider Vertical Accelerations

Select to enable vertical load calculations (YEq).

Force Factor

Enter the force factor applied to the shear vertical load.

Minimum Acceleration Multiplier

Enter the force factor, G_y, applied to the total weight of the vessel.

Sds

Enter the value for S_{DS}, the 5 percent damped, spectral response acceleration parameter at short periods, as defined in Section 11.4.4.

NOTE The software checks Tables 11.6-1 and 11.6-2 to determine the seismic use group. If S_{D1} is greater than or equal to **0.2** and S_1 is greater than or equal to **0.75** then the additional check of equation 16-38 (IBC) is made. The same logic is used if S_{DS} is greater than or equal to **0.50**.

Sd1

Enter the value for S_{DS}, the 5 percent damped, spectral response acceleration parameter at a period (1 second), defined in Section 11.4.4.

NOTE The software checks Tables 11.6-1 and 11.6-2 to determine the seismic use group. If S_{D1} is greater than or equal to **0.2** and S_1 is greater than or equal to **0.75** then the additional check of equation 16-38 (IBC) is made. The same logic is used if S_{DS} is greater than or equal to **0.50**.

Chile NCh2369 Seismic Data

Chile NCh2369 En Español (page 454)

Enter data for the Chilean Code for Seismic Analysis of Vessels.

Eff. Max. Ground Acc. (Ao)

Enter the effective maximum ground acceleration A_o, as defined in Table 5.2 Maximum Effective Acceleration, NCh2369 page 39:

Seismic Zone	A_o
1	0.20 g
2	0.30 g
3	0.40 g

Soil Parameter (T')

Enter the soil parameter *T'*, as defined in Table 5.4 Value of type of soil dependent parameters, NCh2369 page 40:

Type of Soil	T' (s)	n
I	0.20	1.00
II	0.35	1.33
III	0.62	1.80
IV	1.35	1.80

T' : Soil type dependent parameter
n : number of levels parameter dependent by the type of soil

Soil Parameter (n)

Enter the soil parameter *n*, as defined in Table 5.4 Value of type of soil dependent parameters, NCh2369 page 40:

Type of Soil	**T' (s)**	**n**
I	0.20	1.00
II	0.35	1.33
III	0.62	1.80
IV	1.35	1.80

T' : Soil type dependent parameter
n : number of levels parameter dependent by the type of soil

Damping Ratio (Xi)

Enter the damping ratio ξ, as defined in Table 5.5 Damping Ratios, NCh2369 page 40:

Resistant System	**ξ**
Welded steel shell; silos; hoppers; pressure vessels; process towers; piping, etc	**0.02**
Bolted or riveted steel shell;	**0.03**
Welded steel frames with or without bracings	**0.02**
Steel frames with field bolted connections, with or without bracings	**0.03**
Reinforced concrete and masonry structures	**0.05**
Precast reinforced concrete, purely gravitational structures	**0.05**
Precast reinforced concrete structures with wet connections, connected to the non-structural elements and incorporated into the structural model	**0.05**
Precast reinforced concrete structures with wet connections, non-connected to the non- structural elements	**0.03**
Precast reinforced concrete structures with dry connections, non-connected and connected: With bolted connections and connections by means of bars embedded in filling mortar With welded connections	**0.03** **0.02**
Other structures not included in above list nor similar to the foregoing ones	**0.02**

NOTE See notes at the bottom of Table 5.5 in NCh2369, page 40.

Response Mod. Factor (R)

Enter the response modification factor R, as defined in Table 5.6, NCh2369 page 41:

Resistant System		R
1.	Structures designed for remain elastic	**1**
2.	Other Structures not included nor similar to those in this list[1].	**2**
7.	Tanks, vessels, stacks, silos and hoppers	
7.1	Stacks, silos and hoppers with continuous down-to-floor shells	**3**
7.2	Silos, hoppers and tanks supported on columns, with or without bracing between columns.	**4**
7.3	Vertical axis steel tanks with continuous down-to-floor shell	**4**
7.4	Vertical axis reinforced concrete tanks with continuous down-to-floor shell	**3**
7.5	Tanks and conduits of composite synthetic material (FRP, GFRP, HDPE and similar materials)	**3**
7.6	Horizontal vessels supported on cradles with ductile anchorages	**4**
8.	Towers, piping and equipment	
8.1	Process towers	**3**
8.2	Cooling towers made of wood or plastic	**4**
8.3	Electric control cabinets resting on floor	**3**
8.4	Steel piping except their connections	**5**
9.	Storage racks	**4**
1. Except that a study proves that an R value other than 2 can be used. Structures whose resistant system is explicitly included in this table are not assimilable to this classification.		

NOTE For more information, see Table 5.5 in NCh2369, page 41.

Coefficient of Importance (I)

Enter the coefficient of importance I, as defined in Section 4.3.1 Classification, NCh2369 page 20:

Category	Description	Coefficient of Importance
C1	Critical structures and equipment	**1.20**

	based on if its vital, dangerous and essential (such as pressure vessels, tanks, silos).	
C2	Normal structures and equipment, which may be affected by normal easily repairable failures and do not hazard other category C1 structures.	**1.00**
C3	Structures include buildings that are essential to post-earthquake recovery or associated with hazardous facilities.	**0.80**

Chile NCh2369 En Español

Chile NCh2369 Seismic Data (page 451)

Seccion 4.3.1 Clasificación (NCh2369 pagina 20)

La clasificación apropiada según su importancia es así:

Categoria C1: Estructuras críticas y los equipos utilizados se basaron si son vitales, peligrosos y esenciales. (Recipientes de presión, tanques, silos, etc)

Categoria C2: Las estructuras normales y el equipo utilizado, que pueden ser afectados por los fracasos normalmente y fácilmente reparables y que no arriesguen otras estructuras de categoría C1.

Categoria C3: Estructuras menores o provisionales y el equipo utilizado, cuyo fracaso sísmico no crea peligro a otra estructura de categoría C1 y C2.

Nota: Lea NCh2369 4.3.1 Categorias de Clasificacion para mas informacion

Seccion 4.3.2 Coeficiente de Importancia (NCh2369 pagina 20)

El coeficiente de importancia I para cada categoría tiene los valores siguientes:

C1I : 1.20
C2I : 1.00
C3I : 0.80

Seccion 5. Tablas (NCh2369 pagina 35)

Tabla 5.2 Máxima Aceleración Effectiva (NCh2369 pagina 39)

A_o : aceleración máxima efectiva de suelo

Zona Seismica	A_o
1	0.20 g

2	0.30 g

Tabla 5.4 Parámetros Dependientes del valor de tipo de suelo (NCh2369 pagina 40)

Escoja parámetros dependientes del tipo de suelo en la Tabla 5.3 (NCh2369 pagina 39)

T' : parametro dependiente por el tipo de suelo
n : numero de niveles de parametro dependiente por el tipo de suelo

Tipo de Suelo	T' (s)	n
I	0.20	1.00
II	0.35	1.33
III	0.62	1.80
IV	1.35	1.80

Tabla 5.5 Proporcion de Amortiguamiento (NCh2369 pagina 40)

Sistema Resistente	ξ
Recipiente soldado de acero; silos; depósitos de alimentación; recipientes de presión; torres de proceso; tubería, etc	0.02
Recipiente empernado o remachado de acero;	0.03
Recipientes soldados de acero con o sin refuerzos	0.02
Recipientes de acero con conexiones empernadas, con o sin refuerzos	0.03
Cemento reforzado y estructuras de albañilería	0.05
Cemento reforzado prefabricado, estructuras de gravitación puras.	0.05
Estructuras reforzadas de cemento con conexiones prefabricadas, conectadas a elementos no estructurales e integrados en el modelo estructural	0.05
Estructuras reforzadas de cemento con conexiones prefabricadas, no conectadas a estructuras no elementales	0.03
Estructuras reforzadas de cemento con conexiones secas prefabricadas, no conectadas y conectadas: con conexiones empedernadas y conexiones por medio de barras empotradas en mortero con conexiones soldadas	0.03 0.02
Otras estructuras no incluidas en la lista previa o que asimile a las mencionadas	0.02

Nota: Lea Tabla 5.5 notas finales en NCh2369 pagina 40

Tabla 5.6 Valores máximos del factor de modificación de respuesta (NCh2369 pagina 41)

Sistema Resistente		R
1.	Estructuras diseñadas que se quedan elásticas	1
2.	Otras estructuras no incluidas ni semejantes a ésas en esta lista[1].	2
7.	Tanques, recipientes, pilas, silos y depositos de alimentacion	
7.1	Pilas, silos y depositos de alimentacion con continuos recipientes de bajo del piso	3
7.2	Silos, depositos de alimentacion y tankques soportados en columnas, con o sin refuerzo entre columnas.	4
7.3	Tanques de acero con eje vertical con continuo recipiente bajo del piso	4
7.4	Tanques de concreto con eje vertical con continuo recipiente bajo del piso	3
7.5	Tanques y conductos de materia sintética compuesta (FRP, GFRP, HDPE y materiales similares)	3
7.6	Recipientes horizontales apoyados en cunas con anclajes dúctiles	4
8.	Torres, tuberias y equipo	
8.1	Torres de Proceso	3
8.2	Torres de refrigeración de madera o plástico	4
8.3	Gabinetes eléctricos de control que descansan en el piso	3
8.4	Tubería de acero menos sus conexiones	5
9.	Sporte de almacenamiento	4
1. Amenos que un estudio demuestre que el valor de R valga otro que 2 pueden ser utilizados. Estructuras cuyos sistemas resistentes sean incluidos explicitamente en esta tabla no son asimilables a esta clasificacion.		

Nota: Para mas informacion lea Tabla 5.5 en NCh2369 pagina 41

Tabla 5.7 Valores máximos del coeficiente sísmico (NCh2369 pagina 43)

R	Cmax		
	ξ = 0.02	**ξ = 0.03**	**ξ = 0.05**
1	0.79	0.68	0.55

2	0.60	0.49	0.42
3	0.40	0.34	0.28
4	0.32	0.27	0.22
5	0.26	0.23	0.18

Nota: Estos valores son validos para la zona sismica 3. Para aplicacion de zona 2 y 1, estos valores seran multiplicados por 0.75 y 0.50, respectivamente.

Seccion 5. Figuras 5.1 (NCh2369 pagina 35)

Figura 5.1 a) Zonificacion Sismica de las Regiones I, II, y III (NCh2369 pagina 44)

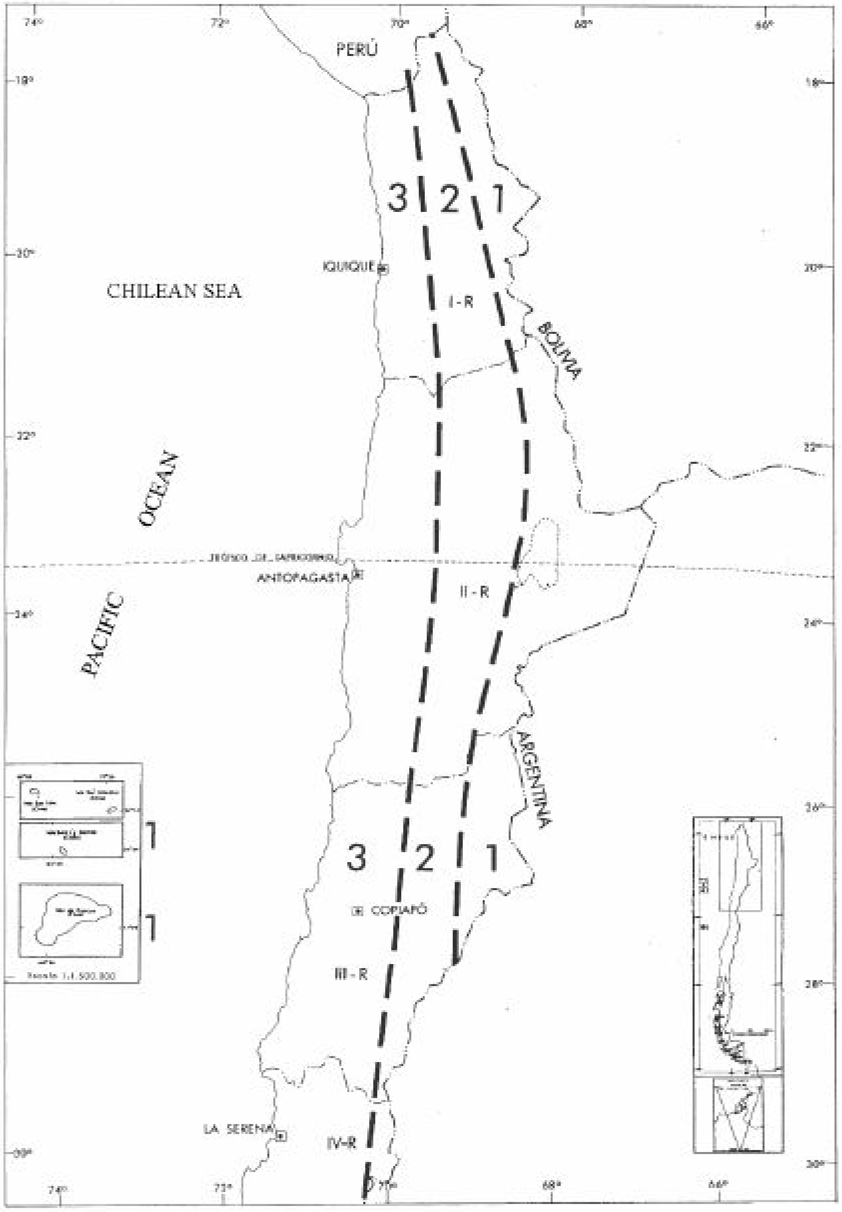


Figura 5.1 b) Zonificacion Sismica de las Regiones IV, V, VI, VII, VIII, IX, X y Region Metropolitana (NCh2369 pagina 45)

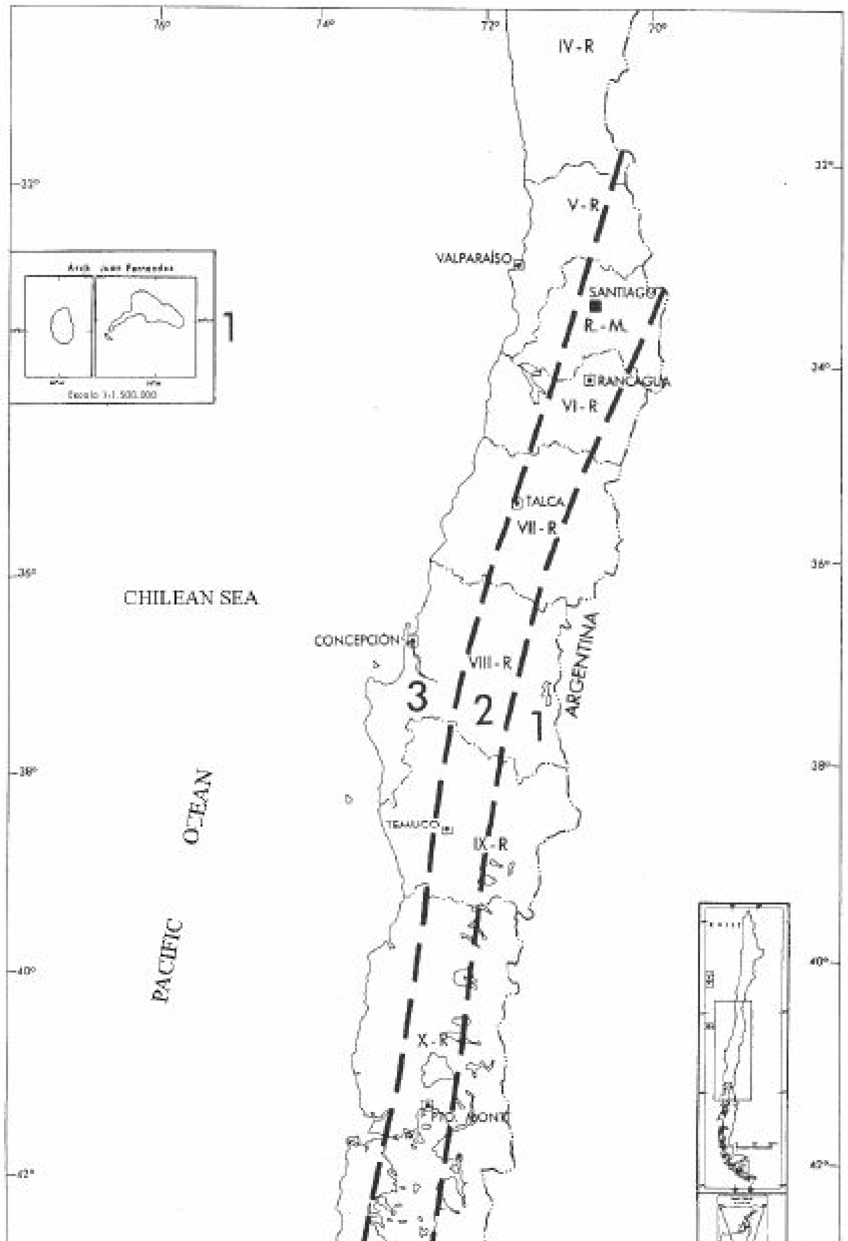
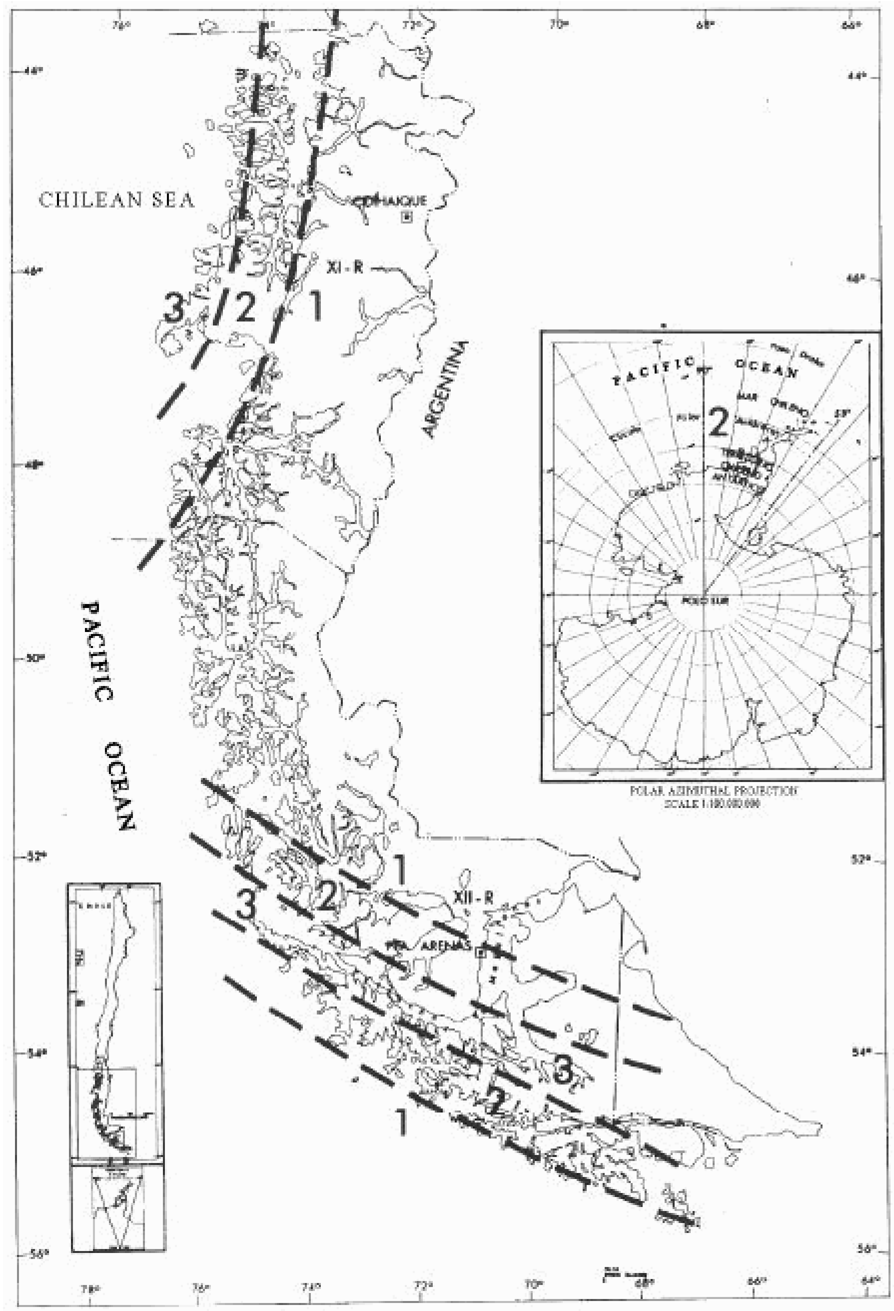


Figura 5.1 c) Zonificacion Sismica de las Regiones XI y XII (NCh2369 pagina 46)



China GB 50011 Seismic Data

Enter data for China's GB 50011-2001 seismic code.

Alpha1 from 5.1.4

Enter a value for the seismic coefficient, α. This value is used to determine F_{Ek} from equation 5.2.1-1. For multistory brick buildings, multistory structures with interior frames, or framed single-story buildings, $\alpha = \alpha_{max}$, the maximum seismic coefficient, as given in Table 5.1.4-1:

Intensity	**VI**	**VII**	**VIII**	**IX**
α_{max}	0.04	0.08 (0.12)	0.16 (0.24)	0.32

DeltaN from 5.2.1

Enter a value for the additional seismic coefficient, δ_n. This value is used in conjunction F_{Ek} to compute ΔF_n.This is an optional entry for additional seismic action at the top level of the building, as given in Table 5.2.1:

T_g (s)	**$T_1 > 1.4*T_g$**	**$T_1 <= 1.4*T_g$**
<= 0.35	$0.08*T_1 + 0.07$	
> 0.35~0.55	$0.08*T_1 + 0.01$	0.0
> 0.55	$0.08*T_1 - 0.02$	

In addition to calculating lateral loads on elements, the software calculates vertical loads according to section 5.3.1.

Costa Rica 2002 Seismic Data

The Costa Rica 2002 seismic design code has these options.

Importance Factor [I]

Enter the importance factor in accordance with Table 4.1.

Group	**Description**	**Factor**
A	Buildings and essential facilities such as hospitals, police and fire stations, airport terminals, and emergency generator facilities.	1.5
B	Buildings and hazardous facilities such as buildings containing toxic chemicals or explosives. Buildings whose failure could put other A or B group buildings in danger.	1.5

C	Buildings for educational activities with more than 300 students. Buildings for adult education with more than 500 students. Health centers with 50 or more disabled residents, but not included in Group A. All constructions with occupancy greater than 5000 people not included in Groups A or B. Buildings and equipment in power generation stations, and other public facilities not included in Group A and required to maintain continuous operation.	1.0
D	All work room, office, retail or industry and other non-residential building not specified in Groups A, B, C and E.	1.0
E	Agricultural construction and low occupancy buildings. Sheds and storage buildings of that do not contain toxic material or have low occupancy. Boundary walls and retaining walls that do not represent high risk to bystanders. Temporary facilities for construction.	0.75

Over resistance factor [SR]

Enter the over resistance factor as defined in Chapter 3, paragraph 3 (d). When using static or dynamic analysis methods of Articles 7.4 and 7.5, the over resistance is equal to 2.0 for frame-like structures and dual wall, and equal to 1.2 for cantilever and other structure types. When using Article 7.7 alternative analysis methods, the over resistance is 1.2 on the ultimate capacity calculated in the analysis for all structural systems.

Soil Type

Select the soil type that matches the local soil conditions:

- S1 - A rock-like material, characterized by a velocity shear wave exceeding 760 m/s or other means proper classification. Hard soil conditions or dense, where the depth of soil is less than 50 m.
- S2 - Medium-dense to dense or medium-stiff to stiff whose depth exceeds 50 m.
- S3 - More than 6 m consistency of soft clay to medium stiff cohesive soils or low or medium density. It includes profiles of more than 12 m of soft clay.
- S4 - Soil characterized by a shear wave velocity less than 150 m/s over 12 m of soft clay.

When the properties of site are not known, use S3. Use S4 if the engineer responsible believes it is needed.

Seismic Zone

Select the seismic zone you need, which are shown in Figure 2.1 in the code.

FIGURA 2.1. Zonificación sísmica

Global Ductility [mu]

The overall ductility μ assigned to each of the structural types in Article 4.2, is defined in Table 4.3 according to the structural system classifications section 4.3, and the local ductility of components and joints according to paragraph 4.4.1.

NOTES

- When the system contains structural elements and precast concrete components that are part of seismic resistant systems, overall ductility is 1.5 unless the engineer responsible for the design justifies a higher value by supporting experimental tests and analytical calculations.
- Buildings with severe irregularities, according to paragraph 4.3.4, have overall ductility of 1.0, unless the engineer responsible for the design justifies a higher value. In any case the overall ductility can be assigned greater than the corresponding structures moderate irregularity.
- Structural systems consisting of walls, frames, and frames braced by mezzanines that behave as rigid diaphragms, as subsection (e) of Chapter 3, cannot have a global ductility greater than 1.5.
- For cantilever structures, use a global ductility of 1.0 on the foundation design.
- For steel structures and OCBF and OMF types as defined in the Chapter 10, you must use a global ductility of 1.5.

Peak acceleration [aef]

Enter the design effective peak acceleration expressed as a fraction of gravity. The value is based on the seismic zoning and soil type from Table 2.2.

Soil Type	Zone II	Zone III	Zone IV
S_1	0.20	0.30	0.40
S_2	0.24	0.33	0.40
S_3	0.28	0.36	0.44
S_4	0.34	0.36	0.36

Spectral Dynamic Factor [FED]

Enter the degree-of-freedom change in acceleration a system suffers with respect to the ground acceleration. The area is a function of seismic soil type of the global ductility μ and the period assigned. This value is shown in Figure 5.1 through Figure 5.12 in the code for each seismic zone and soil type.

EN 1998-1:2004

Enter data for the EN 1998-1:2004 seismic code.

Ground Type

Select a ground type according to EN 1998-1, Table 3.1:

Ground Type	Description of Stratigraphic Profile	Parameters		
		v_s30 (m/s)	N_{SPT} (blows/30cm)	c_u (kPa)
A	Rock or other rock-like geological formation, including at most 5m of weaker material at the surface.	> 800	-	-
B	Deposits of very dense sand, gravel, or very stiff clay, at least several tens of meters in thickness, characterized by a gradual increase of mechanical properties with depth.	360 - 800	> 50	> 250
C	Deep deposits of dense or medium-dense sand, gravel or stiff clay with thickness from several tens to many hundreds of meters.	180 - 360	15 - 50	70 - 250

D	Deposits of loose-to-medium cohesionless soil (with or without some soft cohesive layers), or of predominantly soft-to-firm cohesive soil.	< 180	< 15	< 70
E	A soil profile consisting of a surface alluvium layer with vs values of type C or D and thickness varying between about 5m and 20m, underlain by stiffer material with vs > 800 m/s			

The software uses the **Ground Type** selection to automatically determine the **Lower Limit Period Acceleration Value [Tb]**, **Upper Limit Period Acceleration Value [Tc]**, **Beginning Displacement Range Value [Td]**, and **Soil Factor [S]**, although you can override these values and enter your own.

NOTE The ground classification scheme accounting for deep geology, including S, Tb, Tc, and Td values defining the horizontal and vertical elastic response spectra in accordance with 3.2.2.2 and 3.2.2.3., are located in the National Annex for the corresponding country.

Lower Limit Period Acc. Value [Tb]

Enter the lower limit period acceleration value *Tb*, as defined in EN 1998-1, Table 3.2.

Type 1 Elastic Response Spectra

Ground Type	*S*	*Tb(s)*	*Tc(s)*	*Td(s)*
A	1.0	0.15	0.4	2.0
B	1.2	0.15	0.5	2.0
C	1.15	0.20	0.6	2.0
D	1.35	0.20	0.8	2.0
E	1.4	0.15	0.5	2.0

NOTE This software automatically defines this value based on the selection made in **Ground Type**; however, you can override that value and enter your own.

Upper Limit Period Acc. Value [Tc]

Enter the upper limit period acceleration value *Tc*, as defined in EN 1998-1, Table 3.2.

Type 1 Elastic Response Spectra

Ground Type	*S*	*Tb(s)*	*Tc(s)*	*Td(s)*

A	1.0	0.15	0.4	2.0
B	1.2	0.15	0.5	2.0
C	1.15	0.20	0.6	2.0
D	1.35	0.20	0.8	2.0
E	1.4	0.15	0.5	2.0

NOTE This software automatically defines this value based on the selection made in **Ground Type**; however, you can override that value and enter your own.

Beginning Displacement Range Value [Td]

Enter the beginning displacement range value *Td*, as defined in EN 1998-1, Table 3.2.

Type 1 Elastic Response Spectra

Ground Type	***S***	***Tb(s)***	***Tc(s)***	***Td(s)***
A	1.0	0.15	0.4	2.0
B	1.2	0.15	0.5	2.0
C	1.15	0.20	0.6	2.0
D	1.35	0.20	0.8	2.0
E	1.4	0.15	0.5	2.0

NOTE This software automatically defines this value based on the selection made in **Ground Type**; however, you can override that value and enter your own.

Soil Factor [S]

Enter the soil factor *S*, as defined in EN 1998-1, Table 3.2.

Type 1 Elastic Response Spectra

Ground Type	***S***	***Tb(s)***	***Tc(s)***	***Td(s)***
A	1.0	0.15	0.4	2.0
B	1.2	0.15	0.5	2.0
C	1.15	0.20	0.6	2.0
D	1.35	0.20	0.8	2.0
E	1.4	0.15	0.5	2.0

NOTE This software automatically defines this value based on the selection made in **Ground Type**; however, you can override that value and enter your own.

Design Ground Acceleration [ag]

Enter the reference horizontal peak ground acceleration value. The default value is **0.2**.

NOTE You can derive the reference peak ground acceleration for a **Ground Type** of **A**, agR, for use in a country or parts of the country, from zonation maps in its National Annex.

Behavior Factor [q]

Enter the behavior factor. The factor is used for design purposes to reduce the forces obtained from a linear analysis to account for the non-linear response of a structure associated with the material the structural system and the design procedures. This value ranges from **1.0** to **1.5**.

% of Horizontal Load Applied to Vertical

Enter the percentage of the horizontal load that is applied to the vertical direction. The default value is **30,** with a maximum value of **100**.

G Loading Seismic Data

Enter data for seismic loads in G's

Importance Factor

Enter the importance factor. The value is usually between **1.0** and **1.5**. Loads generated form G loading are multiplied directly by this value.

Long. Acceleration (Gx), **Lateral Acceleration (Gz)**, and **Vertical Acceleration (Gy)**

Enter in the value of *G* in each direction for the vessel. Typical values of lateral G loads are from **0** to **0.4**. Vertical G loads can be positive or negative. Negative G loads increase the saddle loads and vertical G loads decrease the saddle loads.

For vertical vessels, the horizontal component used is the maximum of the *Gx* and *Gz* values. The calculated horizontal force is equal to the weight of the element times this maximum G factor. This force times its distance to the support is calculated and summed with all other forces. The *Gy* value is also considered. This value is usually 2/3 of the *Gx* or *Gz* value. Any of these values can be zero.

For horizontal vessels, the lateral (*Gz*) and longitudinal (Gx) directions are considered independently. The vertical load component (*Gy*) acting on the saddle supports is also calculated.

IBC 2000 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	**F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:**				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	**F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:**				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**

C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 **-** 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Factor tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

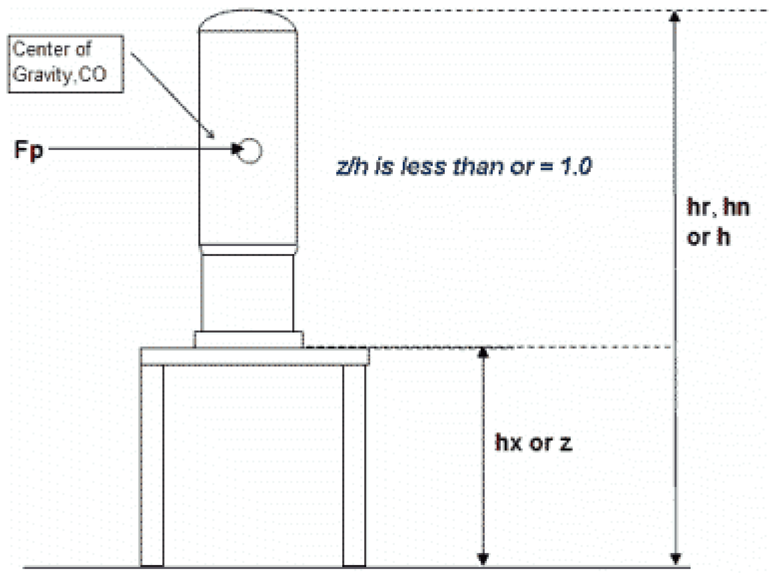
Use ASCE-7 2005 Code?

Select to use ASCE-7 2005 as the code for analysis. This option is only available when

ASCE 7-02/05 is selected for **Seismic Design Code.**

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IBC 2003 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
	S_s <= 0.25	S_s = 0.5	S_s = 0.75	S_s = 1.0	S_s >= 1.25[2]
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.2	1.2	1.1	1.0	1.0
D	1.6	1.4	1.2	1.1	1.0
E	2.5	1.7	1.2	0.9	[0.9][1]
F	See note 1				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
	S_l <= 0.1	S_l = 0.2	S_l = 0.3	S_l = 0.4	S_l >= 0.5[2]
A	0.8	0.8	0.8	0.8	0.8
B	1.0	1.0	1.0	1.0	1.0
C	1.7	1.6	1.5	1.4	1.3
D	2.4	2.0	1.8	1.6	1.5
E	3.5	3.2	2.8	2.4	[2.4][1]
F	See note 1				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Factor tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

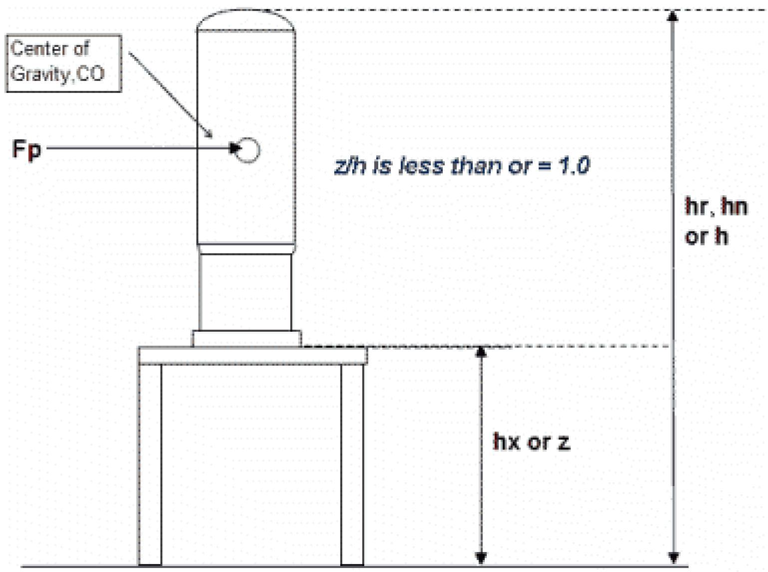
Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code?

Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IBC 2006 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. SI** and **Site Class:**

Site Class	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Factor tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

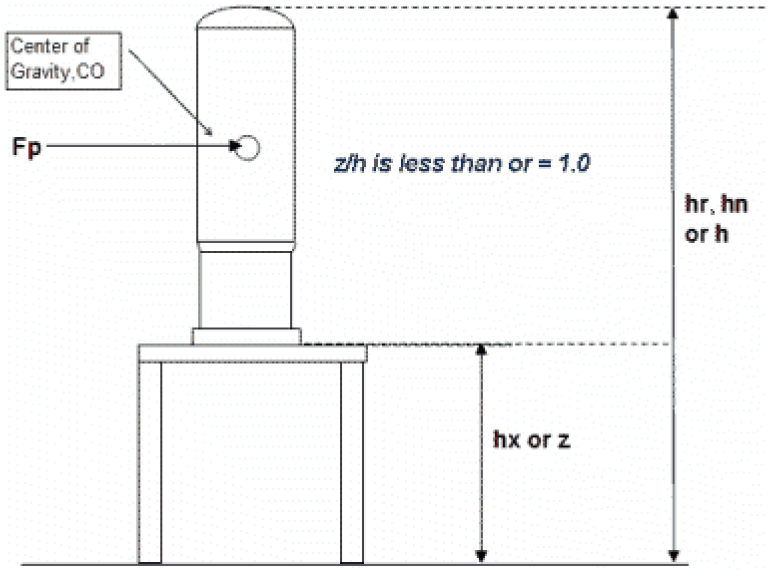
Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Use ASCE-7 2005 Code?

Select to use ASCE-7 2005 as the code for analysis. This option is only available when **ASCE 7-02/05** is selected for **Seismic Design Code.**

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

IBC 2009 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Factor tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

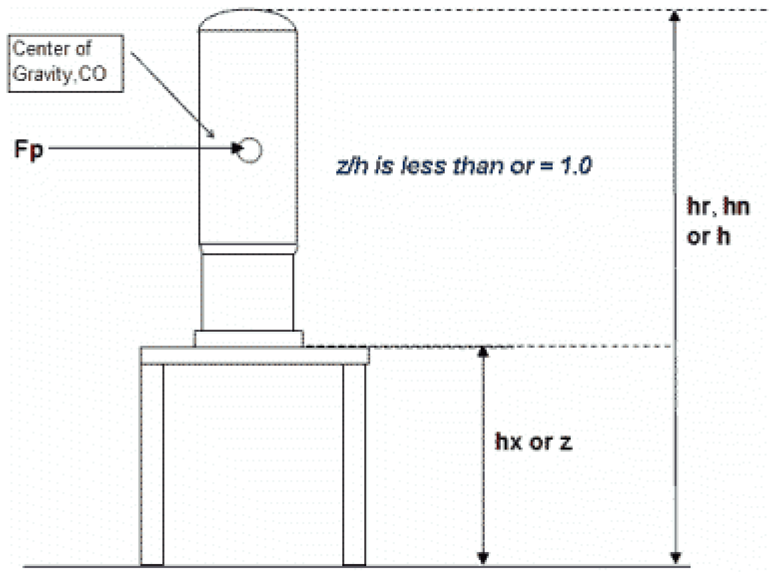
NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

Consider Vertical Accelerations

Select this option to consider the vertical component of the earthquake. Some earthquake design codes provide methods for determining the vertical component of acceleration due to an earthquake event. The vertical component of the earthquake load will cause axial stress on the elements and loads on the supporting elements as well as lugs and legs.

Force Factor

Overrides the computed value of the vertical earthquake load. The force is simply equal to the force factor times the base shear force, V. This is one possible candidate of the vertical loads.

Minimum Acceleration Multiplier

In ASCE-7 98, the vertical load is 0.2 * SDS * Weight of the Vessel. If you enter a minimum Acceleration multiplier value in this box, an additional candidate for the vertical is computed. This value is simply the Minimum Acceleration Multiplier times the weight.

Sds

As defined in ASCE-7 98, S_{DS} is "design, 5% damped, spectral response acceleration at short periods as defined in Section 9.4.1.2;"

Sd1

As defined in ASCE-7 98, S_{D1} is "design, 5% damped, spectral response acceleration at a period of 1 second as defined in Section 9.4.1.2;"

IBC 2012 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based Factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
Site Class	**S_s <= 0.25**	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9][1]**
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based Factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or

Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	**F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:**				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. S1

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Value tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

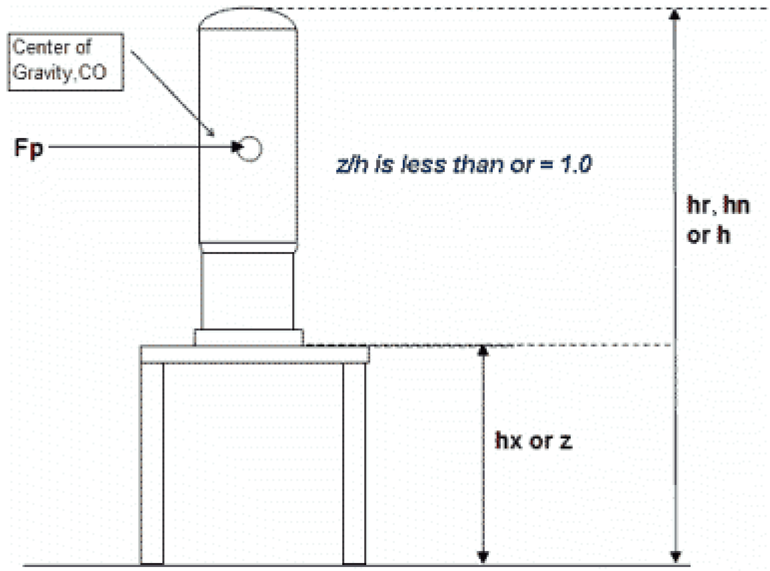
NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

Consider Vertical Accelerations

Select this option to consider the vertical component of the earthquake. Some earthquake design codes provide methods for determining the vertical component of acceleration due to an earthquake event. The vertical component of the earthquake load will cause axial stress on the elements and loads on the supporting elements as well as lugs and legs.

Force Factor

Overrides the computed value of the vertical earthquake load. The force is simply equal to the force factor times the base shear force, V. This is one possible candidate of the vertical loads.

Minimum Acceleration Multiplier

In ASCE-7 98, the vertical load is 0.2 * SDS * Weight of the Vessel. If you enter a minimum Acceleration multiplier value in this box, an additional candidate for the vertical is computed. This value is simply the Minimum Acceleration Multiplier times the weight.

Sds

As defined in ASCE-7 98, S_{DS} is "design, 5% damped, spectral response acceleration at short periods as defined in Section 9.4.1.2;"

Sd1

As defined in ASCE-7 98, S_{D1} is "design, 5% damped, spectral response acceleration at a period of 1 second as defined in Section 9.4.1.2;"

IBC 2015 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based Factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
Site Class	**S_s <= 0.25**	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9][1]**
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.

- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based Factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
Site Class	**S_l <= 0.1**	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4][1]**
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max Mapped Res. Acc. S1

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Value tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

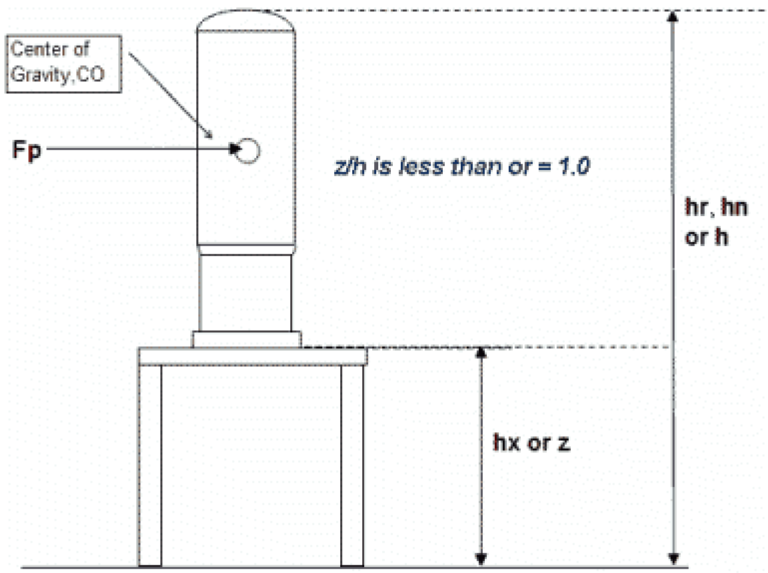
NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

Consider Vertical Accelerations

Select this option to consider the vertical component of the earthquake. Some earthquake design codes provide methods for determining the vertical component of acceleration due to an earthquake event. The vertical component of the earthquake load will cause axial stress on the elements and loads on the supporting elements as well as lugs and legs.

Force Factor

Overrides the computed value of the vertical earthquake load. The force is simply equal to the force factor times the base shear force, V. This is one possible candidate of the vertical loads.

Minimum Acceleration Multiplier

In ASCE-7 98, the vertical load is 0.2 * SDS * Weight of the Vessel. If you enter a minimum Acceleration multiplier value in this box, an additional candidate for the vertical is computed. This value is simply the Minimum Acceleration Multiplier times the weight.

Sds

As defined in ASCE-7 98, S_{DS} is "design, 5% damped, spectral response acceleration at short periods as defined in Section 9.4.1.2;"

Sd1

As defined in ASCE-7 98, S_{D1} is "design, 5% damped, spectral response acceleration at a period of 1 second as defined in Section 9.4.1.2;"

IBC 2018 Seismic Data

Importance Factor I

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property.

Response Factor R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Acc. Based Factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

Site Class	**F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:**				
	S_s <= 0.25	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**

E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based Factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

Site Class	**F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:**				
	S_l <= 0.1	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5**[2]
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_S, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max Mapped Res. Acc. S1

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Moment Reduction Value tau

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one will scale the moments up, while a value that is less than one will lower the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

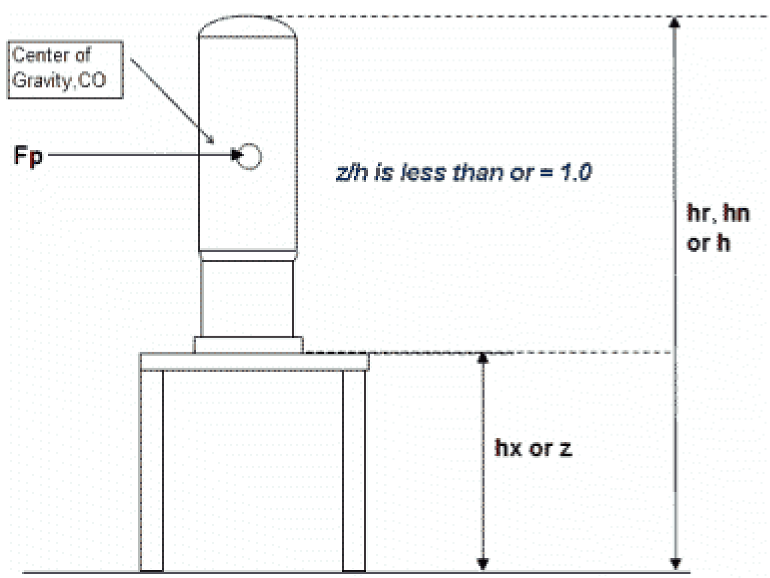
NOTE For ASCE and IBC codes after 2000, this value is no longer used and is therefore not applicable.

Site Class

Select the site class: **A**, **B**, **C**, **D**, **E**, or **F**. The software only uses these values to check the minimum value of *C* according to equation 9.5.3.2.1-4 (ASCE), 1615.1.1 (IBC). This additional check is only performed if **E** or **F** is selected.

Component Elevation ratio z/h

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

Consider Vertical Accelerations

Select this option to consider the vertical component of the earthquake. Some earthquake design codes provide methods for determining the vertical component of acceleration due to an earthquake event. The vertical component of the earthquake load will cause axial stress on the elements and loads on the supporting elements as well as lugs and legs.

Force Factor

Overrides the computed value of the vertical earthquake load. The force is simply equal to the force factor times the base shear force, V. This is one possible candidate of the vertical loads.

Minimum Acceleration Multiplier

In ASCE-7 98, the vertical load is 0.2 * SDS * Weight of the Vessel. If you enter a minimum Acceleration multiplier value in this box, an additional candidate for the vertical is computed. This value is simply the Minimum Acceleration Multiplier times the weight.

Sds

As defined in ASCE-7 98, S_{DS} is "design, 5% damped, spectral response acceleration at short periods as defined in Section 9.4.1.2;"

Sd1

As defined in ASCE-7 98, S_{D1} is "design, 5% damped, spectral response acceleration at a period of 1 second as defined in Section 9.4.1.2;"

IS-1893 RSM Seismic Data

Enter data for India's Earthquake Standard IS-1893 RSM.

IS1893 Code Edition

Select the edition of the code to use: **1984**, **2002**, or **2005 Simplified**. The Seismic Coefficient Method has been abandoned in the 2002 edition.

Importance Factor

Enter the importance factor *I,* according to Table 6, IS: 1893, 2002 Seismic Design Code. The following values are for guidance. You may chose an appropriate factor based on economy, strategy and other considerations. You may choose values greater than those listed here.

Structure	Importance Factor
Important service and community buildings such as hospitals, schools, monumental structures, emergency buildings like telephone exchange, television stations, radio stations, railway stations, fire station buildings; large assembly halls like cinemas, assembly halls, subway stations, power	**1.5**

stations.	
All other buildings	**1**

The older version of the IS 1893 also listed the following additional categories:

Structure	**Importance Factor**
Containment structures for atomic power reactors.	**6**
Dams of all types	**3**
Containers of inflammable or poisonous gases/liquids	**2**

Soil Factor

Enter the soil factor (Beta), according to Table 3, IS:1893 seismic design code. Values of the soil factor generally range between 1.0 and 1.5.

Soil Type	**Soil Factor**
Type I rock or hard soils	**1**
Type II medium soils	**1**
Type II medium soils - well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.2**
Type III soft soils	**Between 1.0 and 1.5**
Well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.5**
Combined or isolated RCC footings with tiebeams	**1.2**
Piles not on raft foundations	**1.2**
Other type III soft soils	**1.0**

Zone Number

Enter the zone number. In the 1984 edition of the standard, India is divided into five zones, from **1** to **5**. In the 2002 edition, the number of zones is reduced to four. See the Figure 1 in IS:1893 to obtain the zone number from the map.

Period of Vibration (Optional)

Enter the period of vibration. This is the first period of vibration of the vessel in the normal operating case. This value is used in conjunction with damping to determine *Sa/g*. The software calculates this value, but entering a value here overrides the calculated value. This entry is optional.

Percent Damping

Enter the damping factor of the vessel as a percentage of critical damping. The effect of internal friction, imperfect elasticity of material, slipping, and sliding, results in reducing the amplitude of vibration. This value is used to determine the average acceleration coefficient *Sa/g*.

In the 1984 edition of the IS:1893, Figure 2 shows various curves for damping and natural period of vibration versus Sa/g. Values for damping on this graph range between 2 and 20 percent.

In the 2002 edition of the IS:1893, Figure 2 shows the Sa/g values for 5% damping. If your damping percentage is different from 5% then Sa/g value from Figure 2 has to be multiplied by damping factor. Table 3 provides the damping factor for damping percentage between 0 to 30 percent.

A value outside of the ranges indicated above should not be entered. If a value outside the range is entered, the software simply uses that extreme value of damping. If the value does not have a direct factor provided (such as 2, 5, 10, 20 percentage, etc.) then linear extrapolation will be used to obtain the intermediate values.

Overriding "Fo" value if > 0

Overrides the **Fo** value calculated by the software. This option defaults to a value of **0**.

Soil Type

Select a soil type: **Rocky or Hard Soils**, **Medium Soils**, or **Soft Soils**.

Force Factor R

Enter the seismic force factor.

Vessel is attached to a building and projects over the roof?

Select if this condition is true.

Optional Sa/g vs. Time Period Input

Click **Review and Edit Spectrum Points** to open the **Spectrum Data Points** dialog box. Enter your values for **Sa/g** at each needed **Time Period (T)**. This table must be completely filled in.

After entering values into the dialog box, select **Do not apply Damping Correction Factor**. The table values are then used in lieu of the internally stored data values of *Sa/g* for various amounts of critical damping. The software does not use any other tables to determine *Sa/g*.

IS-1893 SCM Seismic Data

Enter data for India's Earthquake Standard IS-1893 SCM.

Importance Factor

Enter the importance factor *I,* according to Table 6, IS: 1893, 2002 Seismic Design Code. The following values are for guidance. You may chose an appropriate factor based on economy, strategy and other considerations. You may choose values greater than those listed here.

Structure	**Importance Factor**
Important service and community buildings such as hospitals, schools, monumental structures, emergency buildings like telephone exchange, television stations, radio stations, railway stations, fire station buildings; large assembly halls like cinemas, assembly halls, subway stations, power stations.	**1.5**
All other buildings	**1**

The older version of the IS 1893 also listed the following additional categories:

Structure	**Importance Factor**
Containment structures for atomic power reactors.	**6**
Dams of all types	**3**
Containers of inflammable or poisonous gases/liquids	**2**

Soil Factor

Enter the soil factor (Beta), according to Table 3, IS:1893 seismic design code. Values of the soil factor generally range between 1.0 and 1.5.

Soil Type	**Soil Factor**
Type I rock or hard soils	**1**
Type II medium soils	**1**
Type II medium soils - well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.2**
Type III soft soils	**Between 1.0 and 1.5**
Well foundations or isolated RCC footings without tiebeams or unreinforced strip foundations	**1.5**
Combined or isolated RCC footings with tiebeams	**1.2**
Piles not on raft foundations	**1.2**
Other type III soft soils	**1.0**

Zone Number

Enter the zone number. In the 1984 edition of the standard, India is divided into five zones, from **1** to **5**. In the 2002 edition, the number of zones is reduced to four. See the Figure 1 in IS:1893 to obtain the zone number from the map.

User Alpha H

Enter a value for Alpha H.

KBC 2016 Seismic Data

Enter data for the KBC seismic code.

Importance Factor [I]

Enter a value for the importance factor. The software uses this value directly without modification.

Response Modification Factor [R]

Enter the response modification factor R.

Site Coefficient for Short Periods [Fa]

Enter the appropriate F_a value from the table below.

Site Class		Region of Seismicity		
		S_g <= 0.25	S_g = 0.50	S_g = 0.75
S_A		0.8	0.8	0.8
S_B		1.0	1.0	1.0
S_C	more than 20 m depth to rock	1.2	1.2	1.1
	less than 20 m depth to rock	1.4	1.4	1.3
S_D	more than 20 m depth to rock	1.6	1.4	1.2
	less than 20 m depth to rock	1.7	1.5	1.3
S_E		2.5	1.9	1.3

S_g is 2.5 times the s defined for the design response spectral acceleration (Eq. 0306.3.1). For the values between the specified S_g shown above, linear interpolation shall be used.

Site Coefficient for 1 Second Period [Fv]

Enter the appropriate F_v value from the table below.

Site Class	Region of Seismicity		
	S <= 0.1	S = 0.2	S = 0.3

S_A		0.8	0.8	0.8
S_B		1.0	1.0	1.0
S_C	more than 20 m depth to rock	1.7	1.6	1.5
	less than 20 m depth to rock	1.5	1.4	1.3
S_D	more than 20 m depth to rock	2.4	2.0	1.8
	less than 20 m depth to rock	1.7	1.6	1.5
S_E		3.5	3.2	2.8

S is 2.5 times the s defined for the design response spectral acceleration (Eq. 0306.3.2). For the values between the specified S shown above, linear interpolation shall be used.

Max. Mapped Res. Acc. [Ss]

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the KBC 2016 publication.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Maximum Effective Ground Acceleration [S]

Enter the maximum effective ground acceleration.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value is the table is 25%, enter 0.25.

Moment Reduction Value [tau]

Enter a moment reduction factor, used to reduce the moment at each level. A value greater than one scales the moments up, while a value that is less than one lowers the moments. A value of **1.0** is recommended. The value should not be less than 0.8.

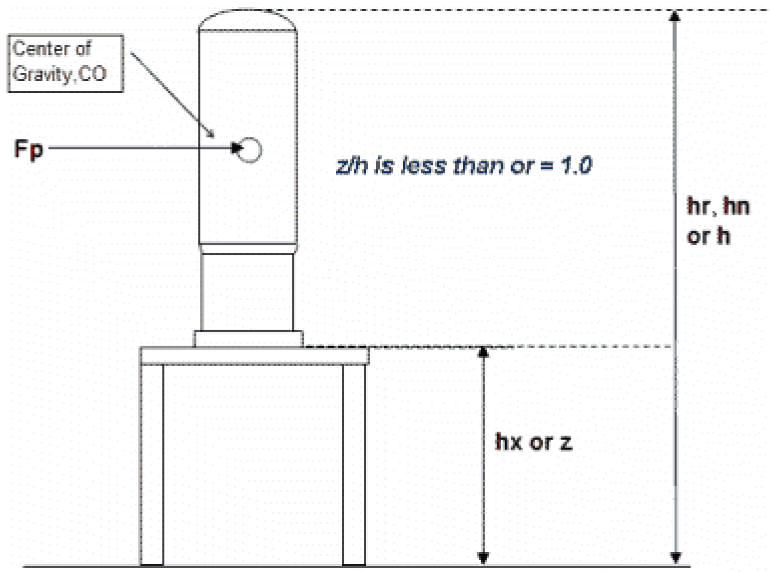
Site Class

Select the site class as defined in the seismic code.

If Building Supported fill in the following values

Component Elevation ratio [z/h]

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**.



Component Amplification Factor [ap]

Enter a value between **1.0** to **2.5**. For vessels, a value of **2.5** is typical.

Optional Input Values

Consider Vertical Accelerations

Select to enable vertical load calculations (YEq).

Force Factor

Overrides the computed value of the vertical earthquake load. The force is equal to the force factor times the base shear force, V. This is one possible candidate of the vertical loads.

Minimum Acceleration Multiplier

Enter the force factor, G_y , applied to the total weight of the vessel.

SDS

Enter the value for S_{DS}, the 5 percent damped, spectral response acceleration parameter at short periods.

SD1

Enter the value for S_{DS}, the 5 percent damped, spectral response acceleration parameter at a period (1 second).

KHK 2012 Seismic Data

Enter data for the KHK seismic code.

Seismic Level

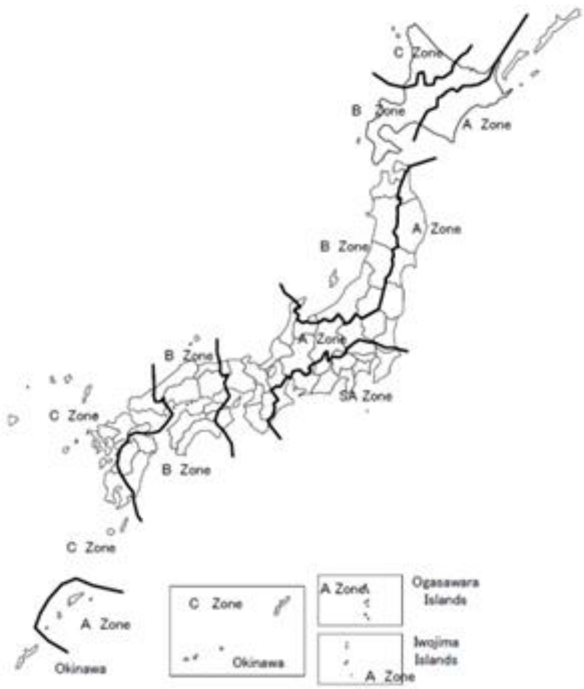
Specifies the earthquake level factor. Select **1** for a level 1 earthquake, or select **2** for a level 2 earthquake. A Level 2 earthquake represents a much more severe event than does level 1. When determining the acceleration values, the initial calculation constant is multiplied by 2. Depending on the choices for zone and soil type, and so forth, some of the results are modified accordingly.

When the level is set to **2**, the final values of the generated loads are multiplied by 0.5 per Article 2.

NOTE There are special rules in KHK for Spherical Pressure Vessels. PV Elite does not use those rules.

Seismic Zone

Specifies the seismic zone, as defined in the following graphic. Select SA, A, B, or C.



Source: Seismic Design Standard for the High Pressure Gas Facilities, High Pressure Gas Safety Institute of Japan (KHK). Article 3. MITI Notice No. 515 dated Oct. 26, 1981. Amended Notice No. 143 dated Mar 25, 1997.

The zone determines the zoning factor (β_2.).

Zone	Level 1 Earthquake β_2	Level 1 Earthquake β_2
SA (special A)	1	1
A	0.8	0.8

B	0.65	0.7
C	0.4	0.7

Soil Profile

Specifies the type of soil. Select 1, 2, 3, or 4.

The soil type determines the soil factor (β_3).

Soil Profile Value	Type of Soil	Soil Factor β_3
1	Before tertiary deposit ground	1.4
2	Diluvial deposit ground	2
3	All types other than 1, 2, and 4	2
4	The thickness of soil fill or alluvial deposit is 25m or more	2

Importance Class

Specifies the importance class as designated by the type of gas, storage capacity (W), and distance of the transmission pipe layout (X). Select Ia, I, II, or III.

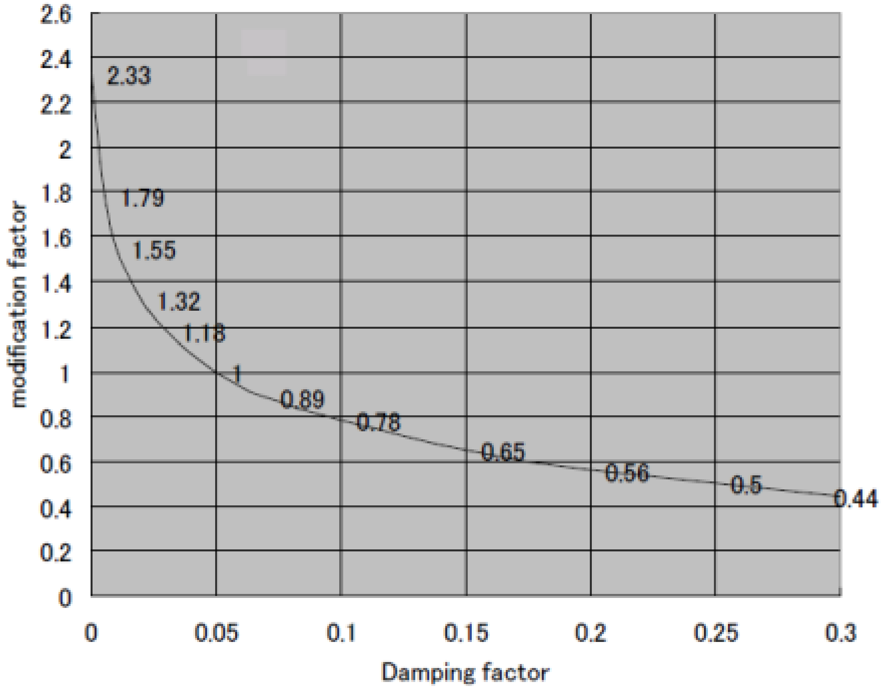
The importance class determines the importance factor (β_1).

Importance Class	Importance Factor β^1
Ia	1
I	0.08
II	0.65
III	0.5

- When $\beta_1 \times \beta_1$ is less than 0.33, the value of β_1 shall be 0.33.
- The importance class is defined by Article 3, Tables (a), (b), and (c) of Seismic Design Standard for the High Pressure Gas Facilities, High Pressure Gas Safety Institute of Japan (KHK). MITI Notice No. 515 dated Oct. 26, 1981. Amended Notice No. 143 dated Mar 25, 1997.

Damping Factor

Specifies the damping factor used to determine the modification (or response compensation) factor. For pressure vessels, a value of 0.03 is common.



Source: Seismic Design Standard for the High Pressure Gas Facilities, High Pressure Gas Safety Institute of Japan (KHK). Article 6, Fig. (c). MITI Notice No. 515 dated Oct. 26, 1981. Amended Notice No. 143 dated Mar 25, 1997.

Natural Period

Specifies the natural period of the seismic structure in seconds.

Vertical Response Magnification Factor

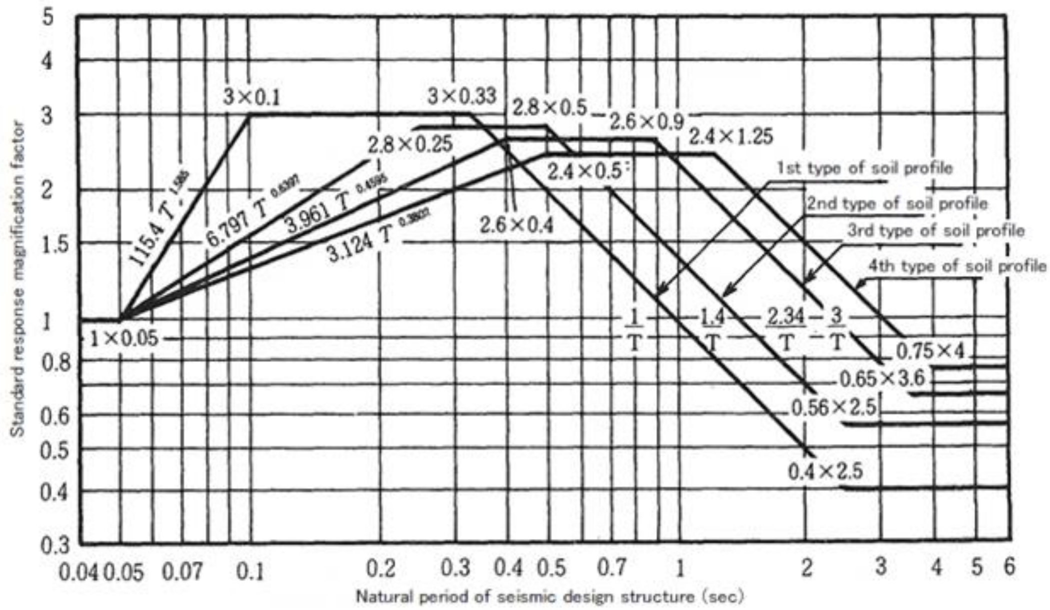
Specifies the vertical response magnification factor (β_6), typically 1.5 for a skirt-supported tower and 2.0 for other structures.

Horizontal Response Magnification Factor

Specifies the horizontal response magnification factor (β_5), the product of the standard response magnification factor and the response compensation factor.

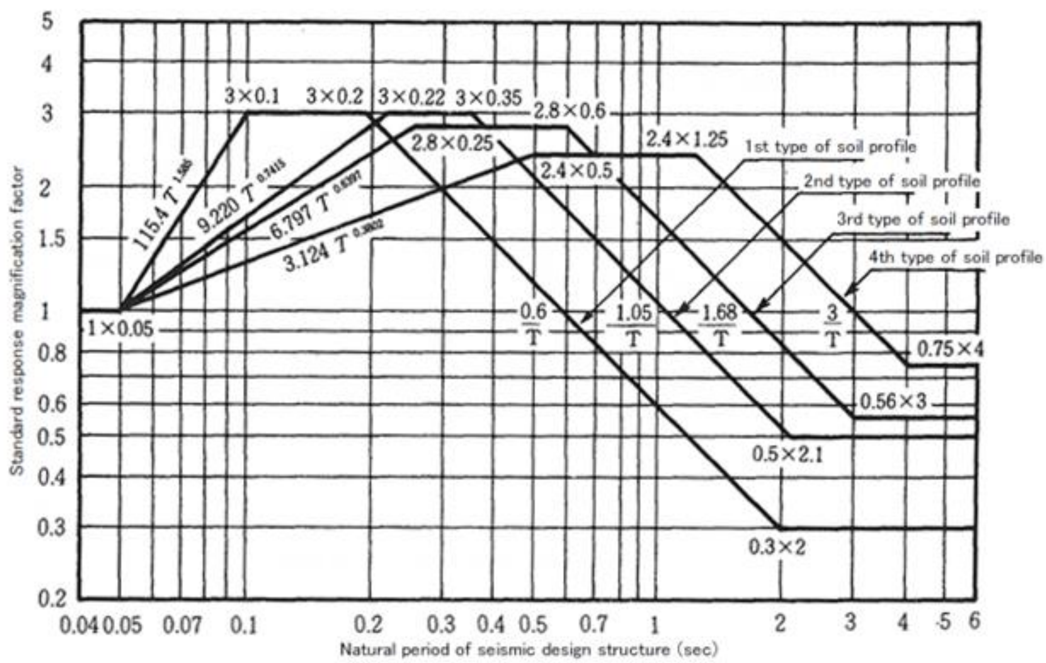
The standard response magnification factor is defined in the following graphics.

For SA and A seismic zones:



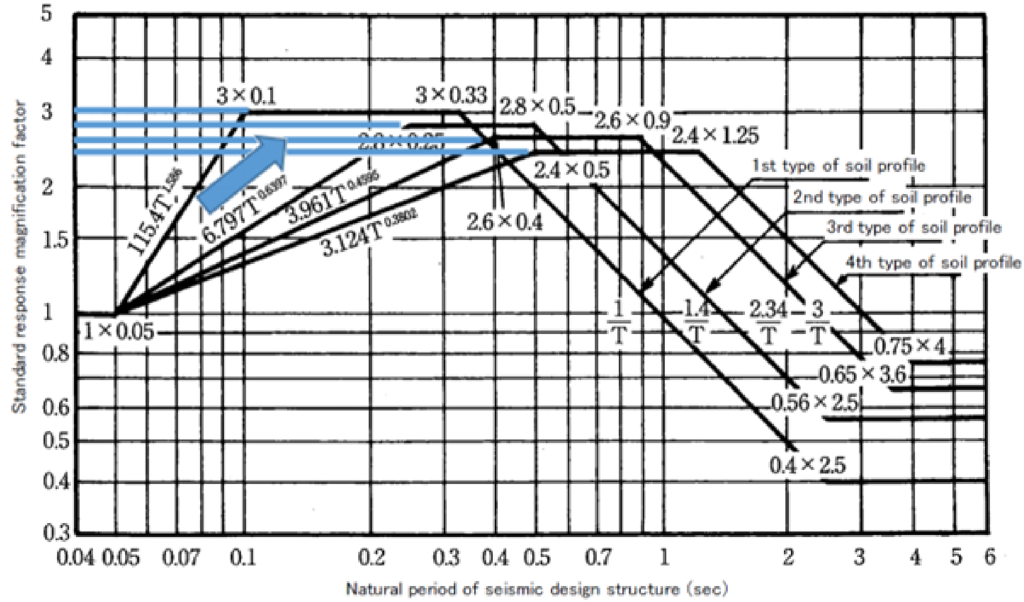
Source: Seismic Design Standard for the High Pressure Gas Facilities, High Pressure Gas Safety Institute of Japan (KHK). Article 6, Fig. (a). MITI Notice No. 515 dated Oct. 26, 1981. Amended Notice No. 143 dated Mar 25, 1997.

For B and C seismic zones:



Source: Seismic Design Standard for the High Pressure Gas Facilities, High Pressure Gas Safety Institute of Japan (KHK). Article 6, Fig. (b). MITI Notice No. 515 dated Oct. 26, 1981. Amended Notice No. 143 dated Mar 25, 1997.

For Seismic Level 2:



Source: Seismic Design Standard for the High Pressure Gas Facilities, High Pressure Gas Safety Institute of Japan (KHK). Fig 7.3-1. MITI Notice No. 515 dated Oct. 26, 1981. Amended Notice No. 143 dated Mar 25, 1997.

NOTE When a plastic response is expected for a strong earthquake, the natural period of the structure might move to a longer period. As a result, the response in a shorter period takes the maximum (peak) value for each response spectrum.

Height from Base to Grade

Type the distance in meters from the bottom of the skirt or bottom of the saddle support to grade level. In most cases, this value is zero. If, however, the vessel is in a steel structure, then you must enter the value.

Use Article 6 (Modified Seismic Coefficient Method) when Possible

In the KHK Standard there are different articles that address different types of analysis possible.

Article 4 Response Analysis

This article discusses using articles 6, 7 and 8. For class II or class III tower vessels that are less than 20 meters off the ground and for horizontal vessels whose storage capacity is less than 100 tons, article 5 can used.

Article 5 Static Seismic Coefficient Method

Article 5 involves the calculation of term β_4 which is a function of the height of the base of the vessel from grade.

H (meters)	β_4
16 or less	2
Between 16 and 35	1.04 + 0.06H

Greater than 35	3.14

Article 6 Modified Seismic Coefficient Method

This method involves computing β_5 and β_6 and using these values to determine the horizontal and vertical acceleration values necessary to ascertain the loads on the vessel during the seismic event. If the Importance Class is II or III and Ht is 20 meters, then you have an option to perform the analysis per article 5 or use article 6. This check box allows you to control the flow of the analysis.

Mexico Sismo Seismic Data

Mexico Sismo En Español (page 502)

Enter data for the Mexican seismic code.

Seismic Zone

Select the seismic zone: **A**, **B**, **C,** or **D**. See the region map on page 1.3.29 of the Manual DE Diseno por Sismo (Seismic Design Manual) for Mexico. Zone D is the zone of highest seismic activity while zone A is the least active.

Structure Group

Select the structure group:

- **A - High Safety** - Structure requiring a high degree of safety during their design, such as towers and tanks.
- **B - Med. Safety** - Structure requiring an intermediate degree of safety during their design.
- **C - Low Safety** - Structure requiring a low degree of safety during their design.

Soil Type

Select the type of soil:

- **I** - Hard soil. Ground deposits formed exclusively by layers with propagation velocity b_0 = 700 m/s or modulus of rigidity >= 85000 t/m^2.
- **II** - Medium soil. Ground deposits with fundamental period of vibration and effective velocity of propagation which meets the condition $\beta_c T_s + \beta_s T_c > \beta_c T_c$.
- **III** - Soft soil. Ground deposits with fundamental period of effective vibration of propagation which meet the condition $\beta_c T_s + \beta_s T_c < \beta_c T_c$.

Behavior Factor Q

Enter the behavior factor *Q*. For chimneys and towers, use **3.0**. For smaller structures like tanks, use **2.0**. For more information, see page 1.3.23 in the Manual Sismo.

Effective Absorption Factor

Enter the absorption factor *zeta e hat,* used to compute the increment factor Xi. This factor should always be less than or equal to 0.02.

Orthogonal Increase Factor

Enter the factor to scale up the earthquake loads in a linear (scalar) fashion. This value is traditionally **1.118** and should always be greater than or equal to 1.0. The Mexican

Earthquake Code considers an SRSS-type effect on the structure.

Specification Factor (Fa)

Enter the specification factor. This value is usually **1.1**.

Consider the Rotational Force Effect

Select this option to consider the rotational force of the earthquake in seismic calculations. This option enables you to enter values for **Site Velocity** and **Site Factor.**

Site Velocity (Vs)

Enter the site velocity value. This value is the speed of the S waves in the upper layers of the soil profile, as obtained through field measurements or laboratory tests of the shear modulus of the soil.

Site Factor (Fs)

Enter the site factor. This value is usually 1, 1<F.s<=1.5, or >1.5. It has a direct correlation with the value of r` and is gathered from information of soil mechanic analysis.

Mexico Sismo En Español

Mexico Sismo Seismic Data (page 501)

Zona Sismica

Se encuentran 4 opciones aqui. A, B, C y D. Estudie el Manual de Diseño por Sismo para Mexico. En la pagina 1.3.29 del manual, se encuentra un mapa con las diferentes regiones como se demuestra aqui. Se puede observar que la zona D es la zona que tiene la más alta actividad sismica, mientras que la zona A es la menos activa. Seleccione la zona correcta de el menu de opciones.

Grupo Estructural

Grupo A	Grado Alto de Seguridad
Grupo B	Grado Intermedio de Seguridad
Grupo C	Grado Bajo de Seguridad

Torres y tanques son considerados Estructuras del Grupo A porque se requiere un grado alto de seguridad durante su diseño. Estructuras Grupo B requieren un grado intermedio de seguridad y aquellos que son parte del grupo C requieren un grado bajo de seguridad.

Tipo de Terreno

I Terreno Duro:	Depositos del suelo formado exclusivamente por capas con velocidad de propagación b_0 = 700 m/s o el módulo de la rigidez >= 85000 t/m^2.
II Terreno Medio:	Depositos del suelo con el período fundamental de vibración y la velocidad efectiva de propagación que demuestra la condición: $\beta_c T_s + \beta_s T_c > \beta_c T_c$.
III Terreno	Depositos del suelo con el periodo fundamental de vibración efectiva de

Suave:	propagación que demuestra la condición: $\beta_c T_s + \beta_s T_c < \beta_c T_c$.

Factor de Conducta Q

El factor de conducta Q se encuentra en el Manual pagina 1.3.20. Para chimeneas y torres, este valor debe de ser 3.0. Para estructuras menores como tanques este valor puede ser 2.0. Vea la pagina 1.3.23 en el Manual de Sismo para informacion adicional.

Factor efectivo de Absorción

Este valor es zeta e hat y es utilizado para calcular el factor de incremento Xi. El EAF siempre debe de ser menor que o igual a 0.02.

Factor ortogonal de Aumento

El Codigo Sismico Mexicano considera un efecto SRSS sobre la estructura. Fundamentalmente, este valor aumenta las cargas sismicas en una forma lineal (Escalar). Este valor es traditionalmente 1.118 y siempre debe de ser > o igual a 1.0.

Notas de Analisis:

Al igual que con cada otro análisis de carga de terremoto, el objetivo es de calcular la fuerza cortante en el centro de masa de cada elemento del recipiente. Una vez que las fuerzas cortantes en cada elevación son encontradas, los momentos pueden ser acumulados a la base, apoyo de pierna u oreja.

El análisis empieza calculando las distancias de los pesos y centroides de todos los elementos del recipiente. Es muy importante modelar la estructura en secciones que son apropiadas de tamaño. Para cilindros, este valor está acerca de 10 o 12 pies (3m). Esto asegura que el programa tiene suficiente información para calcular el período natural de vibración con suficiente certeza.

Con los datos de entrada dados y pesos calculados de terremoto y frecuencia natural, PV Elite determina los valores de la tabla 3.1 del Código Sísmico Mexicano. Los valores son:

a_o	Coordenada Espectral para calcular a
c	Coordenada Espectral para calcular a
T_a(s)	Valor de Periodo para calcular a
T_b(s)	Valor de Periodo para calcular a
r	Exponente usado para calcular a

NOTE Para las estructuras del grupo A, los valores de las coordenadas espectrales (a_o, c) obtenidas de la tabla 3.1 son multiplicadas por 1.5.

Para este tipo de recipientes PV Elite utilizara el metodo de analisis statico como esta resumido en el codigo de sismo Mexicano. Si su pila o columna mide más de 60 metros o 192 pies, usted debe utilizar el método dinámico de análisis.

Después de que todos los varios factores son calculados, las fuerzas cortantes en cada nivel son calculadas en cada nivel según la fórmula en la página 1.3.88 del Código de Sismo. Este es el tipo tradicional de ecuaciones masivas de distribución de carga de suma de altura. Además de las fuerzas cortantes en cada nivel, la fuerza primera también es calculada por la ecuación 1.3.89. Después de que las cargas son calculadas, son multiplicadas por el factor ortogonal de aumento.

Con las fuerzas cortantes conocidas, los momentos son calculados de la cima a la base, utilizando tipicas ecuaciones de tipo estática.

NBC 1995 Seismic Data

Enter data for the Canadian NBC seismic code.

Importance Factor

Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from NBC 4.1.9.1 (10):

Classification	Importance Factor
Post-disaster buildings.	**1.5**
Schools.	**1.3**
All other buildings. **NOTE** Most petrochemical structures use this category.	**1.0**

Soil Type

Select a soil type:

- **Soil 1** - Category 1. From rock to stiff fine-grained soils up to 15 m deep.
- **Soil 2** - Category 2. From compact coarse-grained soils to soft fine-grained soils (stiff cohesionless clay) up to 15 m deep.
- **Soil 3** - Category 3. Very loose and loose coarse-grained soils with depth up to 15 m.
- **Soil 4** - Category 4. Very soft and soft fine-grained soils with depth greater than 15 m.

Force Modification Factor

Enter the force modification factor *R*, according to table 4.1.9.B and paragraphs 4.1.9.1 (8) and 4.1.9.3 (3).

R	Definition
1.0	Case 18 - Elevated tanks (such as equipment on legs), including the special provisions of paragraph 4.1.9.3 (3)
1.5	Case 6 - Ductile structures (such as towers on skirts).

Acceleration Seismic Zone

Select the acceleration seismic zone, according to the city list, Chapter 1 of the supplement to NBC:

Zone	Example City
0	Calgary, Alberta
1	Toronto, Ontario
2	Saint John, New Brunswick
3	Varennes, Quebec
4	Vancouver, British Columbia
5	Duncan, British Columbia
6	Port Hardy, British Columbia

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 6 indicates the greatest chance of an earthquake.

Velocity Seismic Zone

Select the velocity seismic zone, according to the city list, Chapter 1 of the supplement to NBC:

Zone	Example City
0	Steinbach, Manitoba
1	Calgary, Alberta
2	Montreal, Quebec
3	Quebec City, Quebec
4	Dawson, Yukon
5	Victoria, British Columbia
6	Destruction Bay, Yukon

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 6 indicates the greatest chance of an earthquake.IS-1893 SCM

NBC 2005 Seismic Data

Enter data for the NBC seismic code.

Importance Factor (IE)

Enter the earthquake importance factor *IE*, a value that accounts for loss of life and property. This value is referred to in Table 4.1.8.5 but is only shown for Ultimate Load State design. The software uses allowable stress design. This value should normally be greater than or equal to **1.0**.

Site Class

Enter the site class, designating the type of underlying strata the vessel is resting on. Site class A generates the least conservative result and site class E generates the most conservative result for the design base shear. The following table is adapted from Table 4.1.8.4.A, NBC 2005.

Site Class	Ground Profile Name
A	Hard rock
B	Rock
C	Very dense soil and soft rock
D	Stiff Soil
E	Soft Soil

Overstrength Factor (Ro)

Enter a value for the overstrength factor R_o. A value of **1** is conservative. A value greater than 1 leads to a less conservative, lower design base shear. R_o is taken from NBC table 4.1.8.9. The NBC Code is for buildings and support for vessels is lacking. You will not find a good value for R_o in the tables.

Ductility Factor (Rd)

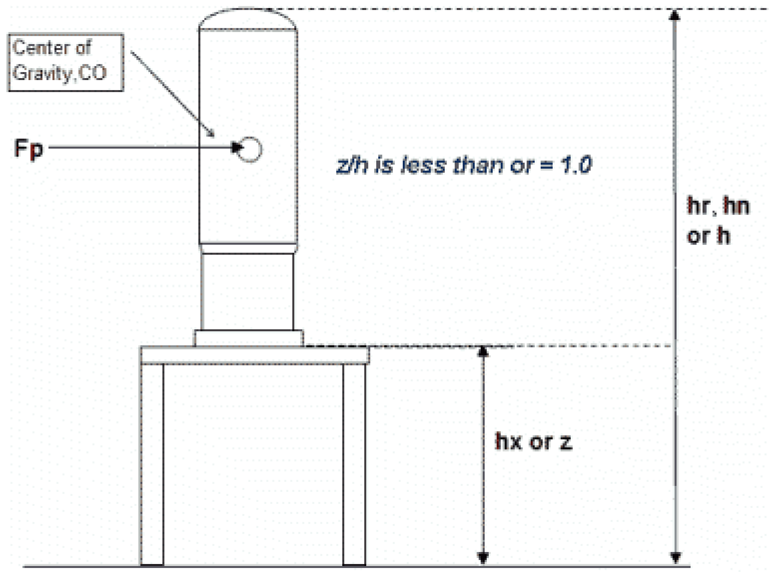
Enter the ductility factor, R_d, which is the ductility-related force modification factor reflecting the capability of a structure to dissipate energy through inelastic behavior. A value of **1** is conservative. A value greater than 1 leads to a less conservative, lower design base shear. R_d is taken from NBC table 4.1.8.9.

Accelerations from Appendix C Division B

Enter acceleration factors S_a for different acceleration periods. The acceleration factors for various cities and regions in Canada are found in Appendix C, Division B, Volume 2 of the NBC Code. These accelerations are used to determine *S(T)* according to paragraph 6, section 4.1.8.4. The software calculates each candidate and performs a final interpolation of the result based on the fundamental period of vibration of the vessel.

Component Elevation Ratio (hx/hn)

Enter a value for the elevation ratio if the vessel is supported by structure, such as a building. The ratio is the height in the structure where the vessel is attached *hx* to height to the top of the building from grade *hn*. Generally, this value is less than or equal to **1**. If the value is zero, the vessel is not building-supported.



Component Amplification Factor (Rp)

Enter the element or component response modification factor R_p, from table 4.1.8.17. For vessels, a value of **2.5** is typical. The larger the value of R_p is, the more conservative the result becomes.

Element or Component Factor (Cp)

Enter the element or component factor C_p, from table 4.1.8.17. A value of **1.0** is typical and is the default value. The larger the value of C_p, the more conservative the result.

Component Force Amp. Factor (Ar)

Enter the element or component force amplification factor *Ar*, from table 4.1.8.17. A value of **2.5** is typical and is the default value. The larger the value of *Ar*, the more conservative the result.

NBC 2010 Seismic Data

Enter data for the NBC 2010 seismic code.

Importance Factor (IE)

Enter the earthquake importance factor of the structure, I_E, as described in Table 4.1.8.5.

Importance Category	Importance Factor, I_E
Low	0.8
Normal	1.0
High	1.3

Post-disaster	1.5

Site Class

Select the site class as defined in Tables 4.1.8.4.A, 4.1.8.4.B, or 4.1.8.4.C.

Overstrength Factor (Ro)

Enter the overstrength factor, R_o, as defined in Table 4.1.8.9.

Ductility Factor (Rd)

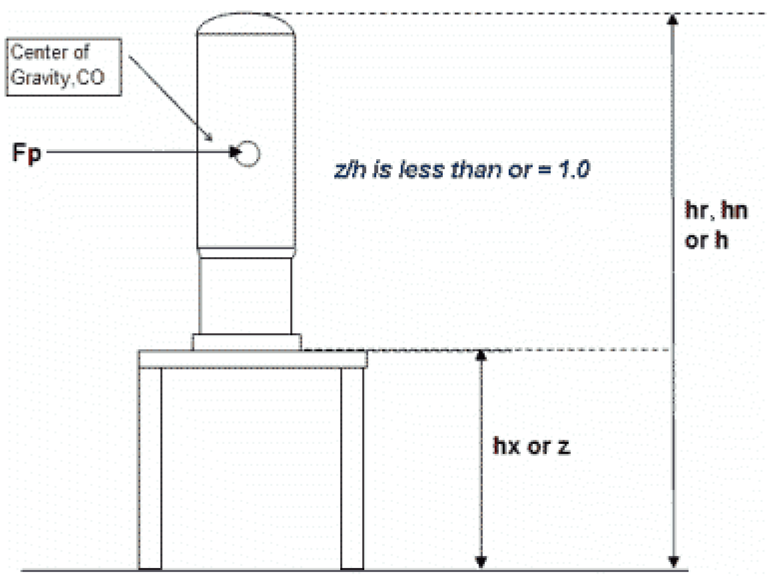
Enter the ductility factor, R_d, as defined in Table 4.1.8.9. When an R_d value is required, you must use the corresponding R_o value from the table.

Accelerations from Appendix C Division B

Enter the accelerations that you want to use. The acceleration factors for various cities and regions in Canada are found in Volume 2, Division B, Appendix C.

Component Elevation Ratio (hx/hn)

Enter a value for the elevation ratio, h_x/h_n , when the vessel is supported by structure, such as a building. The ratio is the height in the structure where the vessel is attached, h_x, to the height to the top of the building from grade, h_n. Generally, this value is less than or equal to **1**. If the value is **0**, the vessel is not building-supported.



Component Amplification Factor (Rp)

Enter the component amplification factor, R_p, as defined in Table 4.1.8.18.

Element or Component Factor (Cp)

Enter the seismic coefficient factor, C_p, as defined in Table 4.1.8.18.

Component Force Amp. Factor (Ar)

Enter the response amplification factor, A_r, as defined in Table 4.1.8.18.

NBC 2015 Seismic Data

Enter data for the NBC seismic code.

Importance Factor [IE]

Enter the earthquake importance factor of the structure, I_E, as described in Table 4.1.8.5.

Importance Category	Importance Factor, I_E
Low	0.8
Normal	1.0
High	1.3
Post-disaster	1.5

Site Class

Select the site class as defined in Tables 4.1.8.4.A, 4.1.8.4.B, or 4.1.8.4.C.

Overstrength Factor [Ro]

Enter the overstrength factor, R_o, as defined in Table 4.1.8.9.

Ductility Factor [Rd]

Enter the ductility factor, R_d, as defined in Table 4.1.8.9. When an R_d value is required, you must use the corresponding R_o value from the table.

Peak Ground Acceleration [PGA]

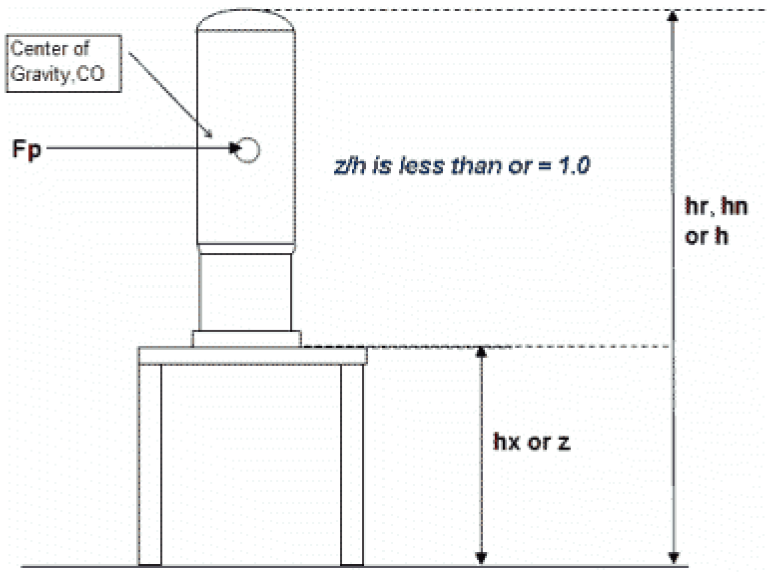
Enter the peak ground acceleration expressed as a ratio to gravitational acceleration as defined in sentence 4.1.8.4(1). The value is based on the province and location from NBC 2015 Volume 1, Division B, Appendix C, Table C-3.

Accelerations from Appendix C Division B

Enter the accelerations that you want to use. The acceleration factors for various cities and regions in Canada are found in Volume 2, Division B, Appendix C.

Component Elevation Ratio [hx/hn]

Enter a value for the elevation ratio, h_x/h_n , when the vessel is supported by structure, such as a building. The ratio is the height in the structure where the vessel is attached, h_x, to the height to the top of the building from grade, h_n. Generally, this value is less than or equal to **1**. If the value is **0**, the vessel is not building-supported.



Component Amplification Factor [Rp]

Enter the component amplification factor, R_p, as defined in Table 4.1.8.18.

Element or Component Factor [Cp]

Enter the seismic coefficient factor, C_p, as defined in Table 4.1.8.18.

Component Force Amp. Factor [Ar]

Enter the response amplification factor, A_r, as defined in Table 4.1.8.18.

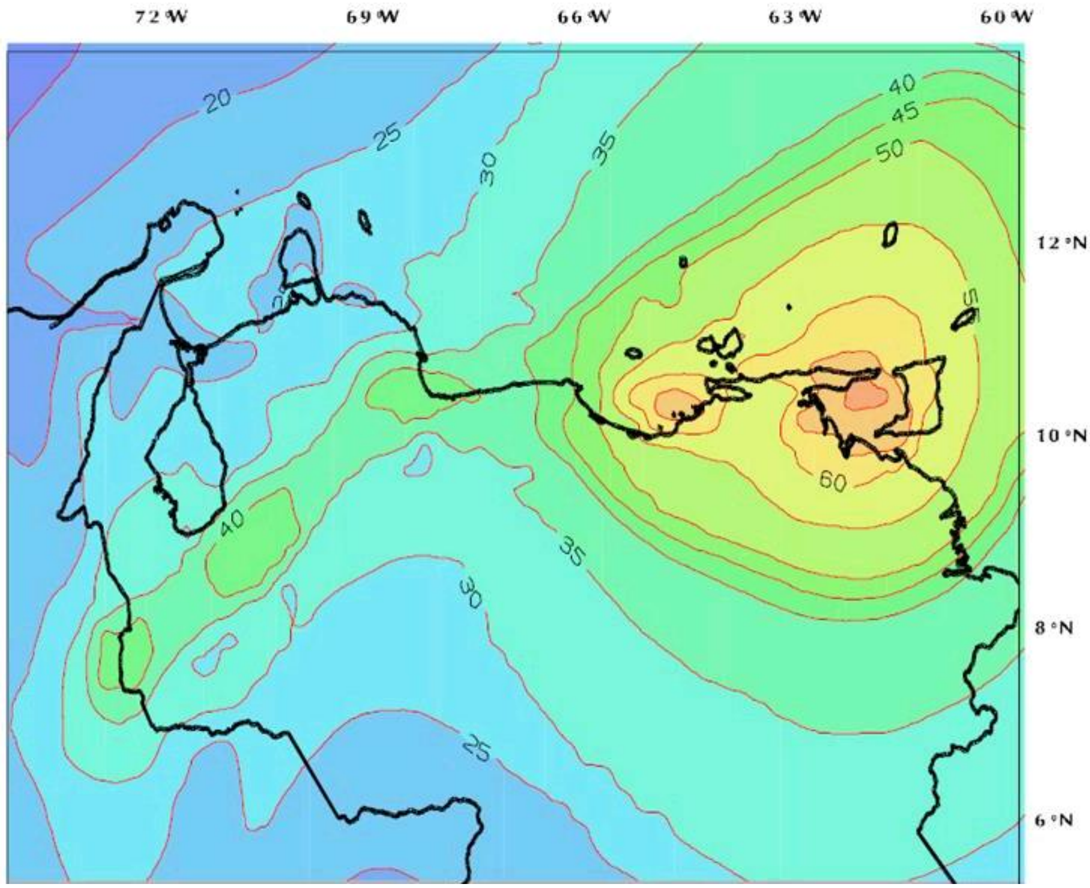
PDVSA Seismic Data

PDVSA En Español (page 516)

Enter data for the Venezuelan Code for Seismic Analysis of Vessels.

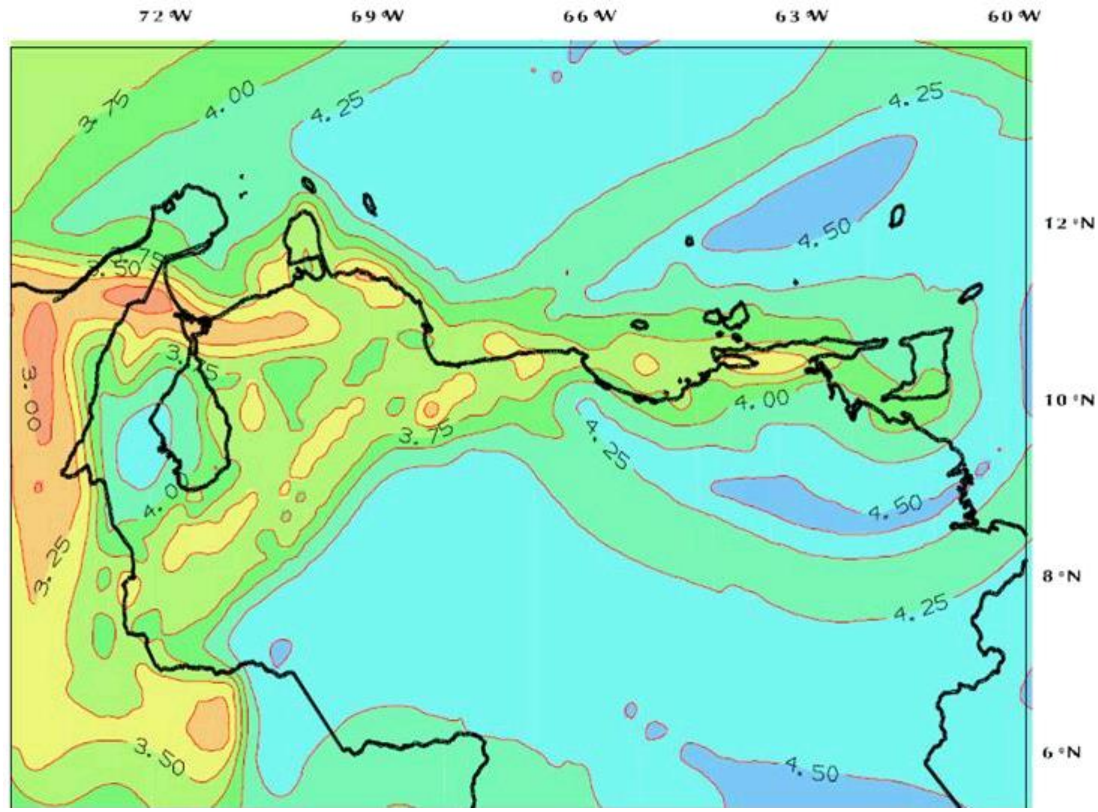
Characteristic Accel. (a*)

Enter the characteristic acceleration *a**. The value is obtained from Figure 6.1 seismic threat map:



Gamma

Enter the gamma factor γ. The value is obtained from Figure 6.2 seismic threat map:



Annual Probability

Enter the risk classification annual probability P_1. Select the risk classification based on the most critical consequences defined in Section 4.1 Reference Scale, JA-221 page 9, Table 4.1 Risk Classification Scale:

Degree of Risk	Conditions				P1 (10^{-3})
	Number of people exposed	Economic Loss		Environmental impact	
		Materials	Lost profits		
A	Few (< 10)	Limited to installation	Worthless	Little or None	**≤ 2**
B	Important (11 to 100)	The installation and any neighbor	Significant. Between 1 and 50 US$	Recovery ≤ 3 years	**≤ 1**
C	Great number of people (100 to 500)	The installation and many neighbors	Between 50 and 250 US$	Recovery 3 to 10 years	**≤ 0.5**
D	> 500 people	Natural	> 250 US$	Irreversible	**≤ 0.1**

		catastrophe			

NOTE For unknown or doubtful cases of risk, select the highest risk classification.

Damping Factor

Enter the damping factor ζ, as defined in Section 6, The Seismic Design Movements, JA-222 page 4, Table 6.1:

Type of Structure	Damping (%)
Welded steel	**3**
Bolted steel	**5**
Reinforced concrete	**5**
Metallic tanks	**See FJ-251**

Ductility Factor

Enter the damping factor *D*, Select *D* for the type of structure that dissipates energy in a way that does not compromise the stability of the vessel system, as defined in Section 8 Ductility Factor, JA-222 page 4, Table 8.1:

Type of Structure	D	Comments
Vertical Vessel (metallic skirt and connection bolts in the foundation)	**2**	The eventual failure of the skirt and connection bolts is ductile. It supplies sufficient deformation to the bolts by means of exposed backups.
Vertical Vessel (metallic skirt and connection bolts in the foundation)	**2**	A ductile failure in the connection bolts on the foundation. Must avoid premature fragile failure in the skirt. It supplies sufficient deformation to the bolts by means of exposed backups.
Vertical Vessel (metallic skirt and connection bolts in the foundation)	**1**	None-ductile failure in the system in the skirt or in the bolts.
Horizontal Vessel (on top of arcs of high hyperstatic moments)	**6**	The mechanical failure in the saddle must be ductile, under the concept of strong column weak beam. No fragile failure between the vessel and the saddle.
Horizontal Vessel (on top of arcs of high hyperstatic moments)	**4**	The mechanical failure in the saddle must be ductile, under the concept of strong column weak beam. No fragile failure between the vessel and the saddle.
Horizontal Vessel (on top of single columns)	**1.5**	Ductile failure on the column.

Horizontal Vessel (on top of a braced arc)	**4**	Ductile failure from the system.
Horizontal Vessel (on top of walls)	**1.5**	In the walls plane.
	2	In the walls vertical plane.
Horizontal Vessel (single diagonal braced columns)	**1.5**	In the non-bracing plane. Ductile failure on the column.
	3	In the bracing plane. It first yields the system of bracing in a ductile form.

Beta Factor

Enter the beta factor β, as defined in Section 6.3 Elastic Response Spectrum, Table 6.1 Spectrum Form Values, JA-221 page 15, Table 6.1 Spectrum Form Values:

Spectral Form	**β**	**T° (s)**	**T* (s)**
S1	2.4	0.10	0.4
S2	2.6	0.20	0.8
S3	2.8	0.30	1.2
S4	3.0	0.40	1.6

Ad - Spectrum acceleration divided by the gravity's acceleration
g - Gravity acceleration
Ao - Maximum acceleration coefficient of the terrain, Ao = a/g
T° - Value of the period that defines part of the normalized elastic spectral, in seconds
T* - Maximum value of the period in the interval where the normalized elastic spectra have a constant value, in seconds
β - One of the parameters that define the spectral form

T* Factor

Enter the T* factor, as defined in Section 6.3 Elastic Response Spectrum, Table 6.1 Spectrum Form Values, JA-221 page 15, Table 6.1 Spectrum Form Values:

Spectral Form	**β**	**T° (s)**	**T* (s)**
S1	2.4	0.10	0.4
S2	2.6	0.20	0.8
S3	2.8	0.30	1.2
S4	3.0	0.40	1.6

Ad - Spectrum acceleration divided by the gravity's acceleration
g - Gravity acceleration
Ao - Maximum acceleration coefficient of the terrain, Ao = a/g
T° - Value of the period that defines part of the normalized elastic spectral, in seconds
T* - Maximum value of the period in the interval where the normalized elastic spectra have a constant value, in seconds
β - One of the parameters that define the spectral form

T+ Factor

Enter the T^+ factor, the minimum value of the period in the interval where the design spectral has a constant value, as defined in Section 7 Spectrum Design, JA-221 page 18, Table 7.1:

	T^+ (s)
D < 5	0.1 (D-1)
D ≥ 5	0.4
Should comply with $T^o \le T^+ \le T^*$	

Phi Factor

Enter the phi factor φ, as defined in Section 5.1 Factor φ and Spectral Form Selection, JA-221 page 11, Table 5.1 Factor φ and Spectral Form Selection:

Material	**V_{sp} (m/s)**	**H (m)**	**Spectral form**	**φ**
Healthy rock / fractured	> 700	Any	S1	**0.85**
Soft rock or moderate	> 400	≤ 50	S1	**0.90**
		> 50	S2	**0.95**
Very hard or very dense floors	> 400	< 30	S1	**0.90**
		30 - 50	S2	**0.95**
		> 50	S3	**1.00**
Hard or dense floors	250 - 400	< 15	S1	**0.90**
		15 - 50	S2	**0.95**
		50 - 70	S3 [2]	**1.00**
		> 70	S4	**1.00**
Firm floors / medium dense	170 - 250	≤ 50	S2 [3]	**1.00**

		> 50	S3 [2]	**1.00**
Soft floors / loose	< 170	≤ 15	S2 [3]	**1.00**
		> 15	S3 [2]	**1.00**
Soft strata interspersed with other more rigid floors [1]	< 170	< H1	S2	**1.00**
		> H1	S3	**0.90**

H - Depth to which the material has a velocity, Vs, greater than 500 m/s.
H1 - Depth from the surface up to the top of the soft strata (m): ≥ 0.25 H.
$\mathbf{V}_{sp}$ - Average velocity on the geotechnic profile (m/s).
φ - Correction factor for the horizontal acceleration coefficient.

NOTES

a. The thickness of the strata should be greater than 0.1 H.

b. If $A_o \leq 0.15$, use S4.

c. If $A_o \leq 0.15$, use S3.

PDVSA En Español

PDVSA Seismic Data (page 510)

Seccion 4.1 Escala de Referencia (JA-221 pagina 9)

Seleccionar el Grado de Riesgo asociado con el renglon de consecuencias mas desfavorables descritas en la Tabla 4.1

p1 : Probabilidad anual de excedencia

Tabla 4.1 Escala de Clasificacion de Riesgos

Grado de Riesgo	**CONDICIONES**				**P1 (10-3)**
	Numero de personas expuestas	**Perdidas economicas**		**Impacto Ambiental**	
		Materiales	**Lucro Cesante**		
A	Pocas (< 10)	Limitado a la instalacion	Despreciable	Poco o Nulo	**≤ 2**
B	Importante (11 to 100)	La instalacion y alguna vecina	Significativo Entre 1 y 50 MMUS$	Recuperacion ≤ 3 años	**≤ 1**
C	Elevado numero de personas (100 to 500)	La instalacion y numerosas vecinas	Entre 50 y 250 MMUS$	Recuperacion 3 a 10 años	**≤ 0.5**

D	> 500 personas	De naturaleza catastrofica	> 250 MMUS$	Irreversible	≤ 0.1

Nota: Cuando se presenten dudas en la seleccion del Grado de Riesgo, se adoptara el grado de mayor riesgo

Seccion 6 Movimientos Sismicos de Diseño (JA-222 pagina 4)

ζ : factor de amortiguamiento

Tabla 6.1 Factor de Amortiguamiento

Tipo de Estructura	Amortiguamiento (%)
Acero soldado	**3**
Acero empernado	**5**
Concreto reforzado	**5**
Tanques metalicos	**Segun Especificacion FJ-251**

Seccion 8 Factor de Ductilidad (JA-222 pagina 4)

El diseño debe garantizar que el mecanismo de disipacion de energia en el cual se fundamenta D no comprometa la estabilidad del sistema

D : factor de ductilidad

Tabla 8.1 Factores de Ductilidad

Tipo de Esctructura	D	Comentarios
Recipiente Vertical (falda metalica y pernos de conexion con la fundacion)	**2**	La eventual falla de la falda, asi como de los pernos de conexion, es ductil. Se suministra suficiente longitud de deformacion a los pernos mediante soportes expuestos ("silla para pernos").
Recipiente Vertical (falda metalica y pernos de conexion con la fundacion)	**2**	Falla ductil en los pernos de conexion con la fundacion. Se debe evitar la falla fragil de la falda. Se suministra suficiente longitud de deformacion a los pernos mediante soportes expuestos
Recipiente Vertical (falda metalica y pernos de conexion con la fundacion)	**1**	Falla no ductil del sistema, en la falda y/o pernos. No se recomienda esta situacion
Recipiente Horizontal (sobre porticos de momento	**6**	El mecanismo de falla del portico debe de ser ductil, bajo el concepto de columna fuerte viga debil. No se produce falla fragil y/o prematura en la conexion

de alta hiperestacidad)		entre el recipiente y el portico
Recipiente Horizontal (sobre porticos de momento de baja hiperestacidad)	**4**	El mecanismo de falla del portico debe de ser ductil, bajo el concepto de columna fuerte viga debil. No se produce falla fragil y/o prematura en la conexion entre el recipiente y el portico
Recipiente Horizontal (sobre monocolumnas)	**1.5**	Falla ductil de la columna
Recipiente Horizontal (sobre portico arriostrado)	**4**	Falla ductil del sistema
Recipiente Horizontal (sobre muros)	**1.5**	En el plano de los muros
	2	En el plano perpendicular a los muros
Recipiente Horizontal (monocolumnas arriostradas con diagonales)	**1.5**	En el plano no arriostrado. Falla ductil de la columna.
	3	En el plano arriostrado. Cede primero el sistema de arriostramiento en forma ductil

Seccion 10 Seleccion del Metodo de Analisis (JA-222 pagina 8)

Tabla 10.1 Seleccion del Metodo de Analisis

Altura de la Estructura (m)	**Ao**	**Grado de Riesgo**			
		A	**B**	**C**	**D**
≤ 10	≤ 0.15	Se	Se	Dy	Dy
	> 0.15	Se	Dy	Dy	Dy
> 10	Any	Dy	Dy	Dy	Dy

Es : Estatico Equivalente
Di: Analisis Dinamico

Nota : El Metodo Estatico Equivalente es el requerimiento minimo para los casos descritos; por tanto, puede ser reemplazado por el Metodo de Analysis Dinamico

Seccion 5.1 Seleccion de la Forma Espectral y del Factor φ (JA-221 pagina 11)

H : Profundidad a la cual se consigue material con velocidad, Vs, mayor que 500 m/s
H1 : Profundidad desde la superficie hasta el tope del estrato blando (m): ≥ 0.25 H
V_{sp} :Velocidad promedio de las ondas de corte en el perfil geotecnico (m/s)
φ :Factor de correccion del coeficiente de aceleracion horizontal

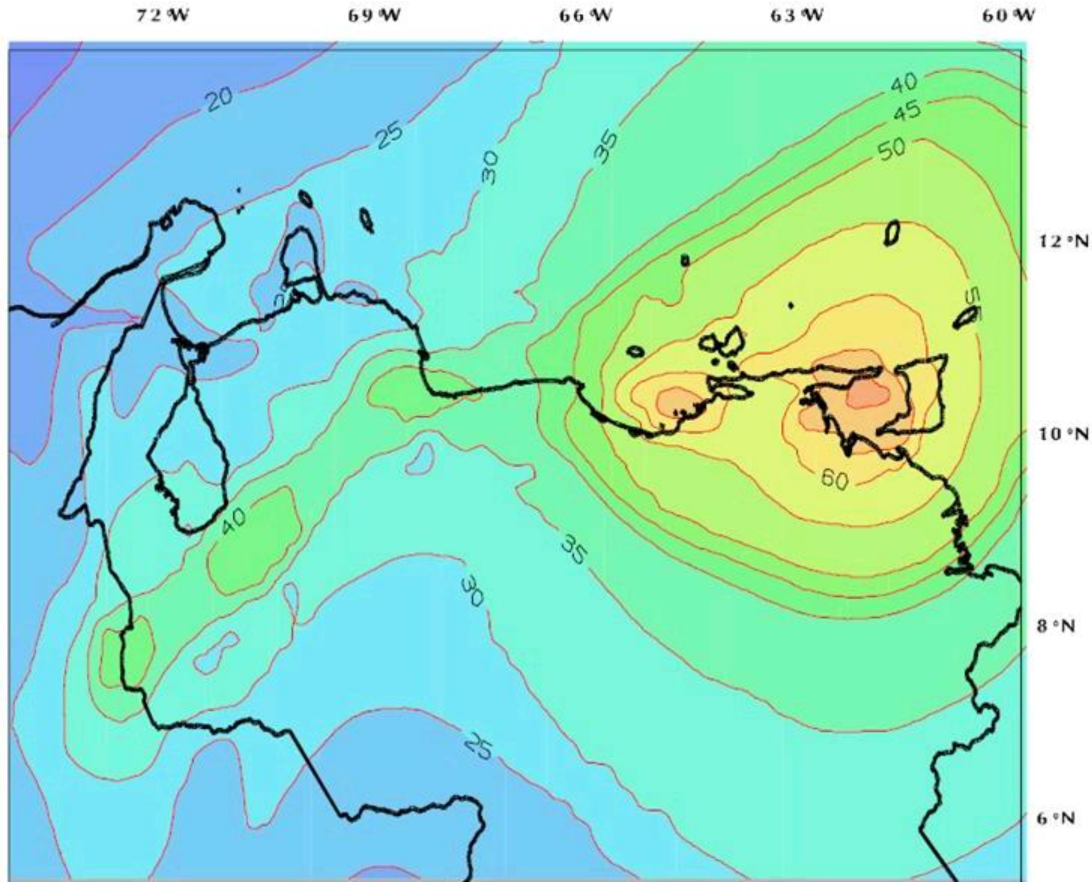
T**able 5.1 Forma Espectral Tipificada y Factor φ**

Material	**V_{sp} (m/s)**	**H (m)**	**Forma espectral**	**φ**
Roca sana / fracturada	> 700	Cualquiera	S1	**0.85**
Roca blanda o moderadamente meteorizada	> 400	≤ 50	S1	**0.90**
		> 50	S2	**0.95**
Suelos muy duros o muy densos	> 400	< 30	S1	**0.90**
		30 - 50	S2	**0.95**
		> 50	S3	**1.00**
Suelos duros o densos	250 - 400	< 15	S1	**0.90**
		15 - 50	S2	**0.95**
		50 - 70	S3 (b)	**1.00**
		> 70	S4	**1.00**
Suelos firmes / medio densos	170 - 250	≤ 50	S2 (c)	**1.00**
		> 50	S3 (b)	**1.00**
Suelos blandos / sueltos	< 170	≤ 15	S2 (c)	**1.00**
		> 15	S3 (b)	**1.00**
Estratos blandos intercalados con otros suelos mas rigidos(a)	< 170	< H1	S2	**1.00**
		> H1	S3	**0.90**

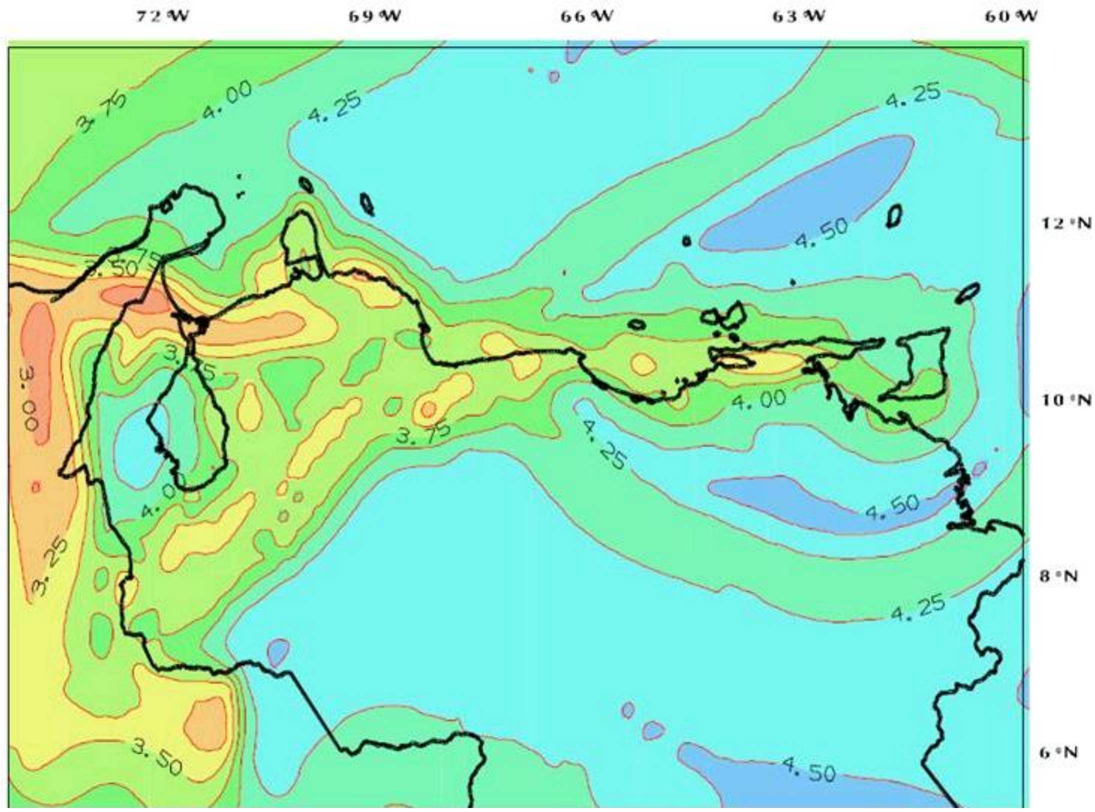
(a) El espesor de los estratos debe ser mayor que 0.1 H
(b) Si $A_o \leq 0.15$, usese S4
(c) Si $A_o \leq 0.15$, usese S3

Seccion 6.1 Mapas de Amenaza Sismica (JA-221 pagina 12)

a*, γ : Valores obtenidos de los Mapas de Amenaza Sismica dados en las Figuras 6.1 y 6.2



Nota: Fig. 6.1 Mapa Amenaza Sismica, valores de a*



Nota: Fig. 6.2 Mapa Amenaza Sismica, valores de γ

Seccion 6.3 Espectro de Respuesta Elastica (JA-221 pagina 15)

Ad : Aceleracion espectral dividida por la aceleracion de gravedad (g)
g : Aceleracion de gravedad
A_o : Coeficiente de aceleracion maxima del terreno, Ao = a/g
T^o : Valor maximo del periodo que define parte del espectro elastico normalizado, en segundos.
T* : Valor maximo del periodo en el intervalo donde los espectros elasticos normalizados tienen un valor constante, en segundos
β : Uno de los parametros que definen la forma de los espectros.

Tabla 6.1 Valores que definen la Forma del Espectro

Forma espectral	β	T^o (s)	T* (s)
S1	2.4	0.10	0.4
S2	2.6	0.20	0.8
S3	2.8	0.30	1.2
S4	3.0	0.40	1.6

Seccion 7 Espectros de Diseño (JA-221 pagina 18)

T^+ : Menor valor del periodo en el intervalo donde los espectros de diseño tienen un valor constante, en segundos.

Tabla 7.1 Valores de T^+ (segundos)

	T^+ (s)
D < 5	0.1 (D-1)
D ≥ 5	0.4
Se debe cumplir $T^o \leq T^+ \leq T^*$	

Res. Spectrum Seismic Data

Select this method to perform a dynamic analysis of the vessel, applying loads based upon the selected seismic response spectrum.

Initially, the vessel is modeled as a two-dimensional structure. An Eigen solution is then performed, which determines system mode shapes and modal natural frequencies. All modes with natural frequencies up through 100 Hz are calculated. The seismic response of each mode is extracted from the response spectrum according to the natural frequency of each mode, and then adjusted according to the mode participation factor. The system response is then determined by combining all of the modal responses.

For tall structures, this analysis gives a much more accurate calculation than the typical static equivalent method. The calculated loads are usually lower in magnitude than those calculated using conventional building code techniques.

Response Spectrum Name

Select the name of the spectra or code to use for the dynamic analysis:

- **User Defined** - When this option is selected, you must select values for **Range Type, Ordinate Type, and Include Missing Mass Components**. You must also click **Edit / Review Spectrum Points** and enter values in the table. The same spectrum is applied in the horizontal and vertical directions.
- **ELCENTRO** - This response spectrum is based on the May 18, 1940 El Centro, California earthquake, North-South component, 5-10% damping, as described in *Introduction to Structural Dynamics* by John Biggs. The same spectrum is applied in the horizontal and vertical directions.
- **ASCE** - Seismic analysis is performed according to the modal analysis procedure of ASCE Standard 7-98. The horizontal spectrum is built according to the ASCE-7 Section 9.4.1.2.6, while the vertical spectrum provides a flat acceleration of 0.2S.
- **IBC** - Seismic analysis is performed according to the modal analysis procedure of the International Building Code 2000, which mirrors ASCE-7. The horizontal spectrum is built according to IBC-2000 Section 1615.1, while the vertical spectrum provides a flat acceleration of 0.2 (according to IBC-2000 Section 1617. 1).

- **1.60D.5** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 0.5% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D2** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 2% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D5** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 5% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D7** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 7% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.
- **1.60D10** - The horizontal (X) and vertical (Y) spectra are specified in the United States Nuclear Regulatory Commission Regulatory Guide 1.60, for systems with 10% of critical damping. This spectrum is normalized, so it must be scaled to the value entered for **Zero Period Acceleration**.

Importance Factor

Enter the importance factor, as given in 9.1.4 (ASCE) 1604.5 (IBC). The value is usually between **1.0** and **1.5**. The importance factor accounts for loss of life and property. For ASCE, this is *I*, the occupancy importance factor according to ASCE-7 Section 9.14. For IBC, this is *Ie*, the occupancy importance factor according to IBC 1616.2.

Shock Scale Factor X | Y dir

Enter values for the shock scale factor in the X and Y directions. The factors are used as multipliers on the horizontal and vertical spectrum data points. The value is usually **1**, but can be higher or lower to scale the spectra up or down. For example, many seismic specifications require that the vertical spectrum be 2/3 the magnitude of the horizontal spectrum. This corresponds to an X value of **1.0** and a Y value of **0.6667**.

In the analysis of vertical vessels, the component in the vertical direction is typically ignored. You can enter a value of **0** in the Y direction field.

Zero Period Acceleration

Enter a value for the ground acceleration *ZPA*. This parameter does double duty, depending upon the analysis type. When used with certain predefined normalized response spectra, it is used as the acceleration factor (in g's) by which the spectrum is scaled. For example, when a spectrum analysis uses a **Response Spectrum Name** beginning with 1.60, such as, **1.60D.5** or **1.60D7**, the software constructs an earthquake spectrum according to the instructions given USNRC Regulatory Guide 1.60. That guide requires that the shape of the response spectrum be chosen from the curves shown in Figures 6-5 and 6-6, based upon the system damping value (such as the ".5" in the spectrum names 1.60H.5). The Reg Guide 1.60 curves are normalized to represent ZPA of one g; the true value is actually site dependent. Therefore, entering a ZPA value here appropriately scales any Regulatory Guide 1.60 curve to meet site requirements.

Combination Method

Enter the spectral summation method mode that most accurately captures the statistical correlation of the responses:

- **SRSS** - Square Root of the Sum of the Squares. This method states that the total system response is equal to the square root of the sum of the squares of the individual modal responses. (This is effectively the same as using the DSRSS method with all correlation coefficients equal to 0.0, or the **Group** method, with none of the modes being closely spaced.) This method is based upon the statistical assumption that all modal responses are completely independent, with the maxima following a relatively uniform distribution throughout the duration of the applied load. This is usually non-conservative, especially if there are any modes with very close frequencies, because those modes will probably experience their maximum DLF at approximately the same time during the load profile.
- **Group** - This method is defined in USNRC Regulatory Guide 1.92. The grouping method attempts to eliminate the drawbacks of the **Absolute** and **SRSS** methods by assuming that modes are completely correlated with any modes with similar (closely spaced) frequencies, and are completely uncorrelated with those modes with widely different frequencies. Effectively, this method dictates that the responses of any modes which have frequencies within 10% of each other first be added together absolutely, with the results of each of these groups then combined with the remaining individual modal results using the **SRSS** method.
- **Absolute** - This method states that the total system response is equal to the sum of the absolute values of the individual modal responses. (This is effectively the same as using the DSRSS method with all correlation coefficients equal to 1.0, or the **Group** method, with all modes being closely spaced.) This method gives the most conservative result, because it assumes that the all maximum modal responses occur at exactly the same time during the course of the applied load. This is usually overly-conservative, because modes with different natural frequencies will probably experience their maximum DLF at different times during the load profile.

Acc. Based Factor Fa

Enter a value for the acceleration factor F_a. This factor is from Table 9.4.1.2.4A (ASCE) or Table 1615.1.2(1) (IBC), and is a function of **Max. Mapped Res. Acc. Ss** and **Site Class:**

	F_a for Mapped Short-Period Maximum Considered Earthquake Spectral Acceleration (S_s) of:				
Site Class	**S_s <= 0.25**	**S_s = 0.5**	**S_s = 0.75**	**S_s = 1.0**	**S_s >= 1.25**[2]
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.2**	**1.2**	**1.1**	**1.0**	**1.0**
D	**1.6**	**1.4**	**1.2**	**1.1**	**1.0**
E	**2.5**	**1.7**	**1.2**	**0.9**	**[0.9]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analysis shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Acc. Based Factor Fv

Enter a value for the acceleration factor F_v. This factor is from Table 9.4.1.2.4B(ASCE) or Table 1615.1.2(2) (IBC), and is a function of **Max. Mapped Res. Acc. Sl** and **Site Class:**

	F_v for Mapped 1-Second-Period Maximum Considered Earthquake Spectral Acceleration (S_l) of:				
Site Class	**S_l <= 0.1**	**S_l = 0.2**	**S_l = 0.3**	**S_l = 0.4**	**S_l >= 0.5[2]**
A	**0.8**	**0.8**	**0.8**	**0.8**	**0.8**
B	**1.0**	**1.0**	**1.0**	**1.0**	**1.0**
C	**1.7**	**1.6**	**1.5**	**1.4**	**1.3**
D	**2.4**	**2.0**	**1.8**	**1.6**	**1.5**
E	**3.5**	**3.2**	**2.8**	**2.4**	**[2.4]**[1]
F	**See note 1**				

NOTES

- Site-specific geotechnical information and dynamic site response analyses shall be performed.
- Site specific studies required according to Section 9.4.1.2.4 may result in higher values of S_s than included on hazard maps, as may the provisions of Section 9.13.
- The software also checks tables 1616.3(1) and 1616.3(2) to determine the seismic use group. If Sd1 is greater than or equal to 0.2 and S1 is greater than or equal to 0.75 then the additional check of equation 16-38 (IBC) is made. The same logic is used if Sds is greater than or equal to 0.50.

Max. Mapped Res. Acc. Ss

Enter the mapped short-period (0.2 second) maximum considered earthquake spectral acceleration parameter S_s, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003).

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Max. Mapped Res. Acc. Sl

Enter the mapped long period (1 second) maximum considered earthquake spectral acceleration parameter S_l, taken from the ASCE 7-98 / IBC 2000/2003 publication. The tables are found on pages 100 - 117 (ASCE 98), page 351 (IBC 2000), or page 323 (IBC 2003). Typical values are 0.0 through 1.5g.

NOTE Do not enter percentage value. Enter a multiplier of g. For example, if the value in the table is 25%, enter 0.25.

Response Modification R

For ASCE 7-98 or ASCE 7-02/05 and IBC 2000, IBC 2003, or IBC 2006, enter the response modification factor *R*, from table 9.5.2.2 or 1617.6.

For ASCE 2010 and IBC 2012, enter the seismic force factor from Table 12.2-1, 12.14-1, 15.4-1, or 15.4-2 as required.

The value is usually **2.5** for inverted pendulum systems and cantilevered column systems and **4.0** for elevated tanks. For horizontal vessels, leg supported vessels, and others, use a value of **3.0**. The larger the value of *R*, the more conservative the analysis becomes.

Coefficient Cd

Enter a value for C_d, the deflection amplitude factor given. For ASCE-7 it is determined from Table 9.5.2.2, while for IBC-2000 it is determined from Table 1617.6. This parameter is used in conjunction with the importance factor and the deflection at level x to calculate the modal deflection at each level. The values in the table range from **2** to **6.5**. The larger value leads to a more conservative result.

Range Type

Select the range type of the abscissa/ordinate axis of the spectrum: **Frequency** or **Period**. Click **Edit / Review Spectrum Points** to enter data points in ascending order.

Ordinate Type

Select the Y-axis ordinate type of the spectrum curve:

- **Displacement** - Diameter
- **Velocity** - Diameter/second
- **Acceleration** - Diameter/second^2 of G's

Include Missing Mass Components

Select to include a correction to the spectrum analysis which represents the contribution of higher order modes not explicitly extracted for the modal/dynamic response, thus providing greater accuracy without additional calculation time. When this option is selected, the software automatically calculates the net (in-phase) contribution of all non-extracted modes and combines it with the modal contributions, avoiding the long calculation time associated with the extraction of the high order modes and the possible excessive conservatism of the summation methods.

During spectrum (either seismic or force spectrum) or time history analyses, the response of a system under a dynamic load is determined by superposition of modal results. One of the advantages of this type of modal analysis is that usually only a limited number of modes are excited and need to be included in the analysis. Only modes with natural frequencies up to about 100 Hz are used. The drawback to this method is that although displacements may be obtained with good accuracy using only a few of the lowest frequency modes, the force, reaction, and stress results may require extraction of far more modes (possibly far into the rigid range) before acceptable accuracy is attained. Select **Include Missing Mass**

Components to avoid this drawback.

Edit / Review Spectrum Points

Click to open the **Spectrum Data Points** dialog box. Enter your values for **Acc. G's** at each needed **Period Secs**. Enter data points in ascending order. Interpolation is made linearly for intermediate range values. A zero value is invalid.

SANS 10160-4:2010 Seismic Data

South African National Standard (SANS) seismic data code guidelines detail standards for structural design and seismic recommendations for buildings and industrial structures. More specifically, SANS 10160-4 specifies the seismic zones in which to design buildings that may be subject to earthquake. The standard focuses on preventing major catastrophic structural failures and loss of life, rather than just general building damage.

Reference Horizontal Peak Ground Acceleration

Specifies the horizontal peak ground acceleration .South African seismic zones are determined from a calculation that figures a peak acceleration with a 10 percent probability of occurring within 50 years. For a map of the seismic hazard zones of South Africa, see *SANS 10160-4:2010 Section 5.2, Figure 1*. The SANS 10160-4 equations define the reference horizontal peak ground acceleration factor as a_g for type 1 ground, where *g* refers to earth gravity acceleration.

Ground Type

Specify the ground type. SANS 10160-4 guidelines list the following ground types:

Ground Type	Description
1	Rock or other rock-like geological formation (with at most five meters of weaker surface material).
2	Dense sand, gravel, or very stiff clay deposits (at least several tens of meters in thickness with a gradual increase of mechanical properties with depth).
3	Dense or medium-dense sand, gravel, or stiff clay with deep deposits (with a thickness from several tens to many hundreds of meters).
4	Loose-to-medium cohesion-less soil (with or without soft cohesive layers) or predominately soft-to-firm cohesive soil.

See SANS 10160-4:2010 Section 5.2, Table 1 for more details on ground types and their parameters. Also see Section 5.3, Table 2 for ground type parameter values.

Behavior Factor

Enter the behavior factor (q). The SANS 10160-4:2010 guidelines advise to use a behavior factor in seismic calculations to reduce the elastic response spectrum. This spectrum helps when analyzing the elasticity of a structure. See SANS 10160-4:2010 Section 8.2 and Table

4 for more information.

UBC 1994 Seismic Data

Enter data for the UBC 1994 seismic code.

Importance Factor

Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from Table 23-L of the UBC standard:

Category	Classification	Importance Factor
I	Essential facilities	**1.25**
II	Hazardous facilities	**1.25**
III	Special occupancy structures	**1.0**
IV	Standard occupancy structures	**1.0**

Soil Type

Select a soil type, defined in Table 23-J of the code:

- **Soil 1** - Soil Profile S1. Rock or stiff soil conditions (S Factor = 1.0).
- **Soil 2** - Soil Profile S2. Deep cohesionless deposits or stiff clay conditions (S Factor = 1.2).
- **Soil 3** - Soil Profile S3. Soft to medium-stiff clays and sands (S Factor = 1.5).
- **Soil 4** - Soil Profile S4. More than 40 ft. of soft clay (S Factor = 12.0).

When soil properties are not known, select **Soil 3**.

Horizontal Force Factor

Enter the seismic force factor for non-building structure *Rw,* according to UBC Table 23-Q:

- **3** - Tanks, vessels, or pressurized spheres on braced or unbraced legs.
- **4 -** Distributed mass cantilever structures such as stacks, chimneys, silos, and skirt supported vertical vessels.

Seismic Zone

Select the seismic zone, according to UBC-97, Figure 16-2:

Seismic Zone	Description	Seismic Zone Factor Z (Table 16-I)
0	Gulf and prairies	0.00
1	Rockies and Appalachian areas	0.075

2a	New England, Carolinas, and Ozarks	0.15
2b	Valley area west of the Rockies and the Pacific Northwest	0.20
3	Sierras	0.30
4	California fault areas	0.40

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 5 indicates the greatest chance of an earthquake.

UBC 1997 Seismic Data

Enter data for the UBC 1997 seismic code.

Importance Factor

Enter the value of the importance factor. The software uses this value directly without modification. Values are taken from Table 16-K of the UBC 1997 standard:

Category	Classification	Importance Factor
1	Essential facilities	**1.25**
2	Hazardous facilities	**1.25**
3	Special occupancy structures	**1.0**
4	Standard occupancy structures	**1.0**

Seismic Coefficient Ca

Enter the value of seismic coefficient *CA* according to the project specifications and seismic table 16-Q of UBC 1997. This value is a function of the seismic zone *Z* and the soil profile type.

NOTE In zone 4, *CA* is also a function of near-source factor *Na,* table 16-S of UBC 1997. *Na* is a function of the Seismic Source Type, table 16-U pf UBC 1997.

Table 16-Q

Soil Profile Type	Seismic Coefficient *Ca* for Seismic Zone Factor Z of:				
	Z=0.075	Z=0.15	Z=0.2	Z=0.3	Z=0.4
Sa	**0.06**	**0.12**	**0.16**	**0.24**	**0.32Na**
Sb	**0.08**	**0.15**	**0.20**	**0.30**	**0.40Na**

Sc	**0.09**	**0.18**	**0.24**	**0.33**	**0.40Na**
Sd	**0.12**	**0.22**	**0.28**	**0.36**	**0.44Na**
Se	**0.19**	**0.30**	**0.34**	**0.36**	**0.36Na**
Sf	Site-specific geotechnical investigation and dynamic site response shall be performed to determine seismic coefficients for soil profile Type *Sf*.				

Table 16-S

Seismic Source Type	**Near-Source Factor *Na*[1] for Closest Distance to Known Seismic Source of: [2,3]**		
	<= 2 km	**5 km**	**>=10 km**
A	**1.5**	**1.2**	**1.0**
B	**1.3**	**1.0**	**1.0**
C	**1.0**	**1.0**	**1.0**

Table 16-U

Seismic Source Type[4]	**Seismic Source Description**	**Maximum Moment Magnitude, M**	**Slip Rate, SR (mm/year)**
A	Faults that are capable of producing large Magnitude events and that have a high rate of seismic activity	M>= 7	SR>=5
B	All Faults other than types A and C	M>=7	SR < 5
		M<7	SR>2
		M>=6.5	SR<2
C	Faults that are not capable of producing large magnitude earthquakes and that have a relatively low rate of seismic activity.	M<6.5	SR<=2

NOTES

a. The near-source factor may be based on the linear interpolation of values for distances other than those shown in the table.

b. The location and type of seismic sources to be used for design shall be established on approved geotechnical data (such as the most recent mapping of active faults by the United States Geological Survey or the California Division of Mines and Geology).

c. The closest distance to seismic source shall be taken as the minimum distance between the site and the area described by the vertical projection of the source on the surface (that is, surface projection from fault plane). The surface projection need not include portions of the source at depths of 10 km or greater. The largest value of the near-source factor considering all sources shall be used for design.

d. Subduction sources shall be evaluated on a site specific basis.

Seismic Coefficient Cv

Enter the value of seismic coefficient *CV* according to the project specifications and seismic table 16-R of UBC 1997. This value is a function of the seismic zone *Z* and the soil profile type.

NOTE In zone 4 *CV* is also a function of **Near Source Factor Nv**.

Table 16-R

Soil Profile Type	Seismic Coefficient *Cv* for Seismic Zone Factor Z of:				
	Z=0.075	**Z=0.15**	**Z=0.2**	**Z=0.3**	**Z=0.4**
Sa	**0.06**	**0.12**	**0.16**	**0.24**	**0.32Nv**
Sb	**0.08**	**0.15**	**0.20**	**0.30**	**0.40Nv**
Sc	**0.13**	**0.25**	**0.32**	**0.45**	**0.56Nv**
Sd	**0.18**	**0.32**	**0.40**	**0.54**	**0.64Nv**
Se	**0.26**	**0.50**	**0.64**	**0.84**	**0.96Nv**
Sf	Site-specific geotechnical investigation and dynamic site response shall be performed to determine seismic coefficients for soil profile Type *Sf*.				

Near Source Factor Nv

Enter the value of the near source factor *Nv*, table 16-Q of UBC 1997. This factor is only used in UBC Seismic Zone 4. This value ranges from 1 to 2 and is a function of the distance relative to the seismic source.

NOTE *Nv* is a function of the Seismic Source Type, table 16-U pf UBC 1997.

Table 16-T

Seismic Source Type	Near-Source Factor *Nv*[1] for Closest Distance to Known Seismic Source of: [2,3]			
	<= 2 km	**5 km**	**10 km**	**>=15 km**
A	**2.0**	**1.6**	**1.2**	**1.0**
B	**1.6**	**1.2**	**1.0**	**1.0**

C	**1.0**	**1.0**	**1.0**	**1.0**

Table 16-U

Seismic Source Type[4]	**Seismic Source Description**	**Maximum Moment Magnitude, M**	**Slip Rate, SR (mm/year)**
A	Faults that are capable of producing large Magnitude events and that have a high rate of seismic activity	M>= 7	SR>=5
B	All Faults other than types A and C	M>=7	SR < 5
		M<7	SR>2
		M>=6.5	SR<2
C	Faults that are not capable of producing large magnitude earthquakes and that have a relatively low rate of seismic activity.	M<6.5	SR<=2

NOTES

a. The near-source factor may be based on the linear interpolation of values for distances other than those shown in the table.

b. The location and type of seismic sources to be used for design shall be established on approved geotechnical data (such as the most recent mapping of active faults by the United States Geological Survey or the California Division of Mines and Geology).

c. The closest distance to seismic source shall be taken as the minimum distance between the site and the area described by the vertical projection of the source on the surface (that is, surface projection from fault plane). The surface projection need not include portions of the source at depths of 10 km or greater. The largest value of the near-source factor considering all sources shall be used for design.

d. Subduction sources shall be evaluated on a site specific basis.

Seismic Zone

Select the seismic zone, according to UBC-91 Figure No. 23-2:

Seismic Zone	**Description**	**Seismic Zone Factor Z (Table 23-I)**
0	Gulf and prairies	0.00
1	Rockies and Appalachian areas	0.075
2a	New England, Carolinas, and Ozarks	0.15
2b	Valley area west of the Rockies and	0.20

	the Pacific Northwest	
3	Sierras	0.30
4	California fault areas	0.40

NOTE Zone 0 indicates the least chance of a major earthquake, while Zone 5 indicates the greatest chance of an earthquake.

Force Factor R or Rp

Enter the seismic force factor R according to Table 16-P, UBC 1997. R is defined as the numerical coefficient representative of the inherent overstrength and global ductility of lateral force resisting systems.

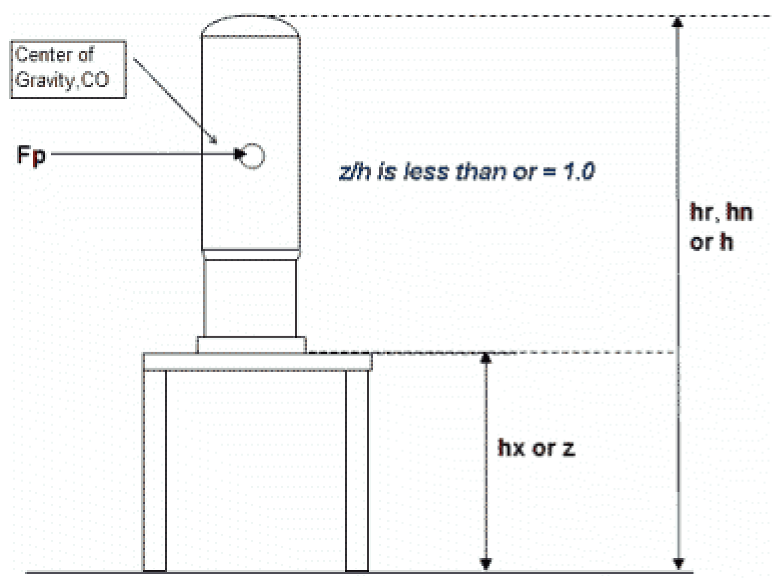
- **2.2** - Tanks on braced or unbraced legs.
- **2.9 -** Distributed mass cantilever structures such as stacks, chimneys, silos, and skirt supported vertical vessels.

Apply Allowables per 1612.3.2

Select to use UBC 1997 provisions for earthquake loadings in paragraph 1612.3.2. Paragraph 1612.3.1 does not allow any increase in allowables, but 1612.3.2, the second (alternate) paragraph does. When this option is selected, the software applies an increase in allowable stresses of 1.33 to the skirt and a value of 1.2 to all other elements. **Apply Allowables per 1612.3.2** also overrides **Use Higher Long. Stress** on the **Design Constraints** tab.

Component Elevation Ratio hx/hr

Enter a value for the elevation ratio if the vessel is attached to another structure, such as a building. The ratio, needed for proper analysis, is the height in the structure where the vessel is attached *hx* to the average height of the roof *hr*. Generally, this value is less than or equal to **1**. For more information, see Section 1632.2, UBC 1997.



Component Amplification Factor ap

Enter a value for the component amplification factor a_p. This value varies from 1.0 to 2.5. For vessels a value of **2.5** is typical.

SECTION 17

PV Elite Analysis

PV Elite performs the analysis of heat exchangers according to the following codes:

- ASME Section VIII Division 1
- TEMA 1998
- PD 5500
- EN-13445 (U-tubes only at the time of this writing)

ASME Tubesheet (heat exchanger) analysis rules were formerly found in Appendix AA, but in 2003, were re-written and moved to the main body of the code, Part UHX. TEMA and PD 5500 methods of analysis have undergone little changes in recent years.

Formerly, tubesheets could only be analyzed using CodeCalc via **Component Analysis** (page 213) . While this command is still available, analysis directly in PV Elite has a number of advantages:

- Tubesheets are integrated into a model with cylindrical shells (main shell and channels), heads, and nozzles.
- The total weight of the heat exchanger is calculated, including all of its component parts.
- Supports, such as saddles, are analyzed directly from the integrated model, ensuring that all weights and applied loads are addressed.
- Tubesheet reports are part of the overall analysis.
- For ASME, the MAWP/MAPnc of the entire exchanger is calculated, including tubesheet, tubes, expansion joint, and floating head. These allowable pressures are calculated for each side shell and channel. The hydrotest for each side can also be calculated.
- The tubesheet design code can be changed between TEMA and ASME with little modification to the input data.

Calculating and Displaying Vessel Analysis Results

Each of these steps calculates and displays specific results of a vessel analysis. A brief description of the key analysis steps is defined below:

Step 0: Error Checking

The input program will have already caught most of the errors that are easily made. However, there are some errors that can only be discovered after the analysis begins. There are also some warnings that may be helpful. This first routine check creates a report in the output. If any of the input errors prevents the software from running, execution stops here. Check the output to determine the exact error discovered by the program.

Step 1: Input Echo

PV Elite provides a complete listing of your input. This includes the geometry and materials for each element (head, shell, cone, flange, skirt, etc.) and the information for any details attached to that element.

Step 2: XY Coordinate Calculations

The program calculates the X and Y locations of the first end of every element.

Step 3: Internal Pressure Calculations

The geometry, material, and loading data from your model are used to calculate the required thickness and maximum allowable working pressure for each element (except skirts and flanges). The calculations are done using the ASME Section VIII, Divisions 1 or 2 rules, or the British Standard PD 5500 rules. The internal design pressure at any point is taken to be the given design pressures for that element, plus the pressure due to liquid head, if any.

If you checked the **Increase Thickness For Internal Pressure** design flag and any element is too thin for the given pressure, the program will automatically (or under interactive control) increase the thickness of the element. There is a computation control (under Utilities on the Main Menu) that allows you to increase the element thickness to exactly that required, or to round the thickness up to the next nominal size.

If the program has increased the thickness, it will recalculate all the required thicknesses and maximum allowable working pressures for the vessel, and create a new table showing these results.

After the internal pressure calculation is complete, PV Elite prints the formulas and substitutions, as well the minimum design metal temperatures for the elements.

Step 4: Hydrotest Calculations

The user specifies what kind of hydrotest (and/or the hydrotest pressure) on the global input screens. The program uses this information to calculate the maximum allowed hydrotest pressure and required thickness at the given pressure for each element.

Step 5: External Pressure Calculations

The user explicitly defines two of the three key variables for external pressure calculations: diameter and thickness. The program calculates the third variable, length of section, for the given geometry. Thus if the vessel has two heads and some number of cylindrical elements with no stiffening rings, the program will calculate the design length for each cylinder using the full length of the vessel plus 1/3 the depth of the heads. If there are stiffening rings, the program will calculate an appropriately shorter value.

The program displays the formulas and substitutions for the external pressure calculations on each element. Then the same results are displayed in tabular form.

If the element is not thick enough for the external pressure (and you checked the design boxes in the input) the program will allow you to increase the thickness and/or add stiffening rings (which are created automatically and added to your model). If the thickness is increased the program has to go back to step 3. For rings it repeats this step with the new lengths.

British Standard PD 5500

When performing the PD 5500 external pressure calculations, the program first computes the length of section for the given geometry. The length of section is either the distance between stiffeners, or, if there are no stiffeners, it is the full length of the vessel plus 0.4 times the depth of the heads.

Using the length of section computed, the program first tests to see whether the thickness of the unsupported cylinder (or distance between supports) is satisfactory for the given pressure. A value of Pmax is determined.

If there are stiffeners, then the program performs the calculations described in section 3.6.2.3. The program first performs the computations described in Method A, and then performs the more rigorous calculations described in Method B. For each of these methods (and each value of n), a value of Pn and Fn are obtained.

Pn is the elastic instability pressure of the stiffened cylinder or cone. The value of Pn must not be less than 1.8*Pext in the case of fabricated or hot formed stiffeners and 2.0*Pext in the case of cold formed stiffeners.

Fn is the maximum stress in the stiffener flange divided by the yield stress of the stiffener. A value for Fn is computed for both fabricated or hot formed stiffeners and cold formed stiffeners. These values must be between 0.0 and 1.0.

Step 6: Weight of Elements

Element weights are calculated in both the corroded and uncorroded conditions. Note that for heads the distance given in the input program is taken as the length of the straight flange on the head. This step also calculates the volume of the element.

Step 7: Weight of Details

Each detail has a separate weight calculation. Of note is the fact that partial volumes of liquid in both the heads and the cylinders and in both the horizontal and vertical directions are correctly calculated.

Step 8: ANSI Flange MAWP

If you entered nozzles, you specified the material and class of the attached flanges. PV Elite has the full ANSI flange tables built in, and tells you the rating of the flanges at the operating temperature.

Steps 9 and 10: Total Weight and Detail Moment

Several weight cases are calculated including: empty, operating, and hydrotest. The various detail weights/loads are included in the following cases:

Detail	Empty	Operating	Hydrotest
Saddle	#	#	#
Platform	#	#	#
Packing	#		

Detail	Empty	Operating	Hydrotest
Liquid	#		
Insulation	#	#	#
Lining	#	#	#
Rings	#	#	#
Nozzles	#	#	#
Saddles	#	#	#
Trays	#		
Legs	#	#	#
Lugs	#	#	#
Weights	#	#	#
Forces/Moments		#	

This step also calculates the moment due to individual details, which may not be on the centerline of the vessel. These are usually small. Finally, this step calculates the forces at the support. The vertical force and bending moment (due to detail weights only) are calculated for the 'one support' case (skirts, legs, lugs) and the vertical force at each support is calculated when there is two saddle supports.

NOTE In addition to computing the above weights PV Elite also computes the fabricated weight, shop test weight, shipping weight, erected weight, empty weight and field test weight. The computed weights may or may not include removable or field installed items such as packing and other details. You can specify where these details are to be installed (either shop or field) in the Global Input. Switch to the global input screen and click the **Installation Miscellaneous Options** button located at the top of the screen. By default the program assumes that all details will be installed in the shop and calculate these various weights based on that assumption.

The cumulative weight on the vessel will look drastically different for horizontal vessels on saddle supports than for vertical vessels on skirts, legs, and lugs:

Horizontal cases: Expect the highest weight forces near the saddles, with almost no weight force at the ends or in the middle.

Vertical cases: Expect the weight forces to increase from zero at the top to a maximum at the support. If there are elements below the support, expect the weight force to be negative.

The cumulative moment includes only the moment due to eccentric details, and is usually quite small (except in the case of a large applied moment).

Step 11: Natural Frequency Calculation

PV Elite uses two classical solution methods to determine the first order natural frequencies of vessels. For vertical vessels, the program uses the Freese method, which is commonly used in industry. For horizontal vessels a similar method attributed to Rayleigh and Ritz is used. Each method works by calculating the static deflection of the vessel (for vertical, the vessel as a horizontal cantilever beam). The natural frequency is proportional to the square root of the deflection. PV Elite uses the matrix solution methods (Eigen Solution) to determine the modes of vibration. Horizontal vessels are assumed to be rigid and as such are assigned a frequency of 33 hertz, which is coincident of a ZPA for a rigid structure.

Step 12: Wind Load Calculation

PV Elite uses the rules of ASCE-7, NBC, UBC, and IS-875 to calculate wind loads. Each of these codes uses a basic wind pressure, a function of the velocity squared, along with several surface and site factors to determine the final wind pressure.

Step 13: Earthquake Load Calculation

The five codes used by the software - ASCE-7, UBC, NBC, IS-1893 RSM and IS-1893 SCM each use a static equivalent load to model the earthquake load. Simple site data and loading data are used to determine an expected static equivalent horizontal load on the vessel.

Step 14: Shear and Bending Moments due to Wind and Earthquake

These loadings generate horizontal loads, which are usually fine on a horizontal vessel, but can cause high overturning moments on a vertical vessel. The program calculates the cumulative shear and bending moment on the vessel, for use in later stress calculations.

Step 15: Wind Deflection

PV Elite calculates the deflection at every point in either horizontal or vertical vessels.

Step 16: Longitudinal Stress Constants

As the program prepares to do structural calculations on the vessel, it first calculates the cross sectional area and section modulus of each element in both the corroded and uncorroded condition.

Step 17: Longitudinal Allowable Stresses

There are four allowable stresses in the longitudinal direction for each element: (1) Longitudinal tension based on the basic allowable stress, often multiplied times 1.2 (as specified on the global input), (2) Hydrotest longitudinal tension - 1.5 times the allowable stress new & cold. (3) Longitudinal compression - based on paragraph UG-23 of the Code, and the material's external pressure chart. (4) Hydrotest allowable compression - the basic allowable compression new & cold, multiplied by 1.5.

Step 18: Longitudinal Stresses Due to . . .

Each load (wind, earthquake, weight, pressure) generates a stress. These are calculated individually and displayed by this routine. Note that bending stresses, though only displayed once, are actually positive on one side of the vessel and negative on the other.

Step 19: Stress Due to Combined Loads

In this step the various load cases combinations defined by the user are evaluated. If there are applied forces and moments in the model, then other identifiers such as BS, BN and so forth may appear in the load case definition.

There can be as many as twenty cases, combining pressure loads, weight loads, and moments in various ways. A fairly complete set of load cases is included as a default:

Load Case	**Definition**
1 NP+EW+WI+FW	No pressure + empty weight + wind
2 NP+EW+EQ+FS	No pressure + empty weight + earthquake
3 NP+OW+WI+FW	No pressure + operating weight + wind
4 NP+OW+EQ+FS	No pressure + operating weight + earthquake
5 NP+HW+HI	No pressure + hydrotest weight + hydro wind
6 NP+HW+HE	No pressure + hydrotest weight + hydro earthquake
7 IP+OW+WI+FW	Internal pressure + operating weight + wind
8 IP+OW+EQ+FS	Internal pressure + operating weight + earthquake
9 EP+OW+WI+FW	External pressure + operating weight + wind
10 EP+OW+EQ+FS	External pressure + operating weight + earthquake
11 HP+HW+HI	Hydrotest pressure + hydrotest weight + hydro wind
12 HP+HW+HE	Hydrotest pressure + hydrotest wind + hydro earthquake
13 IP+WE+EW	Internal pressure + wind empty + empty weight
14 IP+WF+CW	Internal pressure + wind filled + empty weight no ca
15 IP+VO+OW	Internal pressure + vortex shedding (OPE) + operating weight
16 IP+VE+OW	Internal pressure + vortex shedding (EMP) + operating weight
17 IP+VF+CW	Internal pressure+ vortex shedding (Filled) + empty weight no ca

The difference between wind loads and hydrotest wind loads is simply a ratio (percentage) defined by the user. This percentage is specified in the Wind Data definition of Global Data - usually about 33% (thus setting the hydrotest wind load at 33% of the operating wind load). Likewise, the hydrotest earthquake load is a percentage of the earthquake load; this percentage is defined in the Seismic Data definition of Global Data.

Some steps that are not applicable for horizontal vessels, such as natural frequency, will not be printed. Also, if a vessel has no supports, then there will be no calculations that involve wind or seismic loads.

Optional Steps

PV Elite includes the following analyses that are performed under specific circumstances:

1. **Cone Evaluation** - Cones are evaluated for internal and external pressure at the large and small ends, and any stiffening rings near the cones are included and evaluated.
2. **Zick Stresses** - Stresses due to saddle supports are evaluated and compared to allowable stresses using the method of L.P. Zick. Note that the stresses are calculated for each saddle, since in PV Elite each saddle can have different loading. Note also that the stresses are not evaluated at the mid span, because the program automatically does that for all the various load case combinations.
3. **AISC Leg Check:** After the software has computed all of the weights, forces and moments, it can then determine the overall state of stress by using the AISC unity check method. The software typically looks at the worst loads on the legs due to wind or seismic in the operating condition and then applies the AISC method of checking the legs. The unity check must be less than or equal to 1.0. Most typical designs fall in the 0.7 - 0.8 range, which is a good check both in terms of economy and safety.
4. **Lug Support Check -** Similar in manner to the **Leg Check** feature, the software gathers the worst loads on the support lugs and then evaluates them according to a set of acceptable standards. In this case, gussets are checked by the **AISC** method and the lug plates are checked by common industry standard methods. These methods are outlined in common pressure design handbooks.
5. **Baserings -** With known forces and moments at the base and the geometry of the basering, PV Elite will analyze or design the basering and gusset geometry.
6. **Flanges -** For main body flanges, the software computes the required thickness of the flange, all relevant stresses, and MAWP for the given geometry. The results seen in the output are based on the input thickness. The software additionally computes the required thickness of the flange. Please note that the software does not include the forces and moments to determine an equivalent design pressure. There are separate fields in the input that can be entered if these effects are to be considered. In order to do this, two runs would have to be made. After run 1 was made, the forces and moments on the flange could be entered as needed.
7. **Nozzle Analysis -** Complete nozzle evaluation is incorporated into the software based on the rules in the ASME code. Design cases are made for Internal Pressure, External Pressure and MAPnc. The internal pressure can be based on the MAWP of the entire vessel or the exact pressure at the nozzle location. These options are located in the Global Input section of the input. In addition to perpendicular nozzles, hillside geometries are also considered. Nozzles at any angle can be entered by using the ANG=xx.x command in the nozzle description field. The nozzle analysis also computes MDMT, weld size and strength calculations along with provisions for large nozzles as outlined in appendix 1-7 of the ASME Code. Another description option is for small nozzles. If there is a small nozzle that must have area calculations performed, enter the text "#SN" as part of the nozzle description. By default PV Elite will not calculate small openings for Division 1 vessels per UG-36. If local loads have been defined on the nozzle, the nozzle report will display the results from WRC 107 or PD 5500 Annex G, or WRC 297 whichever one was selected.

8. **Fatigue Analysis** - The fatigue analysis is activated when the number of pressure cycles is specified on the **Design/Analysis Constraints** screen. Click the **Perform Fatigue Analysis** button to display the dialog. Change the number of pressure cycles.   This value must be between 1 and 20. This cumulative damage analysis is in accordance with PD 5500 Annex C. In order for this analysis to activate, at least one nozzle must be specified. In the **Nozzle** dialog, there is a check box and a pull down selection menu describing the class of the weld attachment per Annex C. After all of the data is specified, PV Elite produces the **Fatigue Analysis** Report.
9. **Tubesheet Analysis** - When the vessel design code is ASME VIII or PD 5500, tubesheets are allowed to be defined. They can be attached to flange or cylinder parent elements. PV Elite computes tubesheet required thickness, shell and tube stresses per the rules of TEMA, ASME, or PD 5500.
10. **Skirt Hole Opening Analysis** - For vertical skirt supported vessels, PV Elite can compute bending and axial stresses due to missing material in skirt openings typically for pipe openings, vents and access openings.
11. **Half Pipe Jacket Analysis** - If you have specified a helical half pipe jacket, this analysis will be performed per ASME Section VIII, Division 1, Appendix EE or ASME Section VIII, Division 2, Part 4.11.6.
12. **Large Central Opening Analysis** - For welded flat heads, the analysis of large central opening can be performed per ASME Section VIII, Division 1, Appendix 14 or ASME Section VIII, Division 2, Part 4.6.4.
13. **Clip Analysis** - If support clips have been entered in, these items are analyzed using industry standard methods.
14. **Lifting Lugs** - Like clips, the stresses in the shell and lugs are calculated at various angles depending on the final orientation of the vessel.
15. **Tubesheet Analysis** - If the model contains tubesheet data, tubesheet analysis per the chosen Code (TEMA, ASME etc) will be listed in the report output.

Nozzle Analysis

PV Elite calculates required wall thickness and area of reinforcement for a nozzle in a pressure vessel shell or head, and compares this area to the area available in the shell, nozzle and optional reinforcing pad. The software also calculates the strength of failure paths for the nozzles. This calculation is based on the ASME Code, Section VIII, Division 1, Paragraph UG-37 through UG-45. The calculation procedure is based on figure UG-37.1.

The software calculates the required thickness (for reinforcement conditions) based on inside diameter for the following vessel components:

Component	**Paragraph**	**Limitations**
Cylinder	UG-27 (c) (1)	None
2:1 Elliptical Head	UG-32 (d) (1)	None
Torispherical Head	UG-32 (e) (1)	None

Spherical Head or Shell	UG-27 (d) (3)	None

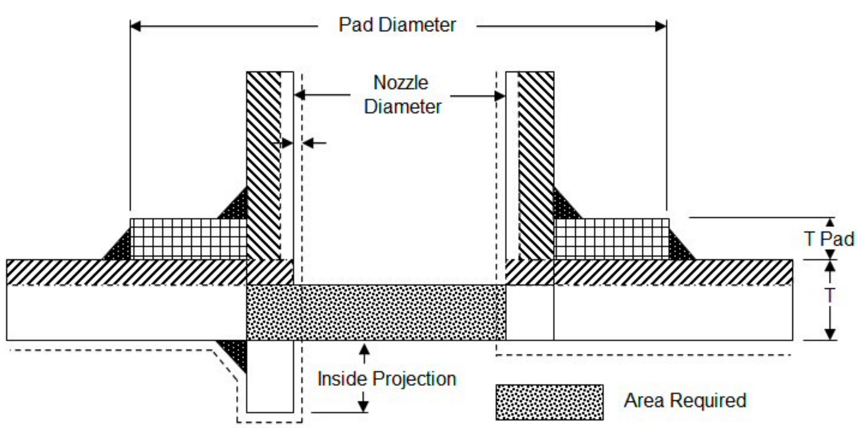
NOTE PV Elite also analyzes a large nozzle in a welded flat head, which is found in this user manual where the flat head is discussed.

The software evaluates nozzles at any angle (less than 90 degrees) away from the perpendicular, allowing evaluation of off angle or hillside nozzles.

Nozzle (page 60) takes full account of corrosion allowance. You enter actual thickness and corrosion allowance, and the software adjusts thicknesses and diameters when making calculations for the corroded condition. **Nozzle** also performs UCS-66 Minimum Design Metal Temperature (MDMT) calculations for nozzles.

As the nozzle data is entered, PV Elite automatically performs the ASME area of replacement or PD 5500/EN-13445 nozzle compensation calculations. A calculation is performed every time the cursor is moved in between input cells. If there is any error in the input that will not allow the analysis to be performed, a status of failed appears at the bottom of the **Nozzle Input/Analysis** dialog box. The calculation is initiated after the pipe size is specified. If you are changing data, such as the pad thickness and are not moving between cells, press **F5** to force PV Elite to re-calculate and display the results. If the calculation has failed, the result will appear in red. A nozzle that has passed will have blue results. The result is typically the area and minimum nozzle overstress per 1-7. The program displays the text failed in brackets, even though the area of replacement may be sufficient. To effectively use this feature, we suggest that the entire vessel be modeled first, along with the liquid and nozzle pressure design options set. Also, for vessels that have ANSI or DIN flanges note that the flange pressure rating will be shown at the bottom of the nozzle dialog.

The figure below displays the nozzle geometry:



SECTION 18

Output Processor

Output from PV Elite analysis is stored in a binary data file that has the same name as the input file but with a ".T80" extension. Use *Review Reports* (page 222) to review every report contained in the output from input echo through stress reports.

Each analysis module creates its own report in the output data file. Most of the reports take the form of tables with the rows related to the elements and the columns holding the values such as thickness, MAWP, and stress.

Output Processor Dialog Box

Create Word Document

Create Microsoft Word document of the selected report.

Create ASME Form

Creates an ASME U-1 or U-2 form for the vessel by creating a Microsoft Excel workbook and running a macro that populates worksheet fields with the calculated results from the .pvu analysis results file. This form is copied from the master template form in the PV Elite\System folder.

After the form is opened in Microsoft Excel, the results data can be transferred into the form by clicking **Import Latest Results** (an Excel macro) at the top of the worksheet.

IMPORTANT Because this is an Excel macro, ensure that macros are allowed to load.

If the analysis is run repeatedly, click **Import Latest Results** again. This overlays the latest results into the spreadsheet. If you have typed non-imported material (like the Inspector's name), this data is not overwritten.

IMPORTANT All information in the form must be correct and match the vessel. Please check the data carefully before submitting the form to the National Board or appropriate authorities. Some form information, such as the drawing number, position, and manufacturer is available by clicking **Tools** > **ASME Form Information** and opening the **Additional Vessel Information** dialog box. For more information, see *Enter in U-1 Form Information for This Vessel* (page 252).

Report Up

Select a report and click to move the report up the list.

Report Down

Select a report and click to move the report down the list.

What do you want to do?

- *Customize report header* (page 545)
- *Customize company name* (page 545)
- *Customize the title page* (page 545)

- *Setting default fonts* (page 546)
- *Save reports to Microsoft Word* (page 546)
- *Save reports to PDF* (page 546)

Customize report header

1. Select the **Home** tab.
2. Click **Input** **> Heading** in the **Input / Output** panel.
3. Enter your header information in Line 1, Line 2, and Line 3 boxes.
4. Click **Save** .

   The custom header is inserted the next time you review results.

   NOTE A header containing the Intergraph and PV Elite trademarks displays when you print an output report or a model of the vessel to PDF. You can customize this header to include your company name and logo, or you can create your own custom header. For more information on how to customize the PDF header, see *PDF Header* (page 351).

Customize company name

1. Using Windows Explorer, navigate to the PV Elite System folder. By default, this location is C:\Users\Public\Documents\Intergraph CAS\PV Elite\2017\System.
2. Using Notepad or another ASCII file editor, edit company.txt.
3. Put your company name in the first line of the file.
4. Save and exit the file.

   The next time you run an analysis, your company name will appear in the report results header.

NOTE This option is only available if you use SmartPlant License Manager.

Customize the title page

1. Using Windows Explorer, navigate to the PV Elite System folder. By default, this location is C:\Users\Public\Documents\Intergraph CAS\PVElite\2017\system.
2. Using Notepad or another ASCII file editor, edit TITLE.HED.
3. Edit the title page text as needed.
4. Save and exit the file.

   The next time you run an analysis, your custom title page text will appear in the Title Page report.

Setting default fonts

1. Click **Review** on the **Home** tab, **Analyze** panel.
2. At the bottom-left of **Output Processor**, select the **Options** tab.

Save reports to Microsoft Word

1. Click **Review** on the **Home** tab, **Analyze** panel.
2. Select the report or reports to output to Microsoft Word.
3. Click **Word Document** .

Save reports to PDF

1. Click **Review** on the **Home** tab, **Analyze** panel on the **Input Processor**.
2. Select the reports to output to PDF from the **Report List** on the **Output Processor**.
3. Click **Preview/Print** on the **File** tab.
4. Click **Print to PDF**.

   Select **Insert 2D Page** or **Insert 3D Page** to include a model of the vessel with the reports.

   NOTES

   - To include a model of the vessel, you must select the **Create 3D PDF Files** icon prior to performing an analysis. For more information about the icon, see *Create 3D PDF Files* (page 220).
   - You can also click **Generate PDF File** to quickly output all the reports to PDF format.

SECTION 19

Material Dialog Boxes

The **Material Database** and **Material Properties** dialog boxes are available in many commands throughout the software.

Material Database Dialog Box (page 547)

Material Properties Dialog Box (page 611)

Material Database Dialog Box

Displays materials and material properties. Select the needed material. To modify material properties, go to the **Tools** tab and select **Edit/Add Materials**.

Below are examples of standard ASME material names.

Plates and Bolting

- SA-516 55
- SA-516 60
- SA-516 65
- SA-516 70
- SA-193 B7
- SA-182-F1
- SA-182 F1
- SA-182 F11
- SA-182 F12
- SA-182 F22
- SA-105
- SA-36
- SA-106 B

Stainless Steels

- SA-240 304
- SA-240 304L
- SA-240 316
- SA-240 316L
- SA-193 B8

Aluminum

- SB-209
- SB-234

Titanium

- SB-265 1
- SB-265 26H

Nickel

- SB-409
- SB-424

NOTE If you used old CodeCalc material names in previous CodeCalc versions, see the CodeCalc appendix for comparisons with ASME code names.

Material Search String

Enter part of the material name to search against.

Find Next Match

Click to go to the next matching material name available.

Color G5 Matls

Click to highlight all G5 materials.

UNS # Search String

Enter part of the UNS # to search against.

Find Next UNS

Click to go to the next matching UNS # available.

Cancel

Exit the dialog box without selecting a material.

Material Database Notes

These notes might not be the most recent version. If using an older database, these notes might not be correct or meaningful as they are periodically changed by ASME. Please refer to official ASME publications.

ASME Section VIII, Division 1 Material Notes for Table 1A (Ferrous Materials) - Customary

(a)	The following abbreviations are used: Norm. rld., Normalized rolled; NT, Normalized and tempered; QT, Quenched and tempered; Smls., Seamless; Sol. ann., Solution annealed; and Wld., Welded.
(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained,; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(c)	For Section VIII and XII applications, stress values in restricted shear such as dowel bolts or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the value in the above table.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in the above Table.
(e)	Stress values for 100°F are applicable for colder temperatures when the toughness requirements of Section III, VIII, or XII are met.
(f)	An alternative typeface is used for stress values obtained from time dependent properties (see notes T1 - T12).

(g)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.
(h)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(i)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings. Notes - General Requirements
G1	To these stress values a casting quality factor as specified in PG-25 of Section I or UG-24 of Section VIII, Division 1, or TM-190 of Section XII shall be applied.
G2	These stress values include a joint efficiency factor of 0.60.
G3	These stress values include a joint efficiency factor of 0.85.
G4	For Section I applications, these stresses apply when used for boiler, water wall, superheater, and economizer tubes that are enclosed within a setting. A joint efficiency factor of 0.85 is included in values above 850ºF.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. For Section III applications, Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in table Y-1, will give allowable stress values that will result in lower values of permanent strain.
G6	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 1500ºF and shall be considered in the design.

G7	For Section VIII applications, these stress values are based on expected minimum values of 45,000 psi tensile strength and yield strength of 20,000 psi resulting from loss of strength due to thermal treatment required for the glass coating operation. UG-85 does not apply.
G8	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the flexibility of the flange and bolts and corresponding relaxation properties.
G9	For Section III applications, the use of these materials shall be limited to materials for tanks covered in Subsections NC and ND, component supports, and for nonpressure-retaining attachments (NC/ND-2190).
G10	Upon prolonged exposure to temperatures above 800ºF, the carbide phase of carbon steel may be converted to graphite. See Nonmandatory Appendix A, A-201 and A-202.
G11	Upon prolonged exposure to temperatures above 875ºF, the carbide phase of carbon–molybdenum steel may be converted to graphite. See Nonmandatory Appendix A, A-201 and A-202.
G12	At temperatures above 1000ºF, these stress values apply only when the carbon is 0.04% or higher on heat analysis.
G13	These stress values at 1050ºF and above shall be used only when the grain size is ASTM No. 6 or coarser.
G14	These stress values shall be used when the grain size is not determined or is determined to be finer than ASTM No. 6.
G15	For Section I applications, use is limited to stays as defined in PG-13 except as permitted by PG-11.
G16	For Section III Class 3 applications, these S values do not include a casting quality factor. Statically and centrifugally cast products meeting the requirements of NC-2570 shall receive a casting quality factor of 1.00.

G17	For Section III Class 3 applications, statically and centrifugally cast products meeting the requirements of NC-2571(a) and (b), and cast pipe fittings, pumps, and valves with inlet piping connections of 2 in. nominal pipe size and less, shall receive a casting quality factor of 1.00. Other casting quality factors shall be in accordance with the following: a. for visual examination, 0.80; b. for magnetic particle examination 0.85; c. for liquid penetrant examination, 0.85; d. for radiography, 1.00; e. for ultrasonic examination, 1.00; and f. for magnetic particle or liquid penetrant plus ultrasonic examination or radiography, 1.00.
G18	See Table Y-1 for yield strength values as a function of thickness over this range. Allowable stresses are independent of yield strength in this thickness range.
G19	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Nonmandatory Appendix A, A-207, and A-208.
G20	These stresses are based on weld metal properties.
G21	For Section I, use is limited to PEB-5.3. See PG-5.5 for cautionary note.
G22	For Section I applications, use of external pressure charts for material in the form of barstock is permitted for stiffening rings only.
G23	For temperatures above the maximum temperature shown on the external pressure chart for this material, Fig. CS-2 may be used for the design using this material. NOTE **PV Elite** automatically uses Fig. CS-2 when the temperature exceeds the limits for a curve using this material. If you do not want to use the CS-2 calculation, you can type another value into the **External Pressure Curve** box in the material properties, click **OK**, and then re-analyze the job.
G24	A factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.
G25	For Section III applications, for both Class 2 and Class 3, the completed vessel after final heat treatment shall be examined by the ultrasonic method in accordance with NB-2542 except that angle beam examination in both the circumferential and the axial directions may be performed in lieu of the straight beam examination in the axial direction. The tensile strength shall not exceed 125,000 psi.

G26	Material that conforms to Class 10, 11, or 12 is not permitted.
G27	Material that conforms to Class 11 or 12 is not permitted.
G28	Supplementary Requirement S15 of SA-781, Alternate Mechanical Test Coupons and Specimen Locations for Castings, is mandatory.
G29	For Section III applications, impact testing in accordance with the requirements of NC-2300 is required for Class 2 components and in accordance with ND-2300 for Class 3 components.
G30	These stresses apply to all product forms (C, H, and P) as defined in SA/EN 10028–7. Notes – Heat Treatment Requirements
H1	For temperatures above 1000ºF, these stress values may be used only if the material is heat treated by heating to the minimum temperature specified in the material specification, but not lower than 1900ºF, and quenching in water or rapidly cooling by other means.
H2	For temperatures above 1000ºF, these stress values may be used only if the material is heat treated by heating to a minimum temperature of 2000ºF, and quenching in water or rapidly cooling by other means.
H3	Deleted.
H4	Solution treated and quenched.
H5	For Section III applications, if heat treatment is performed after forming or fabrication, it shall be performed at 1500ºF to 1850ºF for a period of time not to exceed 10 min at temperature, followed by rapid cooling.
H6	Material shall be solution annealed at 2010ºF to 2140ºF, followed by a rapid cooling in water or air.
H7	Quenched and tempered at 1200°F. Notes - Size Requirements
S1	For Section I applications, stress values at temperatures of 850ºF and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S2	For Section I applications, stress values at temperatures of 900ºF and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S3	For Section I applications, stress values at temperatures of 1000ºF and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.

S4	For Section I applications, stress values at temperatures of 1150ºF and above are permissible but, except for tubular products 3 in. O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S5	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds ¾ in.
S6	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds 1-¼ in.
S7	The maximum thickness of unheat-treated forgings shall not exceed 3-¾ in. The maximum thickness as-heat-treated may be 4 in.
S8	The maximum section thickness shall not exceed 3 in. for double-normalized-and-tempered forgings, or 5 in. for quenched-and-tempered forgings.
S9	Both NPS 8 and larger, and schedule 140 and heavier.
S10	The maximum pipe size shall be NPS 4 (DN 100) and the maximum thickness in any pipe size shall be Schedule 80.
S11	Either NPS 8 and larger and less than schedule 140 wall, or less than NPS 8 and all wall thicknesses. Notes - Time-Dependent Properties [See General Note (f)]
T1	Allowable stresses for temperatures of 700ºF and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 750ºF and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 850ºF and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 900ºF and above are values obtained from time-dependent properties.
T5	Allowable stresses for temperatures of 950ºF and above are values obtained from time-dependent properties.
T6	Allowable stresses for temperatures of 1000ºF and above are values obtained from time-dependent properties.
T7	Allowable stresses for temperatures of 1050ºF and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 1100ºF and above are values obtained from time-dependent properties.

T9	Allowable stresses for temperatures of 1150ºF and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 800ºF and above are values obtained from time-dependent properties.
T11	Allowable stresses for temperatures of 650ºF and above are values obtained from time-dependent properties.
T12	Allowable stresses for temperatures of 1200ºF and above are values obtained from time-dependent properties. Notes - Welding Requirements
W1	Not for welded construction.
W2	Not for welded construction in Section III.
W3	Welded.
W4	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 100 ksi.
W5	Welded, with the tensile strength of the Section IX reduced tension test less than 100 ksi but not less than 95 ksi.
W6	This material may be welded by the resistance technique.
W7	In welded construction for temperatures above 850ºF, the weld metal shall have a carbon content of greater than 0.05%.
W8	Welding and oxygen or other thermal cutting processes are not permitted when carbon content exceeds 0.35% by heat analysis.
W9	For Section I applications, for pressure retaining welds in 2-¼Cr–1Mo materials, other than circumferential butt welds less than or equal to 3-½ in. in outside diameter, when the design metal temperatures exceed 850ºF, the weld metal shall have a carbon content greater than 0.05%.
W10	For Section III applications, material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted for Class 2 and Class 3 construction when a weld efficiency factor of 1.00 is used in accordance with Note W12.
W11	For Section VIII applications, Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Part UF.

W12	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC-2550, shall provide a longitudinal weld efficiency factor of 1.0. Materials welded with filler metal meeting the requirements of NC-2560 shall receive a longitudinal weld efficiency factor of 1.00. Other long. weld efficiency factors shall be in accordance with the following: a. for single butt weld, with filler metal, 0.80; b. for single or double butt weld, without filler metal, 0.85; c. for double butt weld, with filler metal, 0.90; d. for single or double butt weld, with radiography, 1.00.
W13	For Section I applications, electric resistance and autogenous welded tubing may be used with these stresses, provided the following additional restrictions and requirements are met: a. The tubing shall be used for boiler, waterwall, superheater, and economizer tubes that are enclosed within the setting. b. The maximum outside diameter shell be 3.5 in. c. The weld seam of each tube shall be subjected to an angle beam ultrasonic inspection per SA-450. d. A complete volumetric inspection of the entire length of each tube shall be performed in accordance with SA-450. e. Material test reports shall be supplied.
W14	These S values do not include a weld factor. For Section VIII Division 1, and Section XII applications using welds made without filler metal, the tabulated tensile strength values should be multiplied by 0.85. For welds made with filler metal, check UW-12 of Section VIII Division 1 or TW-130.4 for Section XII, as applicable.
W15	The Nondestructive Electric Test requirements of SA-53 Type E pipe are required for all sizes. The pipe shall be additionally marked "NDE" and so noted on the material certification.

ASME Section VIII, Division 1 Material Notes for Table 1A (Ferrous Materials) - Metric

(a)	The following abbreviations are used: Norm. rld., Normalized rolled; NT, Normalized and tempered; QT, Quenched and tempered; Smls., Seamless; and Wld., Welded.

(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(c)	For Section VIII and XII applications, stress values in restricted shear such as dowel bolts or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the values in Division 1 Material Notes for Table 1A (Ferrous Materials) - Customary.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in Division 1 Material Notes for Table 1A (Ferrous Materials) - Customary.
(e)	Stress values for 40°C are applicable for colder temperatures when the toughness requirements of Section III, VIII, or XII are met.
(f)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T12).
(g)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.
(h)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(i)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings.
G1	To these stress values a casting quality factor as specified in PG–25 of Section I; UG–24 of Section VIII, Division 1; or TM–190 of Section XII shall be applied.
G2	These stress values include a joint efficiency factor of 0.60.

G3	These stress values include a joint efficiency factor of 0.85.
G4	For Section I applications, these stresses apply when used for boiler, water wall, superheater, and economizer tubes that are enclosed within a setting. A joint efficiency factor of 0.85 is included in values above 450ºC.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short– time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 662/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. For Section III applications, Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower levels of permanent strain
G6	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 825ºC and shall be considered in the design.
G7	For Section VIII applications, these stress values are based on expected minimum values of 310 MPa tensile strength and yield strength of 140 MPa resulting from loss of strength due to thermal treatment required for the glass coating operation. UG-85 does not apply.
G8	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the flexibility of the flange and bolts and corresponding relaxation properties.
G9	For Section III applications, the use of these materials shall be limited to materials for tanks covered in Subsections NC and ND, component supports, and for nonpressure-retaining attachments (NC/ND-2190).
G10	Upon prolonged exposure to temperatures above 425ºC, the carbide phase of carbon steel may be converted to graphite. See Nonmandatory Appendix A, A-201 and A-202.
G11	Upon prolonged exposure to temperatures above 475ºC, the carbide phase of carbon–molybdenum steel may be converted to graphite. See Nonmandatory Appendix A, A-201 and A-202.

G12	At temperatures above 550ºC, these stress values apply only when the carbon is 0.04% or higher on heat analysis.
G13	These stress values at 575ºC and above shall be used only when the grain size is ASTM No. 6 or coarser.
G14	These stress values shall be used when the grain size is not determined or is determined to be finer than ASTM No. 6.
G15	For Section I applications, use is limited to stays as defined in PG-13 except as permitted by PG-11.
G16	For Section III Class 3 applications, these S values do not include a casting quality factor. Statically and centrifugally cast products meeting the requirements of NC-2570 shall receive a casting quality factor of 1.00.
G17	For Section III Class 3 applications, statically and centrifugally cast products meeting the requirements of NC–2571(a) and (b), and cast pipe fittings, pumps, and valves with inlet piping connections of DN 50 and less, shall receive a casting quality factor of 1.00. Other casting quality factors shall be in accordance with the following: (a) for visual examination, 0.80 (b) for magnetic particle examination, 0.85 (c) for liquid penetrant examination, 0.85 (d) for radiography, 1.00 (e) for ultrasonic examination, 1.00 (f) for magnetic particle or liquid penetrant plus ultrasonic examination or radiography, 1.00
G18	See Table Y-1 for yield strength values as a function of thickness over this range. Allowable stresses are independent of yield strength in this thickness range.
G19	This steel may be expected to develop embrittlement after service at a moderately elevated temperature. See Nonmandatory Appendix A, A-207 and A-208.
G20	These stresses are based on weld metal properties.
G21	For Section 1, use is limited to PEB-5.3. See PG-5.5 for cautionary note.
G22	For Section I applications, use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G23	For temperatures above the maximum temperature shown on the external pressure chart for this material, Fig. CS-2 may be used for the design using this material.

G24	A factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.
G25	For Section III applications, for both Class 2 and Class 3, the completed vessel after final heat treatment shall be examined by the ultrasonic method in accordance with NB-2542 except that angle beam examination in both the circumferential and the axial directions may be performed in lieu of the straight beam examination in the axial direction. The tensile strength does not exceed 860 MPa.
G26	Material that conforms to Class 10, 11, or 12 is not permitted.
G27	Material that conforms to Class 11 or 12 is not permitted.
G28	Supplementary Requirement S15 of SA-781, Alternate Mechanical Test Coupons and Specimen Locations for Castings, is mandatory.
G29	For Section III applications, impact testing in accordance with the requirements of NC-2300 is required for Class 2 components and in accordance with ND-2300 for Class 3 components.
G30	These stresses apply to all product forms (C, H, and P) as defined in SA/EN 10028-7. NOTES - HEAT TREATMENT REQUIREMENTS
H1	For temperatures above 550ºC, these stress values may be used only if the material is heat treated by heating to the minimum temperature specified in the material specification, but not lower than 1040ºC, and quenching in water or rapidly cooling by other means.
H2	For temperatures above 550ºC, these stress values may be used only if the material is heat treated by heating to a minimum temperature of 1095ºC, and quenching in water or rapidly cooling by other means.
H3	DELETED
H4	Solution treated and quenched.
H5	For Section III applications, if heat treatment is performed after forming or fabrication, it shall be performed at 825ºC to 1000ºC for a period of time not to exceed 10 min at temperature, followed by rapid cooling.
S1	For Section I applications, stress values at temperatures of 450ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.

S2	For Section I applications, stress values at temperatures of 475ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S3	For Section I applications, stress values at temperatures of 550ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S4	For Section I applications, stress values at temperatures of 625ºC and above are permissible but, except for tubular products 75 mm O.D. or less enclosed within the boiler setting, use of these materials at these temperatures is not current practice.
S5	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds19 mm.
S6	Material that conforms to Class 10, 11, or 12 is not permitted when the nominal thickness of the material exceeds 32 mm.
S7	The maximum thickness of unheat-treated forgings does exceed 95 mm. The maximum thickness as-heat-treated may be 100 mm.
S8	The maximum section thickness does exceed 75 mm for double-normalized-and-tempered forgings, or 125 mm for quenched-and-tempered forgings.
S9	Both DN 200 and larger, and schedule 140 and heavier.
S10	The maximum pipe size is shall be DN 100 and the maximum thickness in any pipe size shall be Schedule 80.
S11	Either DN 200 and larger and less than schedule 140 wall, or less than DN 200 and all wall thicknesses. NOTES -TIME-DEPENDENT PROPERTIES [See General Note (f)]
T1	Allowable stresses for temperatures of 370ºC and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 400ºC and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 455ºC and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 480ºC and above are values obtained from time-dependent properties.

T5	Allowable stresses for temperatures of 510ºC and above are values obtained from time-dependent properties.
T6	Allowable stresses for temperatures of 540ºC and above are values obtained from time-dependent properties.
T7	Allowable stresses for temperatures of 565ºC and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 595ºC and above are values obtained from time-dependent properties.
T9	Allowable stresses for temperatures of 620ºC and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 425ºC and above are values obtained from time-dependent properties.
T11	Allowable stresses for temperatures of 350ºC and above are values obtained from time-dependent properties.
T12	Allowable stresses for temperatures of 650ºC and above are values obtained from time-dependent properties. Notes - Welding Requirements
W1	Not for welded construction.
W2	Not for welded construction in Section III.
W3	Welded.
W4	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 690 MPa.
W5	Welded, with the tensile strength of the Section IX reduced tension test less than 690 MPa but not less than 655 MPa.
W6	This material may be welded by the resistance technique.
W7	In welded construction for temperatures above 450ºC, the weld metal has a carbon content of greater than 0.05%.
W8	Welding and oxygen or other thermal cutting processes are not permitted when carbon content exceeds 0.35% by heat analysis.
W9	For Section I applications, for pressure retaining welds in 2¼Cr–1Mo materials, other than circumferential butt welds less than or equal to 89 mm in outside diameter, when the design metal temperatures exceed 450ºC, the weld metal has a carbon content greater than 0.05%.

W10	For Section III applications, material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted for Class 2 and Class 3 construction when a weld efficiency factor of 1.00 is used in accordance with Note W12.
W11	For Section VIII applications, Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Part UF.
W12	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC-2550, shall provide a longitudinal weld efficiency factor of 1.0. Materials welded with filler metal meeting the requirements of NC-2560 shall receive a longitudinal weld efficiency factor of 1.00. Other long. weld efficiency factors shall be in accordance with the following: ▪ For single butt weld, with filler metal, 0.80. ▪ For single or double butt weld, without filler metal, 0.85. ▪ For double butt weld, with filler metal, 0.90. ▪ For single or double butt weld, with radiography, 1.00.
W13	For Section I applications, electric resistance and autogenous welded tubing may be used with these stresses, provided the following additional restrictions and requirements are met: ▪ The tubing is used for boiler, waterwall, superheater, and economizer tubes that are enclosed within the setting. ▪ The maximum outside diameter is 89 mm. ▪ The weld seam of each tube is subjected to an angle beam ultrasonic inspection per SA-450. ▪ A complete volumetric inspection of the entire length of each tube is performed in accordance with SA-450. ▪ Material test reports are supplied.
W14	These S values do not include a weld factor. For Section VIII, Division 1 and Section XII applications using welds made without filler metal, the tabulated tensile strength values should be multiplied by 0.85. For welds made with filler metal, consult UW-12 of Section VIII, Division 1, or TW-130.4 for Section XII, as applicable.
W15	The Nondestructive Electric Test requirements of SA-53 Type E pipe are required for all sizes. The pipe shall be additionally marked "NDE" and so noted on the material certification.

ASME Section VIII, Division 1 Material Notes for Table 1B (Non-Ferrous Materials) - Customary

(a)	The following abbreviations are used: ann., annealed; cond., condenser; CW, cold worked; exch., exchanger; extr., extruded; fin., finished; fr., from; HW, Hot worked; rel., relieved; rld., rolled; Smls., Seamless; Sol., Solution; treat., treated; and Wld., Welded.
(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(c)	For Section VIII and XII applications, stress values in restricted shear, such as dowel bolts, rivets, or similar construction in which the shearing is so restricted that the section under consideration would fail without reduction of areas, shall be 0.80 times the values in this table.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in this Table.
(e)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T19).
(f)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 shall be used when SB-407M Grade N08800 is used in construction.
(g)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(h)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings. Notes - General Requirements
G1	For steam at 250 psi (406F), the values given for 400F may be used.
G2	At temperatures over 1000F, these stress values apply only when the carbon is 0.04% or higher.

G3	In the absence of evidence that the casting is of high quality throughout, values not in excess of 80% of those given in the Table shall be used. This is not intended to apply to valves and fittings made to recognized standards.
G4	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 1500F and shall be considered in the design.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. For Section III applications, Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in table Y-1, will give allowable stress values that will result in lower values of permanent strain.
G6	Maximum temperature for external pressure not to exceed 350F.
G7	Use 350F curve for all temperature values below 350F.
G8	The stresses for this material are based on 120 ksi minimum tensile strength because of weld metal strength limitations.
G9	Use Fig. NFC-6 up to and including 300F. Use the 600F curve of Fig. NFC-3 above 300F up to and including 400F. Maximum temperature for external pressure not to exceed 400F.
G10	Maximum temperature for external pressure not to exceed 450F.
G11	Referenced external pressure chart is applicable up to 700F.
G12	Referenced external pressure chart is applicable up to 800F.
G13	For Section VIII and XII applications, use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G14	For Section VIII applications, a factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.

G15	To these stress values a quality factor as specified in ND-3115 of Section III; UG-24 of Section VIII, Division 1; or TM-190 of Section XII shall be applied for castings. This is not intended to apply to valves and fittings made to recognized standards.
G16	Allowable stress values shown are 90% of those for the corresponding core material.
G17	Copper-silicon alloys are not always suitable when exposed to certain media and high temperatures, particularly steam above 212°F. The user should ensure that the alloy selected is satisfactory for the service for which it is to be used.
G18	Because of the occasionally contingent danger from the failure of pressure vessels by stress corrosion cracking, the following is pertinent. These materials are suitable for engineering use under a wide variety of ordinary corrosive conditions with no particular hazard in respect to stress corrosion.
G19	Few alloys are completely immune to stress corrosion cracking in all combinations of stress and corrosive environments and the supplier of the material should be consulted. Reference may also be made to the following sources: (1) Stress Corrosion Cracking Control Measures B.F. Brown, U.S. National Bureau of Standards (1977), available from NACE, Texas; (2) The Stress Corrosion of Metals, H.L. Logan, John Wiley and Sons, New York, 1966.
G20	For plate only.
G21	The maximum operating temperature is arbitrarily set at 500°F because harder temper adversely affects design stress in the creep rupture temperature range.
G22	The minimum tensile strength of reduced tension specimens in accordance with QW-462.1 of Section IX shall not be less than 110,000 psi.
G23	This alloy is subject to severe loss of impact strength at room temperature after exposure in the range of 1000°F to 1400°F.
G24	For stress relieved tempers (T351, T3510, T3511, T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper shall be used.
G25	The tension test specimen from plate 0.500 in. and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values for thickness less than 0.500 in. shall be used.

G26	The tension test specimen from plate 0.500 in. and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values shown are 90% of those for the core material of the same thickness.
G27	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 1000°F to 1250°F.
G28	For external pressure design, the maximum design temperature is limited to 1000°F.
G29	External pressure chart NFN-2 may be used for temperatures between 400°F and 600°F.
G30	Allow N06025 in the solution annealed condition is subject to severe loss of rupture ductility in the approximate temperature range of 1200°F to 1400°F.
G31	For external pressure design, the maximum design temperature is limited to 1200°F.
G32	For Section I use, the y values (see Section 1, PG-27.4.6) shall be as follows: for 1050°F and below, 0.4; for 1100°F, 0.5; and for 1150°F and above, 0.7.
G33	Allowable stress values listed are set equal to those of an annealed temper, as data were not provided to justify higher values.
H1	For temperatures above 1000°F, these stress values may be used only if the material is annealed at a minimum temperature of 1900°F and has a carbon content of 0.04% or higher.
H2	For temperatures above 1000°F, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1900°F and quenching in water or rapidly cooling by other means.
H3	For Section I applications, cold drawn pipe or tube shall be annealed at 1900°F minimum.
H4	The material shall be given a 1725°F to 1825°F stabilizing heat treatment.
T1	Allowable stresses for temperatures of 250°F and above are values obtained from time-dependent properties.
T2	Allowable stresses for temperatures of 300°F and above are values obtained from time dependent properties.

T3	Allowable stresses for temperatures of 350°F and above are values obtained from time dependent properties.
T4	Allowable stresses for temperatures of 400°F and above are values obtained from time dependent properties.
T5	Allowable stresses for temperatures of 500°F and above are values obtained from time dependent properties.
T6	Allowable stresses for temperatures of 550°F and above are values obtained from time dependent properties.
T7	Allowable stresses for temperatures of 600°F and above are values obtained from time dependent properties.
T8	Allowable stresses for temperatures of 750°F and above are values obtained from time dependent properties.
T9	Allowable stresses for temperatures of 800°F and above are values obtained from time dependent properties.
T10	Allowable stresses for temperatures of 850°F and above are values obtained from time dependent properties.
T11	Allowable stresses for temperatures of 900°F and above are values obtained from time dependent properties.
T12	Allowable stresses for temperatures of 950°F and above are values obtained from time dependent properties.
T13	Allowable stresses for temperatures of 1000°F and above are values obtained from time dependent properties.
T14	Allowable stresses for temperatures of 1050°F and above are values obtained from time dependent properties.
T15	Allowable stresses for temperatures of 1100°F and above are values obtained from time dependent properties.
T16	Allowable stresses for temperatures of 1150°F and above are values obtained from time dependent properties.
T17	Allowable stresses for temperatures of 1200°F and above are values obtained from time dependent properties.
T18	Allowable stresses for temperatures of 1250°F and above are values obtained from time dependent properties.
T19	Allowable stresses for temperatures of 450°F and above are values obtained from time dependent properties.

W1	No welding or brazing permitted.
W2	For Section VIII applications, UNF-56(d) shall apply for welded constructions.
W3	For welded and brazed constructions, stress values for O (annealed) temper material shall be used.
W4	The stress values given for this material are not applicable when either welding or thermal cutting is employed.
W5	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC-2550, shall provide a longitudinal weld efficiency factor of 1.00. Materials welded with filler metal meeting the requirements of NC-2560 shall receive a longitudinal weld efficiency factor of 1.00. Other long. weld efficiency factors shall be in accordance with the following: a. for single butt weld, with filler metal, 0.80; b. for single or double butt weld, without filler metal, 0.85; c. for double butt weld, with filler metal, 0.90; d. for single or double butt weld, with radiography or ultrasonic, 1.00.
W6	Filler metal shall not be used in the manufacture of welded pipe or tubing.
W7	Strength of reduced-section tensile specimen required to qualify welding procedures. See QW-150, Section IX.
W8	After welding, heat treat at 1150°F to 1200°F, hold 1-½ hr at temperature for the first inch of cross-section thickness and ½ hr for each additional inch, and air cool. For castings used in pumps, valves, and fittings 2 in. nominal pipe size and less, PWHT is not required for socket welds and attachment welds when the castings have been temper annealed at 1150° to 1200°F prior to welding.
W9	Deleted.
W10	Deleted.
W11	These maximum allowable stress values are to be used in welded or brazed constructions.

W12	These S values do not include a weld factor. For Section VIII, Division 1 applications using welds made without filler metal, the tabulated tensile stress values shall be multiplied by 0.85. For welds made with filler metal, consult UW-12 of Section VIII, Division 1, or TW-130.4 of Section XII, as applicable.
W13	For service at 1200°F or higher, the deposited weld metal shall be of the same nominal chemistry as the base metal.
W14	No welding permitted.
W15	For Section VIII and XII applications, no welding is permitted.
W16	Use NFA-12 when welded with 5356 or 5556 filler metal, all thickness, or 4043 or 5554 filler metal, thickness <= 3/8 in. Use NFA-13 when welded with 4043 or 5554 filler metal, thickness > 3/8 in.
W17	For welded and brazed constructions, stress values for the welded and annealed (WO61) temper material shall be used.

ASME Section VIII, Division 1 Material Notes for Table 1B (Non-Ferrous Materials) - Metric

(a)	The following abbreviations are used: ann., annealed; cond., condenser; CW, cold worked; exch., exchanger; extr., extruded; fin., finished; fr., from; HW, Hot worked; rel., relieved; rld., rolled; Smls., Seamless; Sol., Solution; treat., treated; and Wld., Welded.
(b)	The stress values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(c)	For Section VIII and XII applications, stress values in restricted shear, such as dowel bolts, rivets, or similar construction in which the shearing is so restricted that the section under consideration would fail without reduction of areas, shall be 0.80 times the values in this Table.
(d)	For Section VIII and XII applications, stress values in bearing shall be 1.60 times the values in this Table.
(e)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1-T20).

(f)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 are used when SB-407M Grade N08800 is used in construction.
(g)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(h)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross–section of other pressure parts, e.g., castings and forgings. Notes – General Requirements
G1	For steam at 1700 kPa (208ºC), the values given for 200ºC may be used.
G2	At temperatures over 550ºC, these stress values apply only when the carbon is 0.04% or higher.
G3	In the absence of evidence that the casting is of high quality throughout, values not in excess of 80% of those given in the Table are used. This is not intended to apply to valves and fittings made to recognized standards.
G4	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 825ºC and shall be considered in the design.
G5	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short– time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 662/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. For Section III applications, Table Y–2 lists multiplying factors that, when applied to the yield strength values shown in Table Y–1, will give allowable stress values that will result in lower levels of permanent strain.
G6	Maximum temperature for external pressure not to exceed 175ºC.

G7	Use 175ºC curve for all temperature values below 175ºC.
G8	The stresses for this material are based on 828 MPa minimum tensile strength because of weld metal strength limitations.
G9	Use Fig. NFC-6 up to and including 150ºC. Use the 315ºC curve of Fig. NFC-3 above 150ºC up to and including 200ºC. Maximum temperature for external pressure not to exceed 200ºC.
G10	Maximum temperature for external pressure does not exceed 225ºC.
G11	Referenced external pressure chart is applicable up to 375ºC.
G12	Referenced external pressure chart is applicable up to 425ºC.
G13	For Section VIII and XII applications, use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G14	For Section VIII applications, a factor of 0.85 has been applied in arriving at the maximum allowable stress values in tension for this material. Divide tabulated values by 0.85 for maximum allowable longitudinal tensile stress.
G15	To these stress values a quality factor as specified in ND-3115 of Section III; UG-24 of Section VIII, Division 1; or TM-190 of Section XII shall be applied for castings. This is not intended to apply to valves and fittings made to recognized standards.
G16	Allowable stress values shown are 90% of those for the corresponding core material.
G17	Copper-silicon alloys are not always suitable when exposed to certain media and high temperatures, particularly steam above 100ºC. The user should ensure that the alloy selected is satisfactory for the service for which it is to be used.
G18	Because of the occasionally contingent danger from the failure of pressure vessels by stress corrosion cracking, the following is pertinent. These materials are suitable for engineering use under a wide variety of ordinary corrosive conditions with no particular hazard in respect to stress corrosion.
G19	Few alloys are completely immune to stress corrosion cracking in all combinations of stress and corrosive environments and the supplier of the material should be consulted. Reference may also be made to the following sources: (1) Stress Corrosion Cracking Control Measures B.F. Brown, U.S. National Bureau of Standards (1977), available from NACE, Texas; (2) The Stress Corrosion of Metals, H.L. Logan, John Wiley and Sons, New York, 1966.

G20	For plate only.
G21	The maximum operating temperature is arbitrarily set at 250ºC because harder temper adversely affects design stress in the creep rupture temperature range.
G22	The minimum tensile strength of reduced tension specimens in accordance with QW-462.1 of Section IX is not less than 760 MPa.
G23	This alloy is subject to severe loss of impact strength at room temperature after exposure in the range of 550ºC to 750ºC.
G24	For stress relieved tempers (T351, T3510, T3511, T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper are used.
G25	The tension test specimen from plate 13 mm and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values for thickness less than 13 mm are used.
G26	The tension test specimen from plate 13 mm and thicker is machined from the core and does not include the cladding alloy; therefore, the allowable stress values shown are 90% of those for the core material of the same thickness.
G27	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 550ºC to 675ºC.
G28	For external pressure design, the maximum design temperature is limited to 550ºC.
G29	The maximum allowable stress values for greater than 900ºC are 9.7 MPa (927ºC), 7.6 MPa (954ºC), and 5.0 MPa (982ºC).
G30	The maximum allowable stress values for greater than 900ºC are 5.0 MPa (925ºC), 4.0 MPa (950ºC), 3.2 MPa (975ºC), and 2.6 MPa (1000ºC). The maximum use temperature is 982ºC; the value listed at 1000ºC is provided for interpolation purposes only.
G31	The maximum allowable stress values for greater than 900°C are 7.8 MPa (925°C), 5.2 MPa (950°C), 3.5 MPa (975°C), and 2.4 MPa (1000°C). The maximum use temperature is 982°C; the value listed at 1000°C is provided for interpolation purposes only.
G32	The maximum allowable stress values for greater than 900°C are 6.6 MPa (925°C), 4.4 MPa (950°C), 2.9 MPa (975°C), and 2.0 MPa (1000°C). The maximum use temperature is 982°C; the value listed at 1000°C is provided for interpolation purposes only.

G33	External pressure chart NFN-2 may be used for temperatures between 205°C and 315°C.
G34	Allow N06025 in the solution annealed condition is subject to severe loss of rupture ductility in the approximate temperature range of 650°C to 760°C.
G35	For external pressure design, the maximum design temperature is limited to 650°C.
G36	For Section I use, the *y* values (see Section I, PG-27.4.6) shall be as follows: for 566°C and below, 0.4; for 593°C, 0.5; and for 621°C and above, 0.7.
G37	The maximum allowable stress values for greater than 900°C are 3.4 MPa (925°C), 2.6 MPa (950°C), 2.6 MPa (975°C), and 2.3 MPa (1000°C). The maximum use temperature is 982°C; the value listed at 1000°C is provided for interpolation purposes only.
G38	The maximum allowable stress values for greater than 900°C are 2.9 MPa (925°C), 2.5 MPa (950°C), 2.2 MPa (975°C), and 2.0 MPa (1000°C). The maximum use temperature is 982°C; the value listed at 1000°C is provided for interpolation purposes only.
G39	The maximum allowable stress value at 204°C is 29.6 MPa.
G40	The maximum allowable stress value at 204°C is 17.9 MPa.
G41	Allowable stress values listed are set equal to those of an annealed temper, as data were not provided to justify higher values. Notes - Heat Treatment Requirements
H1	For temperatures above 550ºC, these stress values may be used only if the material is annealed at a minimum temperature of 1040ºC and has a carbon content of 0.04% or higher.
H2	For temperatures above 550ºC, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1040ºC and quenching in water or rapidly cooling by other means.
H3	For Section I applications, cold drawn pipe and tube shall be annealed at 1038ºC minimum.
H4	The material shall be given a 940ºC to 995ºC stabilizing heat treatment. Notes – Time–Dependent Properties [See General Note (e)]
T1	Allowable stresses for temperatures of 125ºC and above are values obtained from time-dependent properties.

T2	Allowable stresses for temperatures of 150ºC and above are values obtained from time dependent properties.
T3	Allowable stresses for temperatures of 175ºC and above are values obtained from time dependent properties.
T4	Allowable stresses for temperatures of 205ºC and above are values obtained from time dependent properties.
T5	Allowable stresses for temperatures of 260ºC and above are values obtained from time dependent properties.
T6	Allowable stresses for temperatures of 290ºC and above are values obtained from time dependent properties.
T7	Allowable stresses for temperatures of 315ºC and above are values obtained from time dependent properties.
T8	Allowable stresses for temperatures of 400ºC and above are values obtained from time dependent properties.
T9	Allowable stresses for temperatures of 425ºC and above are values obtained from time dependent properties.
T10	Allowable stresses for temperatures of 455ºC and above are values obtained from time dependent properties.
T11	Allowable stresses for temperatures of 480ºC and above are values obtained from time dependent properties.
T12	Allowable stresses for temperatures of 510ºC and above are values obtained from time dependent properties.
T13	Allowable stresses for temperatures of 540ºC and above are values obtained from time dependent properties.
T14	Allowable stresses for temperatures of 565ºC and above are values obtained from time dependent properties.
T15	Allowable stresses for temperatures of 595ºC and above are values obtained from time dependent properties.
T16	Allowable stresses for temperatures of 620ºC and above are values obtained from time dependent properties.
T17	Allowable stresses for temperatures of 650ºC and above are values obtained from time dependent properties.
T18	Allowable stresses for temperatures of 675ºC and above are values obtained from time dependent properties.

T19	Allowable stresses for temperatures of 450°C and above are values obtained from time dependent properties.
T20	Allowable stresses for temperatures of 200°C and above are values obtained from time-dependent properties. Notes - Welding Requirements
W1	No welding or brazing permitted.
W2	For Section VIII applications, UNF-56(d) shall apply for welded constructions.
W3	For welded and brazed constructions, stress values for O (annealed) temper material are used.
W4	The stress values given for this material are not applicable when either welding or thermal cutting is employed.
W5	These S values do not include a longitudinal weld efficiency factor. For Section III applications, for materials welded without filler metal, ultrasonic examination, radiographic examination, or eddy current examination, in accordance with NC-2550, shall provide a longitudinal weld efficiency factor of 1.0. Materials welded with filler metal meeting the requirements of NC-2560 shall receive a longitudinal weld efficiency factor of 1.00. Other long. weld efficiency factors shall be in accordance with the following: ▪ For single butt weld, with filler metal, 0.80. ▪ For single or double butt weld, without filler metal, 0.85. ▪ For double butt weld, with filler metal, 0.90. ▪ For single or double butt weld, with radiography or ultrasonic, 1.00.
W6	Filler metal is not used in the manufacture of welded pipe or tubing.
W7	Strength of reduced-section tensile specimen required to qualify welding procedures. See QW-150, Section IX.
W8	After welding, heat treat at 625ºC to 650ºC, hold 1½ hr at temperature for the first 25 mm of cross-section thickness and ½ hr for each additional 25 mm, and air cool. For castings used in pumps, valves, and fittings DN 50 and less, PWHT is not required for socket welds and attachment welds when the castings have been temper annealed at 625ºC to 650ºC prior to welding.
W9	Deleted.
W10	Deleted.

W11	These maximum allowable stress values are to be used in welded or brazed constructions.
W12	These S values do not include a weld factor. For Section VIII, Division 1and and Section XII applications using welds made without filler metal, the tabulated tensile stress values are multiplied by 0.85. For welds made with filler metal, consult UW-12 of Section VIII, Division 1, or TW-130.4 of Section XII, as applicable.
W13	For service at 650ºC or higher, the deposited weld metal is of the same nominal chemistry as the base metal.
W14	No welding permitted.
W15	For Section VIII and XII applications, no welding is permitted.
W16	Use NFA-12 when welded with 5356 or 5556 filler metal, all thicknesses, or 4043 or 5554 filler material, thickness <= 10 mm. Use NFA-13 when welded with 4043 or 5554 filler material, thickness > 10 mm.
W17	For welded and brazed constructions, stress values for the welded and annealed (WO61) temper material shall be used.

Division 1 Superseded Material Notes

Notes for the year 1943

(a)	Allowable working stresses in single shear = 0.8 times the given values.
(b)	Allowable working stresses in double shear = 1.6 times the given values.
(c)	Allowable working stresses in bearing = 1.8 times the given values.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	Values of stresses above 700 F are based upon steel in annealed condition.
1	Limited to plates not over 3/4 in. in thickness and to temperatures not above 750 F.

2	Maximum value for tensile strength permitted in design, 55,000 psi.
3	For present, limited to temperatures not above 750 F.
4	Only seamless steel pipe or tubing, or electric-fusion-welded pipe may be used for temperatures above 750 F.
5	Limited to temperatures not above 450 F.
6	Limited to temperatures not above 750 F.
7	Limited to temperatures not above 850 F.

Notes for the year 1952

(a)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
1	See Par. UG-6
2	Flange quality in this specification not permitted over 850 F.
3	These stress values are one-fourth the specified minimum tensile strength multiplied by a quality factor of 0.92, except for SA-283, Grade D. and SA-7.
4	For service temperatures above 850 F it is recommended that killed steels containing not less than 0.19% residual silicon be used. Killed steels which have been deoxidized with large amounts of aluminum and rimmed steels may have creep and stress-rupture properties in the temperature range above 850 F, which are somewhat less than those on which the values in the above table are based.
5	Between temperatures of 650 F and 1000 F, inclusive, the stress values for Specification SA-201, Grade B, may be used until high temperature test data become available.
6	Only (silicon) killed steel shall be used above 900 F.
7	To these stress values a quality factor as specified in Par. UG-24 shall be applied.
8	These stress values apply to normalized and drawn material only.

9	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints, where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the relative flexibility of the flange and bolts, and corresponding relaxation properties.
10	Between temperatures of —20 to 400 F, stress values equal to the lower of the following will be permitted: 20% of the specified tensile strength, or 25% of the specified yield strength.
11	Not permitted above 450 F; allowable stress value 7000 psi.
12	Between temperatures 750 F to 1000 F, inclusive, the stress values for Specification SA-212, Grade B, may be used until high temperature test data become available.
13	The stress values to be used for temperatures below —20 F when steels are made to conform with Specification SA-300 shall be those that are given in the column for —20 to 650 F.

Notes for the year 1965: (TABLE UCS-23)

(a)	Stress values in restricted shear such as dowel bolts, rivets, or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the given values.
(b)	Stress values in bearing shall be 1.60 times the given values.
(c)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
1	See Par. UCS-6(b).
2	Flange quality in this specification not permitted over 850 F.
3	These stress values are one-fourth the specified minimum tensile strength multiplied by a quality factor of 0.92, except for SA-283, Grade D. SA-7 and SA-36.

4	For service temperatures above 850 F it is recommended that killed steels containing not less than 0.10% residual silicon be used. Killed steels which have been deoxidized with large amounts of aluminum and rimmed steels may have creep and stress-rupture properties in the temperature range above 850 F:, which are somewhat less than those on which the values in the above table are based.
5	Between temperatures of 650 F and 1000 F, inclusive, the stress values for Specification SA-201, Grade B, may be used until high temperature test data become available.
6	Only (silicon) killed steel shall be used above 900 F.
7	To these stress values a quality factor as specified in Par. UG-24 shall be applied.
8	These stress values apply to normalized and drawn material only.
9	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints, where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary JS determined from the relative flexibility of the flange and bolts, and corresponding relaxation properties.
10	Between temperatures of —20 to 400 F, Stress values equal to the lower of the following will be permitted: 20% of the specified tensile strength, or 25% of the specified yield strength.
11	Not permitted above 450 F; allowable stress value 7000 psi.
12	Between temperatures of 750 F to 1000 F. inclusive, the stress values for Specification SA-212, Grade B, may be used until high temperature test data become available.
13	The stress values to be used for temperatures below —20 F when steels are made to conform with Specification SA-300 shall be those that are given in the column for —20 to 650 F.
15	For temperatures below 400 F, stress values equal to 20 per cent of the specified minimum tensile strength will be permitted.
19	These allowable stress values apply also to structural shapes and bars.
20	Stress values apply to normalized, or normalized and tempered or oil quenched and tempered material only, as per applicable specification.
21	Stress values apply to quenched and tempered material only, as per applicable specification.

<table>
<tr><td>22</td><td>Welding not permitted when carbon content exceeds 0.35 per cent by ladle analysis except for repairs or non-pressure attachments as outlined in Part UF.</td></tr>
<tr><td>23</td><td>Welding or brazing not permitted on liquid quenched and tempered material.</td></tr>
<tr><td>24</td><td>Maximum allowable stress values shall be as follows:
<table>
<tr><td>Grade</td><td>Liquid Quenched and Tempered (-20 to 200F)</td><td>Other Than Liquid Quenched and Tempered (-20 to 200F)</td></tr>
<tr><td>I</td><td>15,000</td><td>15,000</td></tr>
<tr><td>II</td><td>18,750</td><td>18,750</td></tr>
<tr><td>III</td><td>22,500</td><td>22,500</td></tr>
<tr><td>IV</td><td>26,250</td><td>26,250</td></tr>
<tr><td>V(A,B&E)</td><td>30,000</td><td></td></tr>
<tr><td>V(C&D)</td><td>30,000</td><td></td></tr>
</table></td></tr>
<tr><td>25</td><td>See Par. UCS-6 (c).</td></tr>
<tr><td>26</td><td>This material shall not be used in thicknesses above 0.58 in.</td></tr>
</table>

Notes for the year 1965:(TABLE UHA-23)

1	Due to the relatively low yield strength of this material, the higher stress values at temperatures from 200 through 1050F were established to permit the use of this material where slip-hay greater deformation is acceptable. The stress values within the above range exceed 621/2 per cent, but do not exceed 90 percent of the yield strength at temperature. These stress values are not recommended for the design of flanges or piping.
2	These stress values at temperatures of 1050F and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6.
3	These stress values shall be considered basic values to be used when no effort is made to control or check the grain size of the steel.
4	These stress values are the basic values multiplied by a joint efficiency factor of 0.85.

5	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints where freedom from leakage over a long period of time without retightening is required, lower values may be necessary as determined from the flexibility of the flange and bolts and corresponding relaxation.
6	These stress values a quality factor as specified in Par. UG-24 shall be applied.
7	These stress values permitted for material that has been carbide-solution treated.
8	For temperatures below 100F, stress values equal to 20 percent of the specified minimum tensile strength will be permitted.
9	This steel may be expected to develop embrittlement at room temperature after service at temperatures above 800F: consequently, its use at higher temperatures is not recommended unless due caution is observed.
10	At temperatures over 1000F, these stress values apply only when the carbon is 0.04 percent or higher.
11	For temperatures above 800F, the stress values apply only when the carbon content is 0.04 percent and above.
12	These stress values shall be applicable to forgings over 5 inches in thickness.

Notes for the year 1974

(a)	Stress values in restricted shear such as dowel bolts, rivets, or similar construction in which the shearing member is so restricted that the section under consideration would fail without reduction of area shall be 0.80 times the given values.
(b)	Stress values in bearing shall be 1.60 times the given values.
(c)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
1	See UCS-6(b).
3	These stress values are one fourth the specified minimum tensile strength multiplied by a qualify factor of 0.92, except for SA-283, Grade D, and SA-36.

4	For service temperatures above 850 F it is recommended that killed steels containing not less than 0.10 percent residual silicon be used. Killed steels which have been deoxidized with large amounts of aluminum and rimmed steels may have creep and stress rupture properties in the temperature range above 850 F. which are somewhat less than those on which the values in the above Table are based.
5	Between temperatures of 650 and 1000 F, inclusive, the stress values for Specification SA-201. Grade B. may be used until high temperature test data become available.
6	Only killed steel shall be used above 850 F.
7	To these stress values a quality factor as specified in UG-24 shall be applied for castings.
8	These stress values apply to normalized and drawn material only.
9	These stress values are established from a consideration of strength only and will be satisfactory for average service. For bolted joints, where freedom from leakage over a long period of time without retightening is required, lower stress values may be necessary as determined from the relative flexibility of the flange and bolts, and corresponding relaxation properties.
11	Not permitted above 450F; allowable stress value 7000 psi.
12	Between temperatures of 750 and 1000 F, inclusive, the stress values for Specification SA-515, Grade 70. May be used until high temperature test data become available.
13	The stress values to be used for temperatures below —20F when steels are made to conform with supplement (5)SA-20 shall be those that are given in the column for —20 to 650 F.
15	For temperatures below 400 F, stress values equal to 20 percent of the specified minimum tensile strength will be permitted.
19	These allowable stress values apply also to structural shapes and bars.
20	Stress values apply to normalized, or normalized and tempered or oil quenched and tempered material only, as per applicable specification.
21	Stress values apply to quenched and tempered material only, as per applicable specification.
22	Welding or brazing is not permitted when carbon content exceeds 0.35 percent by ladle analysis except for limited types of welding as allowed in Part UF.
23	Welding or brazing not permitted on liquid quenched and tempered material.

24	Maximum allowable stress values shall be as follows:

Grade	Normalized or Normalized and Tempered			Liquid Quenched and Tempered				
	-20 to 650	-20 to 100	200	300	400	500	600	650
I	15,000	15,000	15,000					
II	18,750	18,750	18,750					
III	22,500	22,500	22,500					
IV	26,250	26,250	25,050	24,600	24,600	24,600	24,600	24,600
VA	30,000	28,850	28,850	28,850	28,850	28,850	28,850	
VB	30,000	29,050	28,500	28,500	28,200	27,800	26,750	
VE	30,000	29,800	28,700	28,700	28,700	28,700	27,500	
VC&D		30,000	30,000					
VIII		33,700	32,300	32,100	31,900	31,600	31,400	30,000

26	This material shall not be used in thicknesses above 0.58 in.
27	Upon prolonged exposure to temperatures above 800 F. the carbide phase of carbon steel may be converted to graphite.
28	Upon prolonged exposure to temperatures above 875 F, the carbine phase of carbon-molybdenum steel may be converted to graphite.
29	The material shall not be used in thickness above 0.375 in.
30	For temperatures above which stresses are given, the allowable stresses for the annealed plate shall be used.
31	Where the fabricator performs the heat treatment the requirements of UHT-81 shall be met.
32	Section IX, QW-250 Variables QW404.12, QW406.3, QW407.2, and QW-409.1 of QW-422 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Part UF of Division I.

ASME Section VIII, Division 2, Class 1 Material Notes for Table 2A (Ferrous Materials) - Customary

(a)	The following abbreviations are used: Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E2).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 are used when SA-516M Grade 485 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross–section of other pressure parts, e.g., castings and forgings. Notes - Experience Criterion
E1	For values at 650ºF and above, the design stress intensity values are based on successful experience in service.
E2	For values at 700ºF and above, the design stress intensity values are based on successful experience in service.
E3	For values at 850°F and above, the design stress intensity values are based on successful experience in service.
E4	For values at 900°F, the design stress intensity values are based on successful experience in service. NOTES – GENERAL REQUIREMENTS
G1	Material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted.
G2	Material that conforms to Class 11 or 12 is not permitted.

G3	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds ¾ in.
G4	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds 1¼ in.
G5	For Section III applications, a product analysis is required on this material.
G6	SA-723 is not used for minimum permissible temperature below +40°F.
G7	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66 2/3 % but do not exceed 90% of the yield strength at temperature. Use of these stresses may results in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. **Table Y-2** lists multiplying factors that, when applied to the yield strength values shown in **Table Y-1**, give allowable stress values that will result in lower values of permanent strain.
G8	This material has reduced toughness at room temperature after exposure at high temperature. The degree of embrittlement depends on composition, heat treatment, time and temperature. The lowest temperature of concern is about 500ºF. See **Appendix A, A-360**.
G9	At temperatures over 1000ºF, these stress intensity values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 1000ºF are published.
G10	For temperatures above 1000°F, these stress intensity values may be used only if the material has been heat treated by heating to a minimum temperature of 1900°F and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 1000ºF are published.
G11	These stress intensity values at temperatures of 1050°F and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 1000°F are published.
G12	These stress intensity values are considered basic values to be used when no effort is made to control or check the grain size of the steel.
G13	This steel may be expected to develop embrittlement after service at moderately elevated temperature. See **Appendix A, A-340** and **A-360**.

G14	All forgings have a maximum tensile strength not in excess of 25 ksi above the specified minimum.
G15	Fabricated from SA-387 Grade 12 Class 1 plate.
G16	Fabricated from SA-387 Grade 12 Class 2 plate.
G17	A factor of 4 was used for tensile strength to obtain the stress intensity value.
G18	A quality factor of 0.85 has been applied in arriving at the design stress intensity values for this material.
G19	The tensile strength shall not be in excess of 20,000 psi above the specified minimum.
G20	For Section VIII applications, SA–723 is exempt from the requirement in Section VIII, Division 2, 6.7.6.3(b) that the average of the individual Brinell hardness numbers shall not be more than 10% below or 25% above the number corresponding to the tensile strength
G21	See Section VIII, Division 2, 3.4. Notes – Heat Treatment Requirements
H1	Annealed.
H2	Deleted
H3	For Section III applications, pieces that are formed (after quenching and tempering) at a temperature lower than 25ºF below the final tempering temperature are heat-treated after forming when the extreme fiber strain from forming exceeds 3%. Heat treatment shall be 1075ºF minimum, but not higher than 25ºF below the final tempering temperature for a minimum time of one hour per inch of thickness. Pieces formed at temperatures within 25ºF higher than the original tempering temperature are requenched and tempered, either before or after welding into the vessel.
H4	Liquid quenched and tempered.
H5	Normalized, normalized and tempered, or quenched and tempered.
H6	For Section VIII applications involving consideration of heat treatment after forming or welding, see Section VIII, Division 2, Table 6.15
H7	Quenched and tempered at 1200°F. Notes – Size Requirements
S1	The maximum thickness of forgings does not exceed 3¾ in. (4 in. as heat treated).

S2	Both NPS 8 and larger, and schedule 140 and heavier.
S3	The minimum thickness of pressure-retaining parts is ¼ in..
S4	For Section III applications, the minimum thickness of shells, heads, and other pressure-retaining parts is ¼ in.. The maximum thickness is limited only by the ability to develop the specified mechanical properties.
S5	Either NPS 8 and larger and less than schedule 140 wall, or less than NPS 8 and all wall thicknesses.
S6	The maximum section thickness shall not exceed 3 in. for double–normalized–and–tempered forgings, or 5 in. for quenched–and–tempered forgings. Notes – Welding Requirements
W1	Not for welded construction.
W2	In welded construction, for temperatures above 850°F, the weld metal has a carbon content of greater than 0.05%.
W3	The following, in addition to the variables in Section IX, QW-250, are considered as essential variables requiring requalification of the welding procedure. ▪ An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures does not exceed 150ºF. ▪ A change in the thickness T of the welding procedure qualification test plate as follows: a. For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness qualified in all cases is ¼ in.). b. For welded joints that are not quenced and tempered after welding, any changes as follows: (a) For T less than 5/8 in., any decrease in thickness (the maximum thickness qualified is 2T); (b) for T equal to 5/8 in. and over, any departure from the range of 5/8 in. to 2T.
W4	Welded, with the tensile strength of the Section IX reduced section tension test less than 100 ksi but not less than 95 ksi.
W5	For Section VIII applications, welding not permitted when carbon content exceeds 0.35% by ladle analysis except for limited types of welding, as allowed in Section VIII, Division 2, Part 6.
W6	For Section VIII applications, Section IX, QW–250 Variables QW–404.12, QW–406.3, QW–407.2, and QW–409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Section VIII, Division 2, Part 6.

W7	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 100 ksi.

ASME Section VIII, Division 2, Class 1 Material Notes for Table 2A (Ferrous Materials) - Metric

(a)	The following abbreviations are used: Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E4).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross–section of other pressure parts, e.g., castings and forgings. Notes – Experience Criterion
E1	For values at 350°C and above, the design stress intensity values are based on successful experience in service.
E2	For values at 375°C and above, the design stress intensity values are based on successful experience in service.
E3	For values at 450°C and above, the design stress intensity values are based on successful experience in service.

E4	For values at 475°C, the design stress intensity values are based on successful experience in service. Notes – General Requirements
G1	Material that conforms to Class 10, 13, 20, 23, 30, 33, 40, 43, 50, or 53 is not permitted.
G2	Material that conforms to Class 11 or 12 is not permitted.
G3	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds 19 mm.
G4	Material that conforms to Class 11 or 12 is not permitted when the nominal thickness of the material exceeds 32 mm.
G5	For Section III applications, a product analysis is required on this material.
G6	SA-723 is not used for minimum permissible temperature below +5°C.
G7	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. **Table Y-2** lists multiplying factors that, when applied to the yield strength values shown in **Table Y-1**, will give allowable stress values that will result in lower levels of permanent strain.
G8	This material has reduced toughness at room temperature after exposure at high temperature. The degree of embrittlement depends on composition, heat treatment, time, and temperature. The lowest temperature of concern is about 250°C. See Appendix A, A-207.
G9	At temperatures over 550°C, these stress intensity values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 550°C are published.
G10	For temperatures above 550°C, these stress intensity values may be used only if the material has been heat treated by heating to a minimum temperature of 1040°C and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 550°C are published.
G11	These stress intensity values at temperatures of 575°C and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 550°C are published.

G12	These stress intensity values are considered basic values to be used when no effort is made to control or check the grain size of the steel.
G13	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Appendix A, A-207 and A-208.
G14	All forgings have a maximum tensile strength not in excess of 175 MPa above the specified minimum.
G15	Fabricated from SA-387 Grade 12 Class 1 plate.
G16	Fabricated from SA-387 Grade 12 Class 2 plate.
G17	A factor of 4 was used for tensile strength to obtain the stress intensity value.
G18	A quality factor of 0.85 has been applied in arriving at the design stress intensity values for this material.
G19	The tensile strength shall not be in excess of 140 MPa above the specified minimum.
G20	For Section VIII applications, SA–723 is exempt from the requirement in Section VIII, Division 2, 6.7.6.3(b) that the average of the individual Brinell hardness numbers shall not be more than 10% below or 25% above the number corresponding to the tensile strength.
G21	See Section VIII, Division 2, 3.4. Notes - Heat Treatment Requirements
H1	Annealed.
H2	Deleted.
H3	For Section III applications, pieces that are formed (after quenching and tempering) at a temperature lower than 15°C below the final tempering temperature shall be heat treated after forming when the extreme fiber strain from forming exceeds 3%. Heat treatment shall be 580°C minimum, but not higher than 15°C below the final tempering temperature for a minimum time of 1 h per 25 mm of thickness. Pieces formed at temperatures within 15°C higher than the original tempering temperature shall be requenched and tempered, either before or after welding into the vessel.
H4	Liquid quenched and tempered.
H5	Normalized, normalized and tempered, or quenched and tempered.
H6	For Section VIII applications involving consideration of heat treatment after forming or welding, see Section VIII, Division 2, Table 6.15 for P–No. 10K, Group No. 1 materials.

H7	Quenched and tempered at 650°C. Notes - Size Requirements
S1	The maximum thickness of forgings does not exceed 95 mm (100 mm as heat treated).
S2	Both DN 200 and larger, and schedule 140 and heavier.
S3	The minimum thickness of pressure-retaining parts is 6 mm.
S4	For Section III applications, the minimum thickness of shells, heads, and other pressure–retaining parts shall be 6 mm. The maximum thickness shall be limited only by the ability to develop the specified mechanical properties.
S5	Either DN 200 and larger and less than schedule 140 wall, or less than DN 200 and all wall thicknesses.
S6	The maximum section thickness shall not exceed 75 mm for double–normalized–and–tempered forgings, or 125 mm for quenched–and– tempered forgings. Notes - Welding Requirements
W1	Not for welded construction.
W2	In welded construction, for temperatures above 450°C, the weld metal has a carbon content of greater than 0.05%.
W3	The following, in addition to the variables in Section IX, QW-250, is considered as essential variables requiring requalification of the welding procedure: ▪ An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures shall not exceed 85°C. ▪ A change in the thickness T of the welding procedure qualification test plate as follows: a. For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness qualified in all cases is 6 mm). b. For welded joints that are not quenched and tempered after welding, any change as follows: (a) for T less than 16 mm, any decrease in thickness (the maximum thickness qualified is 2T) (b) for T equal to 16 mm and over, any departure from the range of 16 mm to 2T.
W4	Welded, with the tensile strength of the Section IX reduced section tension test less than 690 MPa but not less than 655 MPa.
W5	For Section VIII applications, welding not permitted when carbon content exceeds 0.35% by ladle analysis except for limited types of welding, as allowed in Section VIII, Division 2, Part 6.

W6	For Section VIII applications, Section IX, QW–250 Variables QW–404.12, QW–406.3, QW–407.2, and QW–409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Section VIII, Division 2, Part 6.
W7	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 690 MPa.

ASME Section VIII, Division 2, Class 1 Material Notes for Table 2B (Non-Ferrous Materials) - Customary

(a)	The following abbreviations are used: ann., annealed; fin., finished; rel., relieved; Smls., Seamless; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E4).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 are used when SB-407M Grade N08800 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross–section of other pressure parts, e.g., castings and forgings. Notes - Experience Criterion
E1	For values at 800°F, the design stress intensity values are based on successful experience in service.

E2	For values at 250°F and above, the design stress intensity values are based on successful experience in service.
E3	For values at 400°F and above, the design stress intensity values are based on successful experience in service.
E4	For values at 750°F and above, the design stress intensity values are based on successful experience in service. Notes – General Requirements
G1	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short– time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 662/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y–2 lists multiplying factors that, when applied to the yield strength values shown in Table Y–1, will give allowable stress values that will result in lower levels of permanent strain.
G2	Use of external pressure charts for material in the form of barstock is permitted for stiffening rings only.
G3	SB-163 Supplementary Requirement S2 is met.
G4	Design stress intensity values for 100°F may be used at temperatures down to –325°F without additional specification requirements.
G5	A joint efficiency factor of 0.85 has been applied in arriving at the maximum allowable design stress intensity values for this material.
G6	Maximum temperature for external pressure design not to exceed 350°F.
G7	These alloys are occasionally subject to the hazard of stress corrosion cracking. Even though they are suitable for engineering use under a wide variety of corrosive conditions, with no particular hazard with respect to stress corrosion, the supplier of the material should be consulted before applying them.
G8	Design stress intensity values for 100°F may be used at temperatures down to –452°F without additional specification requirements.
G9	For stress relieved tempers (T451, T4510, T4511, T651, T6510, and T6511), stress values for materials in the basic temper shall be used.

G10	Copper–silicon alloys are not always suitable when exposed to certain media and high temperature, particularly steam above 212°F. The user should satisfy him/herself that the alloy selected is satisfactory for the service for which it is to be used. Notes – Size Requirements
S1	Thickness ≤ 0.100 mm. Notes – Welding Requirements
W1	Welding except for seal welds is not permitted.
W2	For welded construction, stress intensity values for material of O temper shall be used.
W3	The stress intensity values given for this material are not applicable when either welding or thermal cutting is employed.

ASME Section VIII, Division 2, Class 1 Material Notes for Table 2B (Non-Ferrous Materials) - Metric

(a)	The following abbreviations are used: ann., annealed; fin., finished; rel., relieved; Smls., Seamless; and Wld., Welded.
(b)	An alternative typeface is used for stress values based on successful experience in service (see Notes E1 and E2).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table are applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 are used when SB-407M Grade N08800 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See **Nonmandatory Appendix A** for more information.

(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross–section of other pressure parts, e.g., castings and forgings. Notes - Experience Criterion
E1	For values at 425°C, the design stress intensity values are based on successful experience in service.
E2	For values at 125°C and above, the design stress intensity values are based on successful experience in service.
E3	For values at 200°C and above, the design stress intensity values are based on successful experience in service.
E4	For values at 400°C and above, the design stress intensity values are based on successful experience in service.
E5	For values at 150°C and above, the design stress intensity values are based on successful experience in service. NOTES – GENERAL REQUIREMENTS
G1	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short– time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 662/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y–2 lists multiplying factors that, when applied to the yield strength values shown in Table Y–1, will give allowable stress values that will result in lower levels of permanent strain.
G2	Use of external pressure charts for material in the form of barstock is permitted for stiffening rings only.
G3	SB-163 Supplementary Requirement S2 is met.
G4	Design stress intensity values for 40°C may be used at temperatures down to –200°C without additional specification requirements.
G5	A joint efficiency factor of 0.85 has been applied in arriving at the maximum allowable design stress intensity values for this material.
G6	Maximum temperature for external pressure design not to exceed 175°C.

G7	These alloys are occasionally subject to the hazard of stress corrosion cracking. Even though they are suitable for engineering use under a wide variety of corrosive conditions, with no particular hazard with respect to stress corrosion, the supplier of the material should be consulted before applying them.
G8	Design stress intensity values for 40°C may be used at temperatures down to –270°C without additional specification requirements.
G9	For stress relieved tempers (T451, T4510, T4511, T651, T6510, and T6511), stress values for materials in the basic temper shall be used.
G10	Copper–silicon alloys are not always suitable when exposed to certain media and high temperature, particularly steam above 100°C. The user should satisfy him/herself that the alloy selected is satisfactory for the service for which it is to be used. NOTES – SIZE REQUIREMENTS
S1	Thickness ≤ 2.5 mm. NOTES – WELDING REQUIREMENTS
W1	Welding except for seal welds is not permitted.
W2	For welded construction, stress intensity values for material of O temper shall be used.
W3	The stress intensity values given for this material are not applicable when either welding or thermal cutting is employed.

ASME Section VIII, Division 2, Class 2 Table 5A Notes for Ferrous Materials

(a)	The following abbreviations are used: NT, Normalized and tempered; QT, Quenched and tempered; Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T10).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.

(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form; wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings. Notes - General Requirements
G1	For temperatures above 1000°F, these stress values may be used only if the material is heat treated by heating to a minimun temperature of 2000°F, and quenching in water or rapidly cooling by other means.
G2	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short– time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 662/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y–2 lists multiplying factors that, when applied to the yield strength values shown in Table Y–1, will give allowable stress values that will result in lower levels of permanent strain.
G3	At temperatures over 1000°F, these stress values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 1000°F are published.
G4	For temperatures above 1000°F, these stress values may be used only if the material has been heat treated by heating to a minimum temperature of 1900°F and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 1000°F are published.
G5	These stress values at temperatures of 1050°F and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 1000°F are published.

G6	A quality factor of 0.85 has been applied in arriving at the maximum allowable stress values for this material.
G7	These stress values shall be considered basic values to be used when no effort is made to control or check the grain size of the steel.
G8	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Nonmandatory Appendix Appendix A, A-207 and A-208.
G9	The tensile strength shall not be in excess of 20,000 psi above the specified minimum.
G10	All forgings shall have a maximum tensile strength not in excess of 25 ksi above the specified minimum.
G11	SA-723 is exempt from the requirement in Section VIII, Division 2, 6.7.6.3(b) that the average of the individual Brinell hardness numbers shall not be more than 10% below or 25% above the number corresponding to the tensile strength.
G12	See Section VIII, Division 2, 3.4.
G13	Upon prolonged exposure to temperatures above 800°F, the carbide phase of carbon steel may be converted to graphite. See Nonmandatory Appendix A, A–201 and A–202.
G14	Upon prolonged exposure to temperatures above 875°F, the carbide phase of carbon–molybdenum steel may be converted to graphite. See Nonmandatory Appendix A, A–201 and A–202.
G15	This material may be susceptible to temper embrittlement. See Nonmandatory Appendix A, A-203.
G16	These stresses apply to all product forms (C, H, and P) as defined in SA/EN 10028–7. NOTES – HEAT TREATMENT REQUIREMENTS
H1	Annealed.
H2	Deleted.
H3	Deleted.
H4	Liquid quenched and tempered.
H5	Normalized, normalized and tempered, or quenched and tempered.
H6	For applications involving consideration of heat treatment after forming or welding, see Section VIII, Division 2, Table 6.15 for P-No. 10K, Group No. 1 materials.

H7	Deleted. Notes - Size Requirements
S1	The maximum thickness of forgings shall not exceed 33/4 in. (4 in. as heat treated).
S2	The maximum section thickness shall not exceed 3 in. for double–normalized–and–tempered forgings, or 5 in. for quenched–and–tempered forgings.
S3	Both NPS 8 and larger, and schedule 140 and heavier.
S4	Either NPS 8 and larger and less than schedule 140 wall, or less than NPS 8 and all wall thicknesses. NOTES – TIME–DEPENDENT PROPERTIES
T1	Allowable stresses for temperatures of 650°F and above are values obtained from time–dependent properties.
T2	Allowable stresses for temperatures of 700°F and above are values obtained from time–dependent properties.
T3	Allowable stresses for temperatures of 750°F and above are values obtained from time–dependent properties.
T4	Allowable stresses for temperatures of 800°F and above are values obtained from time–dependent properties.
T5	Allowable stresses for temperatures of 850°F and above are values obtained from time–dependent properties.
T6	Allowable stresses for temperatures of 900°F and above are values obtained from time–dependent properties.
T7	Allowable stresses for temperatures of 950°F and above are values obtained from time–dependent properties.
T8	Allowable stresses for temperatures of 1000°F and above are values obtained from time–dependent properties.
T9	Allowable stresses for temperatures of 1050°F and above are values obtained from time–dependent properties.
T10	Allowable stresses for temperatures of 1100°F and above are values obtained from time–dependent properties. NOTES – WELDING REQUIREMENTS
W1	Not for welded construction.

W2	Welding is not permitted when carbon content exceeds 0.35% by ladle analysis except for limited types of welding, as allowed in Section VIII, Division 2, Part 6.
W3	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 100 ksi.
W4	Welded, with the tensile strength of the Section IX reduced section tension test less than 100 ksi but not less than 95 ksi.
W5	In welded construction, for temperatures above 850°F, the weld metal shall have a carbon content of greater than 0.05%.
W6	Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Section VIII, Division 2, Part 6.
W7	The following, in addition to the variables in Section IX, QW–250, shall be considered as essential variables requiring requalification of the welding procedure: (a) An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures shall not exceed 150°F. (b) A change in the thickness T of the welding procedure qualification test plate as follows: (1) For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness qualified in all cases is 1/4 in.). (2) For welded joints that are not quenched and tempered after welding, any change as follows: (–a) for T less than 5/8 in., any decrease in thickness (the maximum thickness qualified is 2T); (–b) for T equal to 5/8 in. and over, any departure from the range of 5/8 in. to 2T.

ASME Section VIII, Division 2, Class 2 Table 5A Notes for Ferrous Materials - Metric

(a)	The following abbreviations are used: NT, Normalized and tempered; QT, Quenched and tempered; Smls., Seamless; Temp., Temperature; and Wld., Welded.
(b)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T11).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SA-516/SA-516M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SA-516 Grade 70 shall be used when SA-516M Grade 485 is used in construction.

(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form; wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings. Notes - General Requirements
G1	For temperatures above 550°C, these stress values may be used only if the material is heat treated by heating to a minimum temperature of 1095°C, and quenching in water or rapidly cooling by other means.
G2	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower levels of permanent strain.
G3	At temperatures over 550°C, these stress values apply only when the carbon is 0.04% or higher. This note is applicable only when stresses above 550°C are published.
G4	For temperatures above 550°C, these stress values may be used only if the material has been heat treated by heating to a minimum temperature of 1040°C and quenching in water or rapidly cooling by other means. This note is applicable only when stresses above 550°C are published.
G5	These stress values at temperatures of 575°C and above should be used only when assurance is provided that the steel has a predominant grain size not finer than ASTM No. 6. This note is applicable only when stresses above 550°C are published.

G6	A quality factor of 0.85 has been applied in arriving at the maximum allowable stress values for this material.
G7	These stress values shall be considered basic values to be used when no effort is made to control or check the grain size of the steel.
G8	This steel may be expected to develop embrittlement after service at moderately elevated temperature; see Nonmandatory Appendix Appendix A, A-207 and A-208.
G9	The tensile strength shall not be in excess of 140 MPa above the specified minimum.
G10	All forgings shall have a maximum tensile strength not in excess of 175 MPa above the specified minimum.
G11	SA-723 is exempt from the requirement in Section VIII, Division 2, 6.7.6.3(b) that the average of the individual Brinell hardness numbers shall not be more than 10% below or 25% above the number corresponding to the tensile strength.
G12	See Section VIII, Division 2, 3.4.
G13	Upon prolonged exposure to temperatures above 425°C, the carbide phase of carbon steel may be converted to graphite. See Nonmandatory Appendix A, A–201 and A–202.
G14	Upon prolonged exposure to temperatures above 475°C, the carbide phase of carbon–molybdenum steel may be converted to graphite.
G15	This material may be susceptible to temper embrittlement. See Nonmandatory Appendix A, A-203.
G16	These stresses apply to all product forms (C, H, and P) as defined in SA/EN 10028–7. NOTES – HEAT TREATMENT REQUIREMENTS
H1	Annealed.
H2	Deleted.
H3	Deleted.
H4	Liquid quenched and tempered.
H5	Normalized, normalized and tempered, or quenched and tempered.

H6	For applications involving consideration of heat treatment after forming or welding, see Section VIII, Division 2, Table 6.15 for P-No. 10K, Group No. 1 materials.
H7	Deleted. Notes - Size Requirements
S1	The maximum thickness of forgings shall not exceed 95 mm (100 mm as heat treated).
S2	The maximum section thickness shall not exceed 75 mm for double–normalized–and–tempered forgings, or 125 mm for quenched–and– tempered forgings.
S3	Both DN 200 and larger, and schedule 140 and heavier.
S4	Either DN 200 and larger and less than schedule 140 wall, or less than DN 200 and all wall thicknesses. NOTES – TIME–DEPENDENT PROPERTIES
T1	Allowable stresses for temperatures of 350°C and above are values obtained from time–dependent properties.
T2	Allowable stresses for temperatures of 375°C and above are values obtained from time–dependent properties.
T3	Allowable stresses for temperatures of 400°C and above are values obtained from time–dependent properties.
T4	Allowable stresses for temperatures of 425°C and above are values obtained from time–dependent properties.
T5	Allowable stresses for temperatures of 450°C and above are values obtained from time–dependent properties.
T6	Allowable stresses for temperatures of 475°C and above are values obtained from time–dependent properties.
T7	Allowable stresses for temperatures of 500°C and above are values obtained from time–dependent properties.
T8	Allowable stresses for temperatures of 525°C and above are values obtained from time–dependent properties.
T9	Allowable stresses for temperatures of 550°C and above are values obtained from time–dependent properties.
T10	Allowable stresses for temperatures of 575°C and above are values obtained from time–dependent properties.

T11	Allowable stresses for temperatures of 600°C and above are values obtained from time–dependent properties. NOTES – WELDING REQUIREMENTS
W1	Not for welded construction.
W2	Welding is not permitted when carbon content exceeds 0.35% by ladle analysis except for limited types of welding, as allowed in Section VIII, Division 2, Part 6.
W3	Nonwelded, or welded if the tensile strength of the Section IX reduced section tension test is not less than 690 MPa.
W4	Welded, with the tensile strength of the Section IX reduced section tension test less than 690 MPa but not less than 655 MPa.
W5	In welded construction, for temperatures above 450°C, the weld metal shall have a carbon content of greater than 0.05%.
W6	Section IX, QW-250 Variables QW-404.12, QW-406.3, QW-407.2, and QW-409.1 shall also apply to this material. These variables shall be applied in accordance with the rules for welding of Section VIII, Division 2, Part 6.
W7	The following, in addition to the variables in Section IX, QW–250, shall be considered as essential variables requiring requalification of the welding procedure: a. An increase in the maximum or a decrease in the minimum specified preheat or interpass temperatures. The specified range of preheat temperatures shall not exceed 85°C. b. A change in the thickness T of the welding procedure qualification test plate as follows: (1) For welded joints that are quenched and tempered after welding, any increase in thickness (the minimum thickness qualified in all cases is 6 mm). (2) For welded joints that are not quenched and tempered after welding, any change as follows: (–a) for T less than 16 mm, any decrease in thickness (the maximum thickness qualified is 2T); (–b) for T equal to 16 mm and over, any departure from the range of 16 mm to 2T.

ASME Section VIII, Division 2, Class 2 Table 5B Notes for Non-Ferrous Materials

(a)	The following abbreviations are used: ann., annealed; extr., extruded; fin., finished; rel., relieved; Smls., Seamless; and Wld., Welded.

(b)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T14).
(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 shall be used when SB-407M Grade N08800 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings. Notes - General Requirements
G1	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower levels of permanent strain.
G2	Use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G3	Maximum allowable stress values for 100ºF may be used at temperatures down to –325ºF without additional specification requirements.

G4	Maximum allowable stress values for 100°F may be used at temperatures down to –452ºF without additional specification requirements.
G5	Maximum temperature for external pressure design not to exceed 350°F.
G6	These alloys are occasionally subject to the hazard of stress corrosion cracking. Even though they are suitable for engineering use under a wide variety of corrosive conditions, with no particular hazard with respect to stress corrosion, the supplier of the material should be consulted before applying them.
G7	A joint efficiency factor of 0.85 has been applied in arriving at the maximum allowable stress values for this material.
G8	For stress relieved tempers (T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper shall be used.
G9	Copper-silicon alloys are not always suitable when exposed to certain media and high temperature, particularly steam above 212°F. The user should satisfy him/herself that the alloy selected is satisfactory for the service for which it is to be used.
G10	At temperatures over 1000ºF, these stress values apply only when the carbon is 0.04% or higher.
G11	This alloy is subject to severe loss of impact strength at room temperatures after exposure in the range of 1000ºF to 1400ºF.
G12	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 1000ºF to 1250ºF.
G13	Creep-fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 1500ºF and shall be considered in the design. Notes - Heat Treatment Requirements
H1	For temperatures above 1000ºF, these stress values may be used only if the material is annealed at a minimum temperature of 1900ºF and has a carbon content of 0.04% or higher.
H2	For temperatures above 1000ºF, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1900ºF and quenching in water or rapidly cooling by other means.
T1	Allowable stresses for temperatures of 250ºF and above are values obtained from time-dependent properties.

T2	Allowable stresses for temperatures of 300ºF and above are values obtained from time-dependent properties.
T3	Allowable stresses for temperatures of 350ºF and above are values obtained from time-dependent properties.
T4	Allowable stresses for temperatures of 400ºF and above are values obtained from time-dependent properties.
T5	Allowable stresses for temperatures of 500ºF and above are values obtained from time-dependent properties.
T6	Allowable stresses for temperatures of 700ºF and above are values obtained from time-dependent properties.
T7	Allowable stresses for temperatures of 750ºF and above are values obtained from time-dependent properties.
T8	Allowable stresses for temperatures of 800ºF and above are values obtained from time-dependent properties.
T9	Allowable stresses for temperatures of 850ºF and above are values obtained from time-dependent properties.
T10	Allowable stresses for temperatures of 900ºF and above are values obtained from time-dependent properties.
T11	Allowable stresses for temperatures of 950ºF and above are values obtained from time-dependent properties.
T12	Allowable stresses for temperatures of 1000ºF and above are values obtained from time-dependent properties.
T13	Allowable stresses for temperatures of 1050ºF and above are values obtained from time-dependent properties.

ASME Section VIII, Division 2, Class 2 Table 5B Notes for Non-Ferrous Materials - Metric

(a)	The following abbreviations are used: ann., annealed; extr., extruded; fin., finished; rel., relieved; Smls., Seamless; and Wld., Welded.
(b)	An alternative typeface is used for stress values obtained from time-dependent properties (see Notes T1 through T14).

(c)	Where specifications, grades, classes, and types are listed in this Table, and where the material specification in Section II, Part A or Part B is a dual-unit specification (e.g., SB-407/SB-407M), the values listed in this Table shall be applicable to either the customary U.S. version of the material specification or the SI units version of the material specification. For example, the values listed for SB-407 Grade N08800 shall be used when SB-407M Grade N08800 is used in construction.
(d)	The values in this Table may be interpolated to determine values for intermediate temperatures. The values at intermediate temperatures shall be rounded to the same number of decimal places as the value at the higher temperature between which values are being interpolated. The rounding rule is: when the next digit beyond the last place to be retained is less than 5, retain unchanged the digit in the last place retained; when the digit next beyond the last place to be retained is 5 or greater, increase by 1 the digit in the last place retained.
(e)	The properties of steels are influenced by the processing history, heat treatment, melting practice, and level of residual elements. See Nonmandatory Appendix A for more information.
(f)	Where a size limit appears in the Size/Thickness column, the limit applies to the dimension appropriate to the product form: wall thickness of tubing, pipe, pipe fittings, and hollow forgings; thickness of plate, flat bar and forgings, and polygonal bar; diameter of solid bar and bolting; and thickest cross-section of other pressure parts, e.g., castings and forgings. Notes - General Requirements
G1	Due to the relatively low yield strength of these materials, these higher stress values were established at temperatures where the short-time tensile properties govern to permit the use of these alloys where slightly greater deformation is acceptable. The stress values in this range exceed 66-2/3% but do not exceed 90% of the yield strength at temperature. Use of these stresses may result in dimensional changes due to permanent strain. These stress values are not recommended for the flanges of gasketed joints or other applications where slight amounts of distortion can cause leakage or malfunction. Table Y-2 lists multiplying factors that, when applied to the yield strength values shown in Table Y-1, will give allowable stress values that will result in lower levels of permanent strain.
G2	Use of external pressure charts for material in the form of bar stock is permitted for stiffening rings only.
G3	Maximum allowable stress values for 40°C may be used at temperatures down to –200°C without additional specification requirements.
G4	Maximum allowable stress values for 40°C may be used at temperatures down to –270°C without additional specification requirements.

G5	Maximum temperature for external pressure design not to exceed 175°C.
G6	These alloys are occasionally subject to the hazard of stress corrosion cracking. Even though they are suitable for engineering use under a wide variety of corrosive conditions, with no particular hazard with respect to stress corrosion, the supplier of the material should be consulted before applying them.
G7	A joint efficiency factor of 0.85 has been applied in arriving at the maximum allowable stress values for this material.
G8	For stress relieved tempers (T451, T4510, T4511, T651, T6510, T6511), stress values for materials in the basic temper shall be used.
G9	Copper–silicon alloys are not always suitable when exposed to certain media and high temperature, particularly steam above 100°C. The user should satisfy him/herself that the alloy selected is satisfactory for the service for which it is to be used.
G10	At temperatures over 550°C, these stress values apply only when the carbon is 0.04% or higher.
G11	This alloy is subject to severe loss of impact strength at room temperatures after exposure in the range of 550°C to 750°C.
G12	Alloy N06022 in the solution annealed condition is subject to severe loss of impact strength at room temperatures after exposure in the range of 550°C to 675°C.
G13	Creep–fatigue, thermal ratcheting, and environmental effects are increasingly significant failure modes at temperatures in excess of 825°C and shall be considered in the design. Notes – Heat Treatment Requirements
H1	For temperatures above 550°C, these stress values may be used only if the material is annealed at a minimum temperature of 1040°C and has a carbon content of 0.04% or higher.
H2	For temperatures above 550°C, these stress values may be used only if the material is heat treated by heating it to a minimum temperature of 1040°C and quenching in water or rapidly cooling by other means. Notes – Time–Dependent Properties
T1	Allowable stresses for temperatures of 125°C and above are values obtained from time–dependent properties.
T2	Allowable stresses for temperatures of 150°C and above are values obtained from time–dependent properties.

T3	Allowable stresses for temperatures of 175°C and above are values obtained from time–dependent properties.
T4	Allowable stresses for temperatures of 200°C and above are values obtained from time–dependent properties.
T5	Allowable stresses for temperatures of 275°C and above are values obtained from time–dependent properties.
T6	Allowable stresses for temperatures of 325°C and above are values obtained from time–dependent properties.
T7	Allowable stresses for temperatures of 375°C and above are values obtained from time–dependent properties.
T8	Allowable stresses for temperatures of 400°C and above are values obtained from time–dependent properties.
T9	Allowable stresses for temperatures of 425°C and above are values obtained from time–dependent properties.
T10	Allowable stresses for temperatures of 450°C and above are values obtained from time–dependent properties.
T11	Allowable stresses for temperatures of 500°C and above are values obtained from time–dependent properties.
T12	Allowable stresses for temperatures of 525°C and above are values obtained from time–dependent properties.
T13	Allowable stresses for temperatures of 550°C and above are values obtained from time–dependent properties.
T14	Allowable stresses for temperatures of 575°C and above are values obtained from time–dependent properties. NOTES – WELDING REQUIREMENTS
W1	Welding except for seal welds is not permitted.
W2	For welded construction, stress values for material at O temper shall be used.
W3	The stress values given for this material are not applicable when either welding or thermal cutting is employed.
W4	Use NFA–12 when welded with 5356 or 5556 filler metal, all thicknesses, or 4043 or 5554 filler metal, thickness ≤ 10 mm. Use NFA–13 when welded with 4043 or 5554 filler metal, thickness > 10 mm.

Material Properties Dialog Box

Displays properties for the selected material. You can modify some properties. Doing so only changes the properties locally. It does not modify the database.

NOTE Properties available in this dialog box vary depending on the command used.

Material Name

Displays the ASME code material specification for the selected item.

Listing #

Indicates a unique identification number for the material.

Yield Stress, Design

Enter the yield stress for the material at the operating temperature. You can find these values in the ASME Code, Section 2 Part D; they are not stored in the material database. If the yield stress at operating temperature is significantly different than the yield stress at ambient temperature, and if some of the items in the model make use of yield stress, such as vessel legs, then you should carefully check and enter this value.

When you select a material from the material database, the software looks up the material's operating yield stress in the yield stress database and automatically fills in this value. If there are duplicate entries in the yield stress database, then the software displays a message. You can then select from among the duplicates.

Allowable Stress, Design

Enter the allowable stress for the element material at operating temperature. The operating temperature for most vessels is defined to be the same as the design metal temperature for the internal pressure. You can find this value in the ASME Code, Section II, Part D, Table 1A, 1B, and 3.

If you enter a valid material name in **Material Input**, the software searches its database and determines the allowable stress for the material at ambient temperature, and populates this field.

NOTE The software also determines the allowable stress when you select a material name from the **Material Selection** window.

Allowable Stress, Ambient

Enter the allowable stress for the element material at ambient temperature. The ambient temperature for most vessels will be 70° F, 100° F, or 30° C. You can find this value in the ASME Code, Section II, Part D, Table 1A, 1B, and 3.

NOTE The software also determines the allowable stress when you select a material name from the **Material Selection** window.

Nominal Material Density

Enter the nominal density of the material. The software uses this value to calculate component weights for this analysis. The typical density for carbon steel is 0.2830 lbs/in^3.

Nominal Thickness for this P Number

Enter the thickness for the P number.

Table UCS-57 of the ASME Code, Section VIII, Division 1 lists the maximum thickness above which full radiography is required for welded seams. This thickness is based on the P number for the material listed in the allowable stress tables of the Code.

> **NOTE** If a seam is partially radiographed and the required thickness exceeds the P number thickness, P**V Elite** automatically changes the joint efficiency to 1.0 as stated in the Code.

External Pressure Curve Name

UCS-66 Curve

Is the Material Normalized?

Click to use the ASME normalized curve for the material. For more information, see **UCS-66 Curve**.

Elastic Modulus ID

The elastic modulus reference number is a value that points to or corresponds to a set of data set forth in ASME Section II Part D, tables TM-1, 2 and so on. Unfortunately, many materials have a composition or UNS number that does not match the criteria of what is supplied in the ASME Code. In these cases, the reference number will be brought in as zero. If this happens, you will need to enter in an appropriate value.

Thermal Expansion Coefficient ID

The thermal expansion reference number is a value that points to or corresponds to a set of data set forth in ASME Section II Part D, tables TE-1, 2 and so on. Unfortunately, many materials have a composition or UNS number that does not match the criteria of what is supplied in the ASME Code. In these cases, the reference number will be brought in as zero. If this happens, you will need to enter in an appropriate value. Thermal expansion coefficients are important especially if you are analyzing a heat exchanger.

Yield Stress

Opens the **Yield Stress Record** dialog box, which displays yield stress details of the selected material.

APPENDIX A

Vessel Example Problems

Example problems are located in the program installation folder/Examples folder.

APPENDIX B

Keyboard and Mouse Commands

Keyboard

The following software actions are defined for the keyboard:

Begin line	<Home>
Begin list	<Home>
Delete character	<Del.>
Delete prev. char	<Backspace>
Delete window	<Alt+F4>
End line	<End>
End list	<End>
Exit	<Shift+F3>
Help	<F1>
Hot key	<Alt+char(with '_')>
Insert toggle	<Ins>
Left word	<Ctrl+left-arrow>
Mark	<Ctrl+F5>
Maximize	<Alt +>
Menu control	<Alt>
Minimize	<Alt ->
Move window	<Alt+F7>
New model	Ctrl+N
Next cell	<Down_arrow>
Next Character	<Right_arrow>
Next field	<Tab>
Next window	<Alt+F6>

Page down	<Page Down>
Page up	<Page Up>
Previous cell	<Up_arrow>
Previous character	<Left_arrow>
Previous field	<Shift+Tab>
Refresh	<F5>
Right word	<Ctrl+right_arrow>
Select	<Enter>
Size window	<Alt+F8>
System button	<Alt .>

Mouse

The following software actions are defined for the mouse:

In Window Objects:	
Choose	<Left-down-click>
Select	<Left-release>

In Vessel Graphics:	
Select element	<Left-release>
Select detail	<Right-down-click>

Glossary

remaining strength factor

FCA

future corrosion allowance

gl

Length of groove-like flaw.

gw

Width of groove-like flaw.

LTA

Local thin area.

MAPnc

Maximum Allowable Pressure in a new and cold condition

MAWP

Maximum Allowable Working Pressure

Slen

Maximum length between ring stiffeners

wmax

maximum pit depth

Index

M

N

O

P

R

S