Pressure Vessel Design Procedure

by 3D-LABS

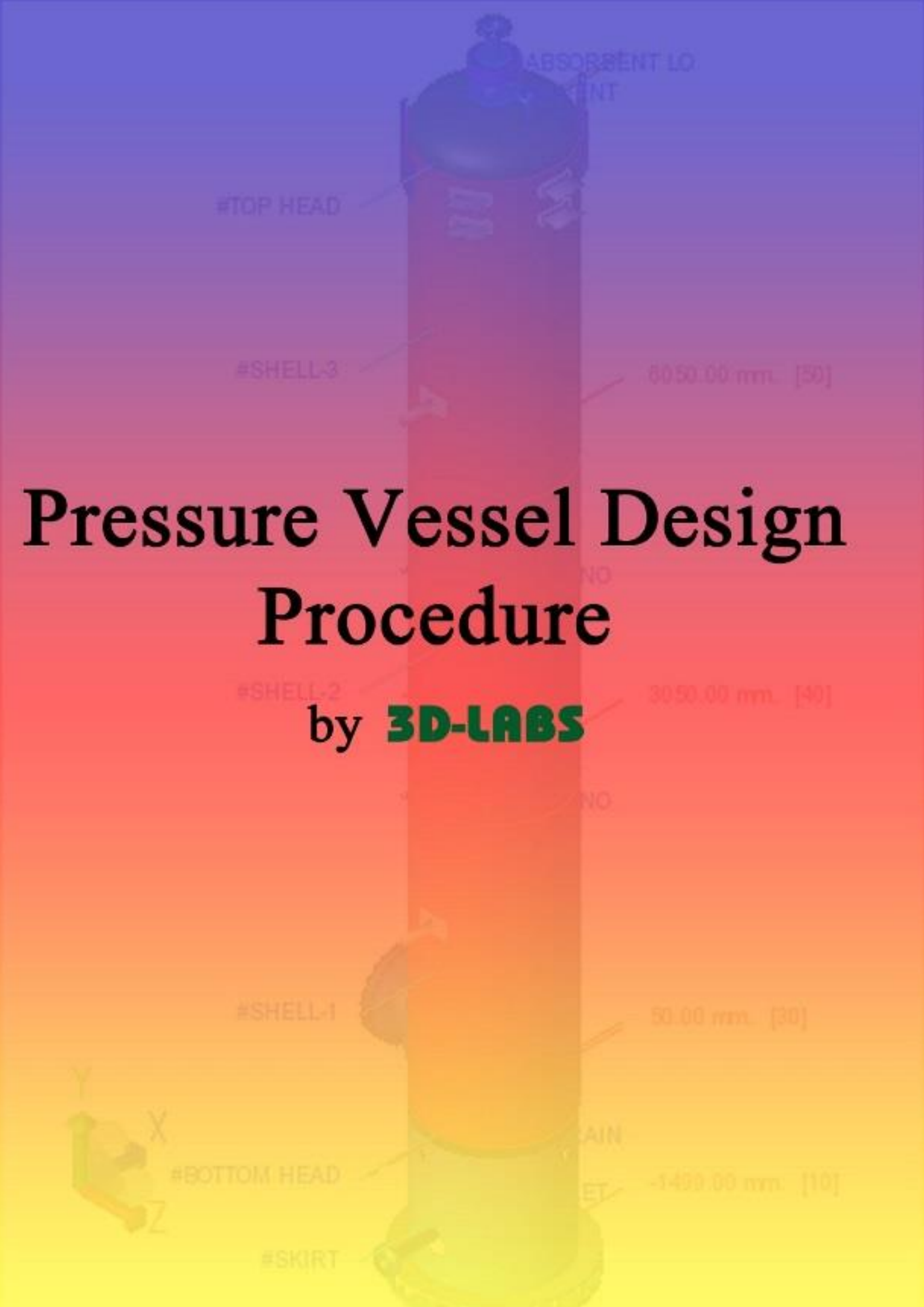


Table of Contents

1. INTRODUCTION

PV Elite is a complete solution for vessel and heat exchanger design, analysis and evaluation. Users of PV Elite have designed equipment for the most extreme uses and have done so quickly, accurately and profitably.

In this article we are going to discuss about the PV Elite sotware.

2. ANALYSIS OPTIONS

PV Elite performs calculations in accordance with ASME Section VIII, Divisions 1 & 2, PD 5500, and EN 13445.

We can select the required code for analysis of an equipment i.e vessel, heat exchanger.

After opening the PV Elite, we need to select the code for which the equipment has to be design and analyse.Using **New icon** at **file pannel** in the **home tab**.

Open PV Elite

Click New

Select the Code

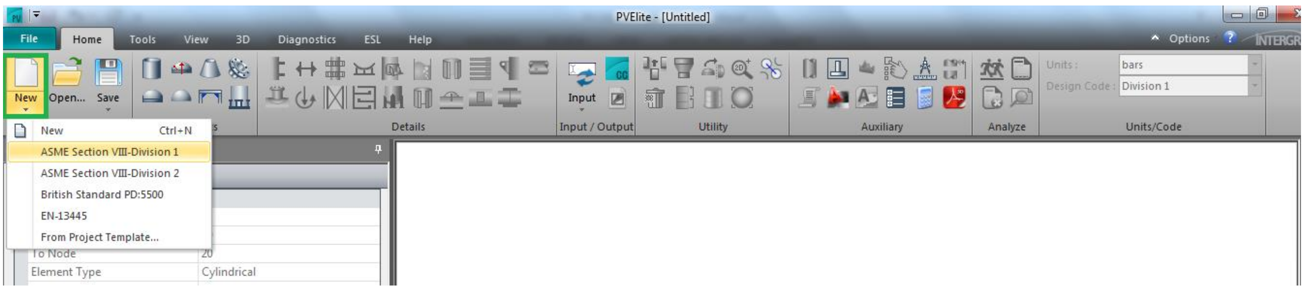
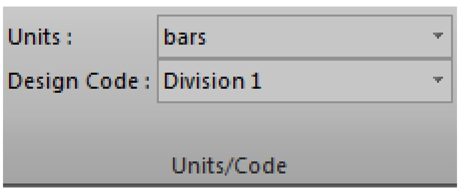


Figure 1 Create a design model file

Also we can use the **Units/Code** panel to select or change the code and units of a vessel.



3. DATA COLLECTION

PV Elite makes defining pressure boundary conditions for vessels and exchangers easy, even for load sets that require significant data input. PV Elite streamlines data entry by breaking the input down into sensible subsets. Help on any input item is only a keystroke away.

3.1 DESIGN DATA

The below (refer figure 2) image shows the home tab of the PV Elite software. In this the highlighted (bottom left corner) options are the basic input option for an equipment.

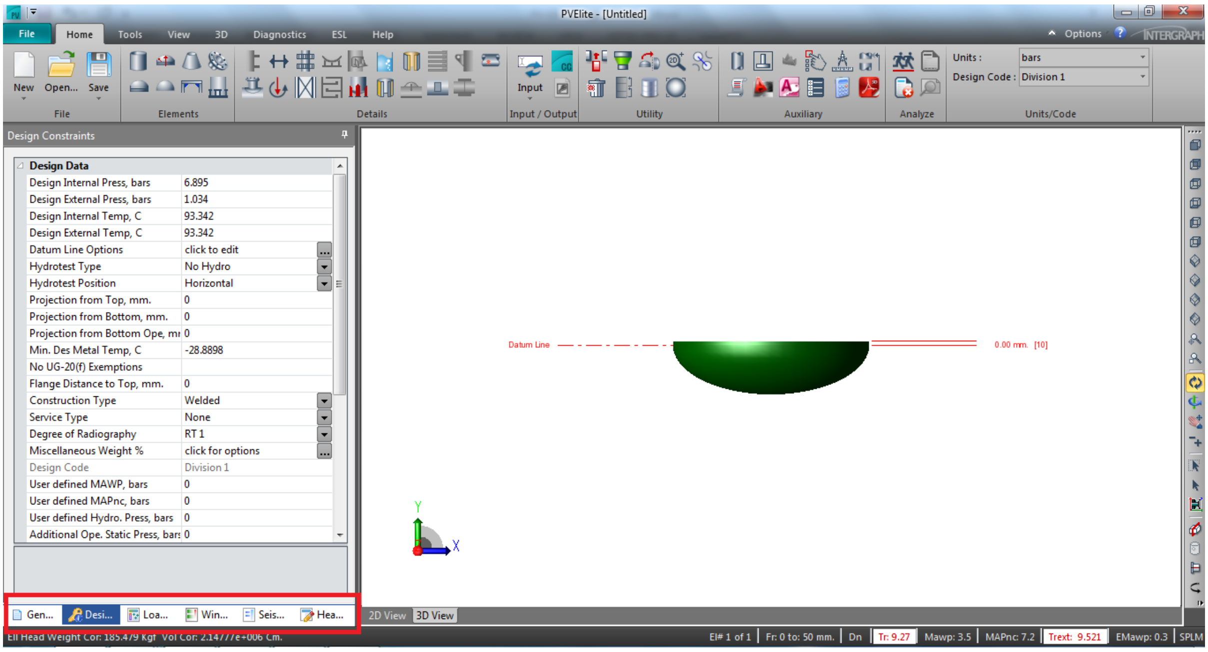


Figure 2 Home tab with Design data

The followings are the basic input data,

- General Input
- Design data
- Load cases
- Wind data
- Seismic data
- Heading

Here we are going to know about design data.Before filling design data we have to gothrough the inputs i.e datasheet, project specification and other required inputs.

Some important details in design data option,

Design internal pressure:

The pressure of the equipment that holds a liquid or gas.Internal design pressure plays an important role in design and analysis of the vessel.

Design external pressure:

If the equipment subject to external pressure, we need to fill the external pressure.

Design internal temperature:

The internal temperature of the equipment plays important role for selecting material allowable stress values for thickness calculations.

Design external temperature:

Enter the design external pressure if any, otherwise fill the internal temperature in that place.

Datum line option:

Datum line is reference line which is assigned by the design engineer for placing the nozzle or cleat or other things in an equipment.

The datum line is an optional reference position for nozzle and some other vessel detalis.

Just click on the datum line option, a Datum line options dialouge box (refer figure-3) will appear. Using the option in that dialouge box, we can assign the datum line for an eqipment.

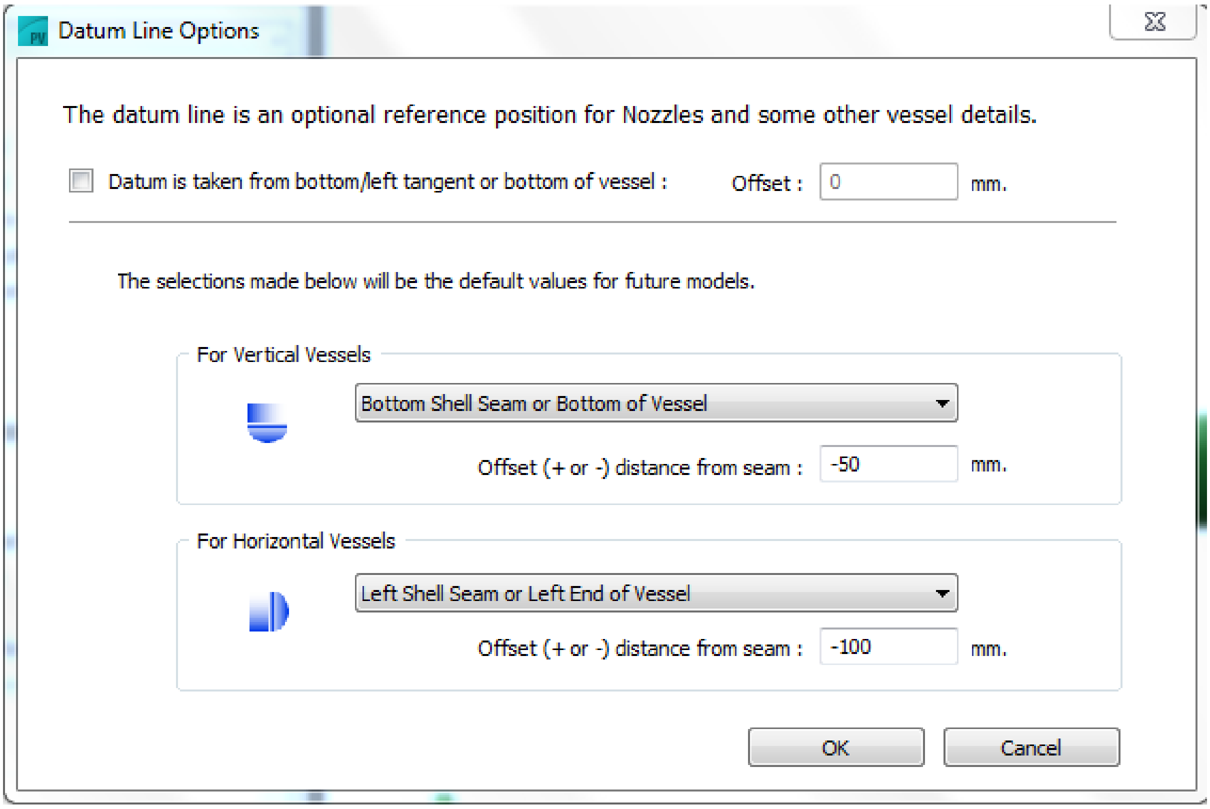


Figure 3 Datum line option dialouge box

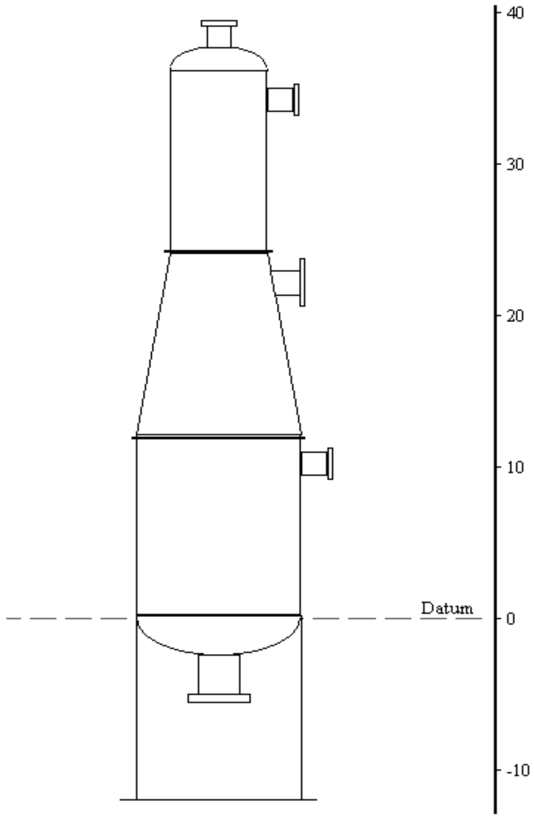


Figure 4 Vessel with datum line

Datum line option dialouge 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

Hydrotest type:

Select the hydrotest type based on the design code and testing requirements.

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 the hydrotest position in which the vessel will be tested.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.

Minimum Design Metal Temperature:

The minimum design metal temperature (MDMT) is one of the design conditions for pressure vessels engineering calculations, design and manufacturing according to the ASME Boilers and Pressure Vessels Code.

The entered value of MDMT does not affect the calculation of the vessel, it just compare the calculated value with enterd MDMT value. This value is listed in the Internal Pressure Calculations report for comparison with the calculated UCS-66 minimum temperature.

UG-20f 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**,
- **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**
- **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

Miscellaneous Weight %:

Click  to open the **Miscellaneous Weight Percent Inclusion** dialog box

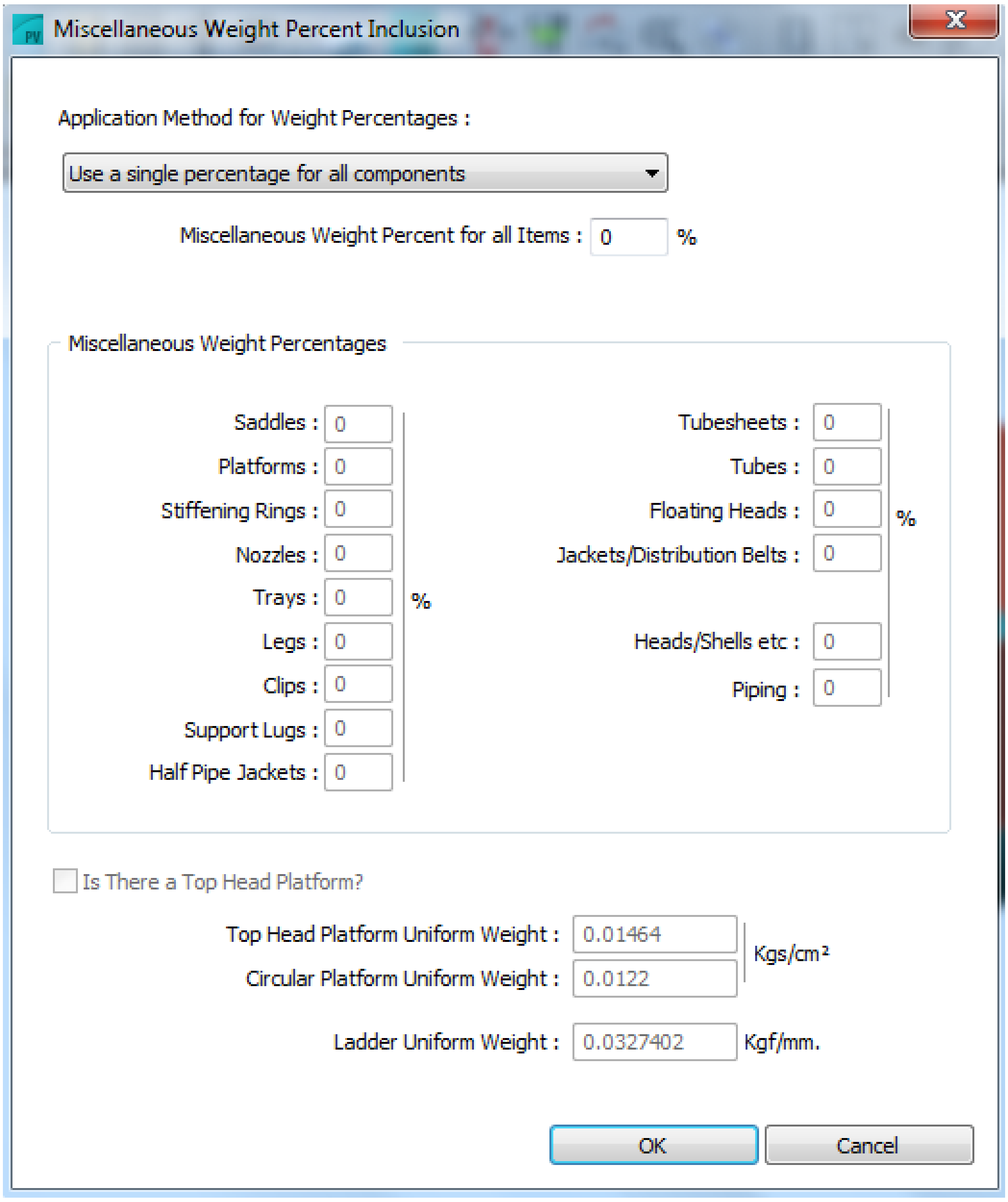


Figure 5 Miscellaneous Weight percent inclusion dialouge box

This option is used to increase the weight of the various components in an equipment.

- **Use a single percentage for all components:** using this option, we can increse the weight of the all metalic components. Enter the percentage value at **Miscellaneous weight percent for all items** box to add additional weight.

- **Use individual percentages for the different components:** using this option, we can increase the weight of an item by the percentage specified for the item in the **Miscellaneous Weight Percentages** section.
- **Add percentages to increase weights of shells and heads:** using this option, we can increase the weight of shells and heads by the sum of the percentage increase for nozzles, clips, and piping.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.

User Defined MAWP:

MAWP is the highest pressure at which the equipment may be operated at its design temperature.Using this option we can 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.. 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.

No Vortex Shedding for H/D <= 15:

For vertical vessels in which the height to diameter ratio 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 component is a heatexchanger, select this option.

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.

3.1.1 DESIGN MODIFICATION

Select Wall Thickness for Internal Pressure:

This option (**Yes**) is used to automatically set the wall thickness of a component for internal pressure.When the required element thickness for internal pressure exceeds the value of Finished Thickness defined for Element Data on the General Input tab, then 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 configuration Dialog Box.

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:

This option (**Yes**) is used to automatically set the wall thickness of a component for external pressure. 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 and increases the value of Finished Thickness. The software notifies you when the thickness is changed.

Select Wall Thickness for External Pressure option value is **Yes**, then *Select Stiffening Rings for External Pressure* is set to **No**.

Select Stiffening Rings for External Pressure:

This option (Yes) is used to automatically add stiffeners for external pressure. The software calculates the location and size of stiffening rings needed for external pressure requirements.

When the program calculates the allowable length between stiffeners, the result must be a reasonable value.

If the value is too small, the rings are not added. In this case, you must increase the thickness of the shell and try the stiffening ring design again.

Select Stiffening Rings for External Pressure option value is **Yes**, then *Select Wall Thickness for External Pressure* is set to **No.**

Select Wall Thickness for Axial Stress:

This option (Yes) is used to automatically increase the thickness for axial stress in vertical vessels. The software calculates the axial stress and required thickness of each element 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 Configuration Dialog Box.

3.2 LOAD CASE

Stress Combination Load Cases:

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.

The software provides the following set of default load cases refer figure 6.

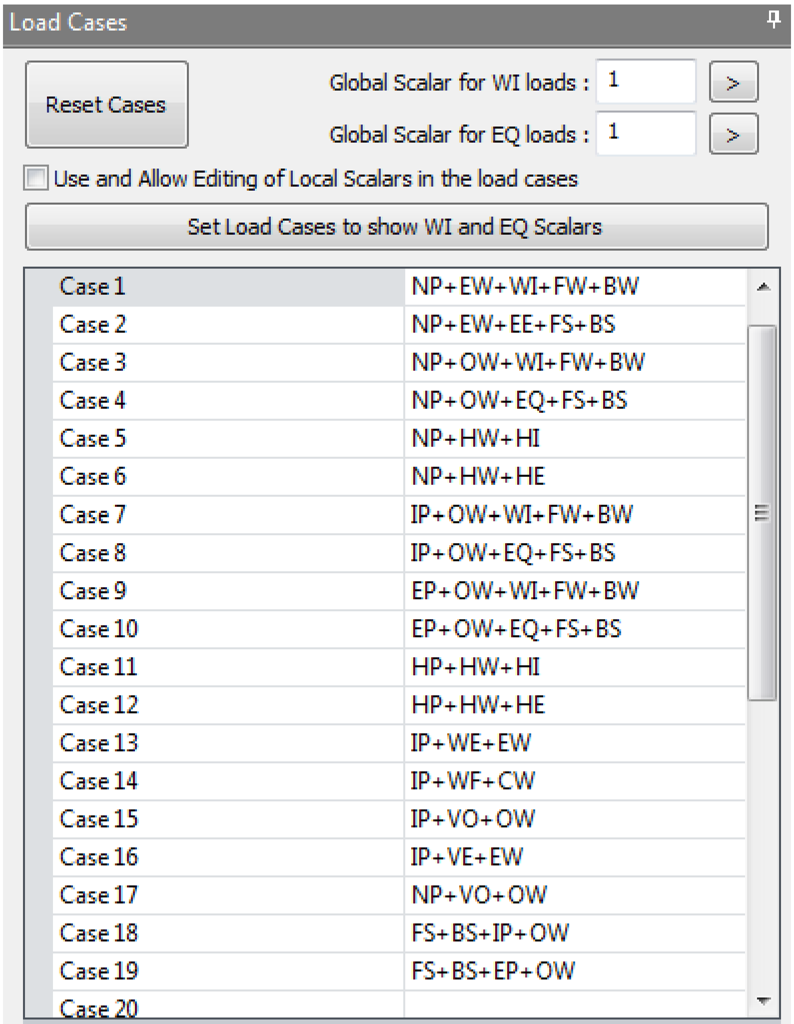


Figure 6 Stress combination load cases

<u>The following abbreviations are used for loads:</u>

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

If Consider Vortex Shedding is selected on the Design Constraints Tab**,** 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.)

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.

Global Scalar for EI 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.

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.

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 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.

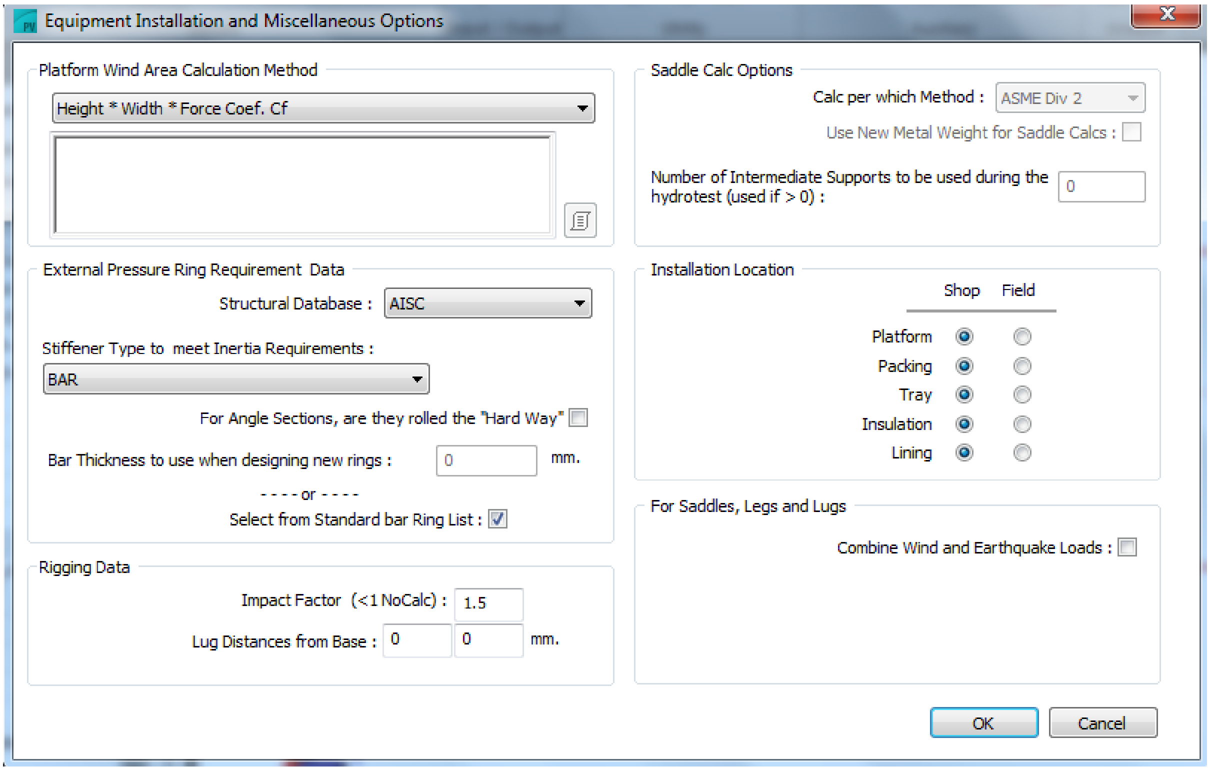


Figure 7 Equipment Installation and Miscellaneous options

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?**(**refer figure 9)in the **Nozzle Input Analysis** dialog box for each nozzle.

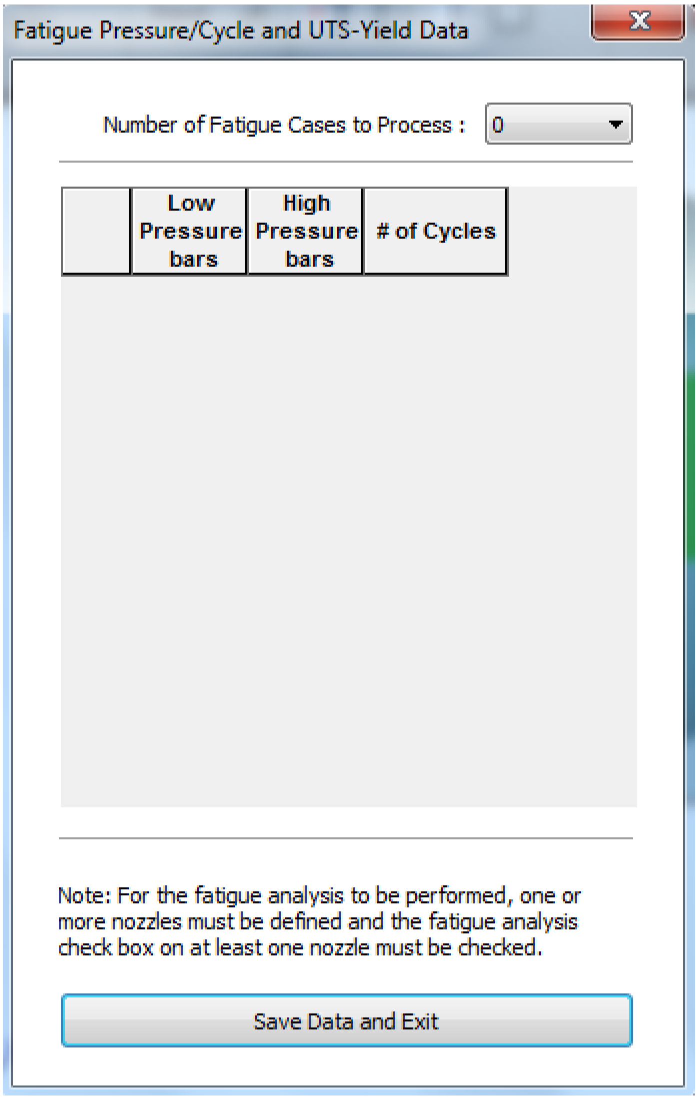


Figure 8 Fatigue analysis

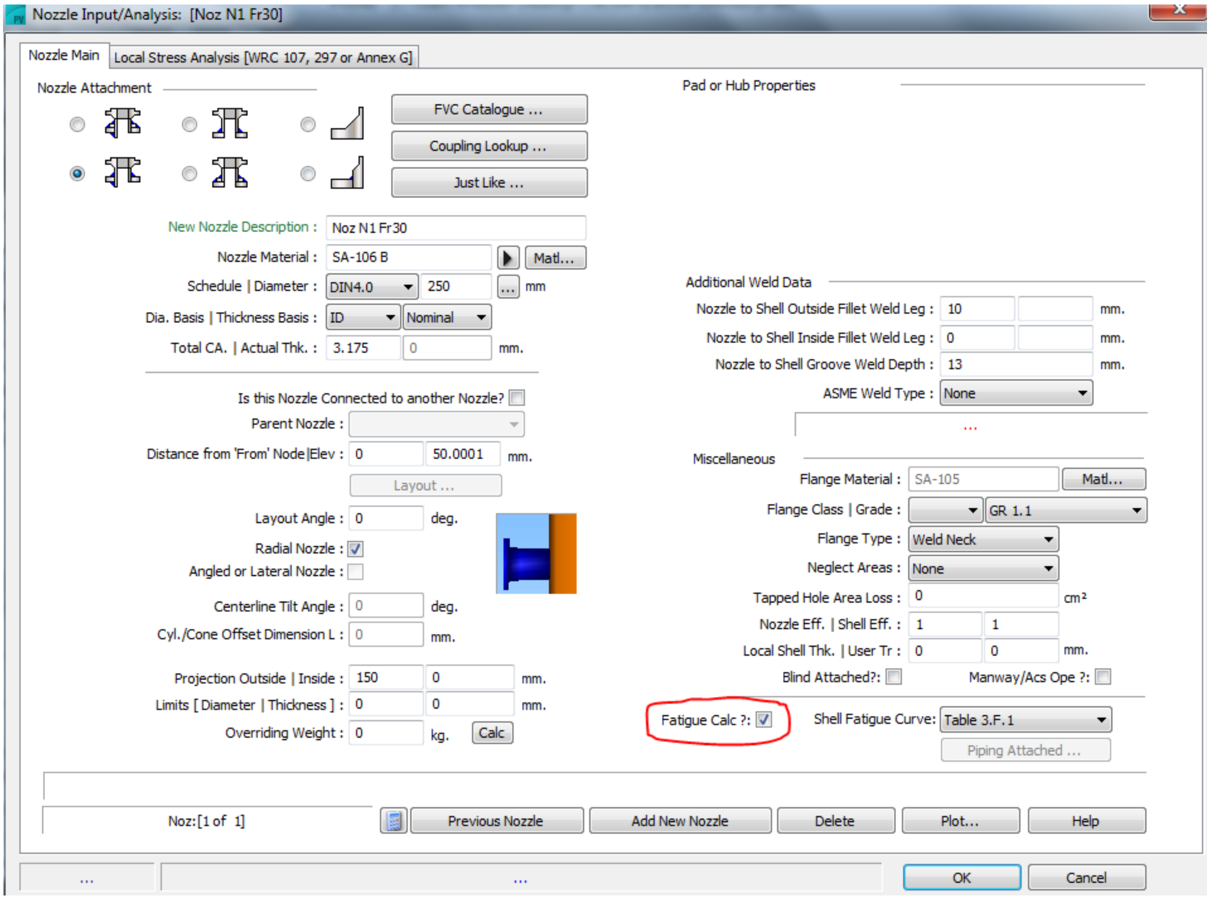


Figure 9 Nozzle input dialouge box (Fatigue calc.)

3.2.1 NOZZLE DESIGN OPTION

Nozzle/Clip Design Pressure Options:

Select one of the following design pressure methods for calculating the pressure at the nozzle:

- 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.
- 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 the most conservative option.
- 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 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 Input** 

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.

- **By Name Ascending**
- **By Name Descending**
- **By Diameter**
- **Ascending**
- **By Diameter**
- **Descending**
- **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.**

3.3 WIND LOAD DATA

The wind load details are filled using this **wind load data tab.** We can select the required code (refer figure 10) from the available code list.

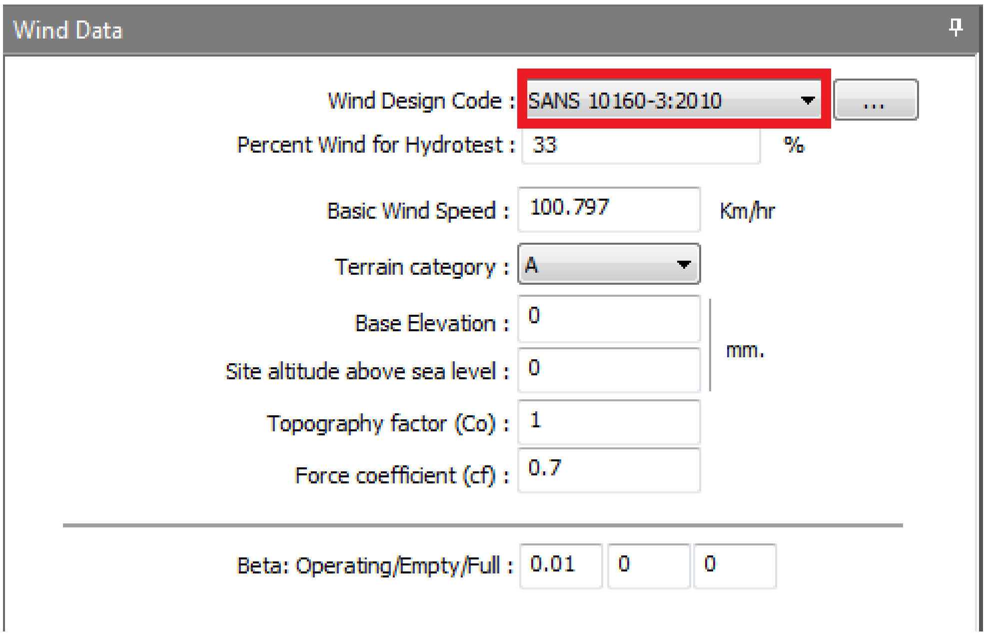


Figure 10 Wind data tab

3.4 SEISMIC LOAD DATA

The seismic load details are filled using this **Seismic load data tab.** We can select the required code (refer figure 11) from the available code list.

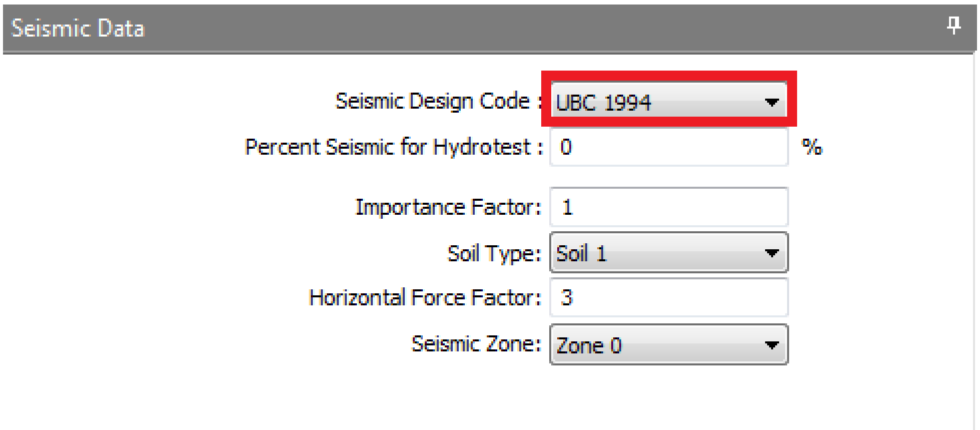


Figure 11 Seismic data tab

3.5 HEADING

This informations are reflected in the report only.We can add the project name and other details using this option.

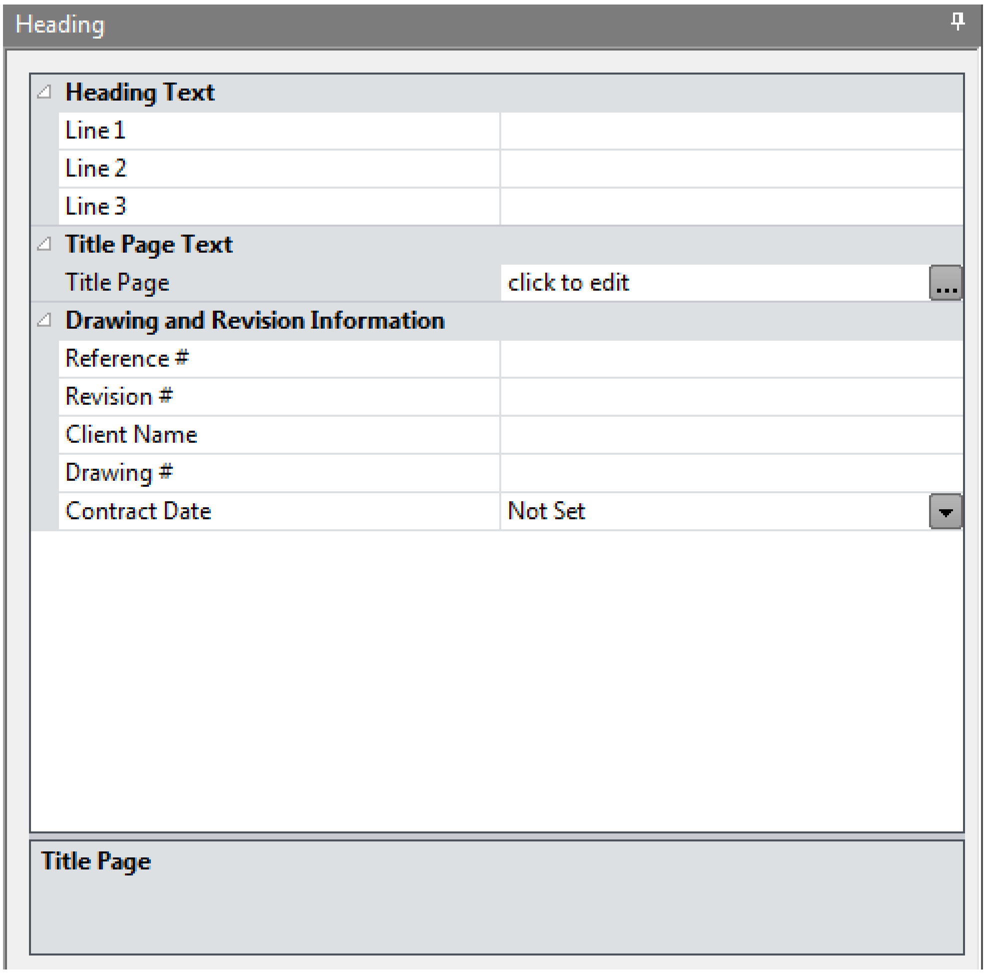


Figure 12 Heading

3.6 GENERAL INPUT

This tab is used for model a component of an equipment.The following are the details which are to be entered by a designer.

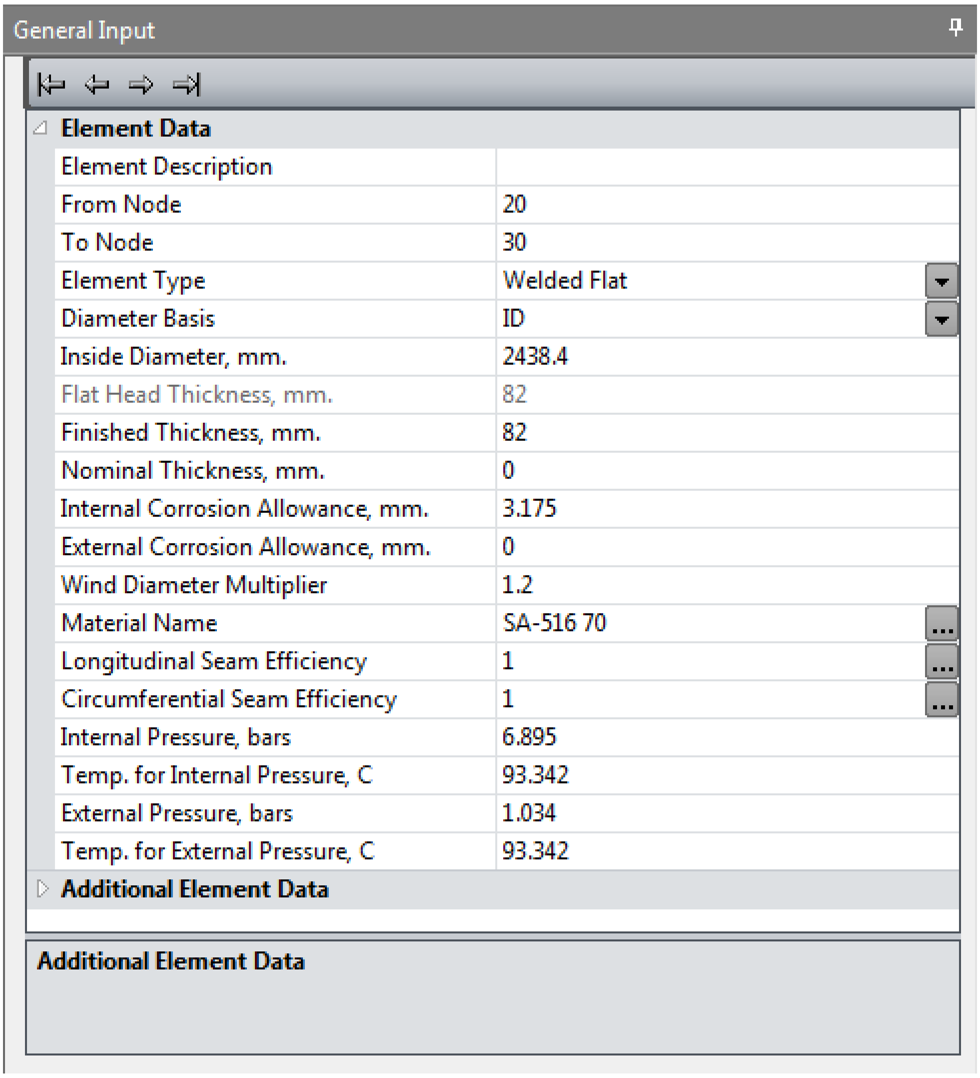


Figure 13 General Input tab

From node / To node:

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.

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.

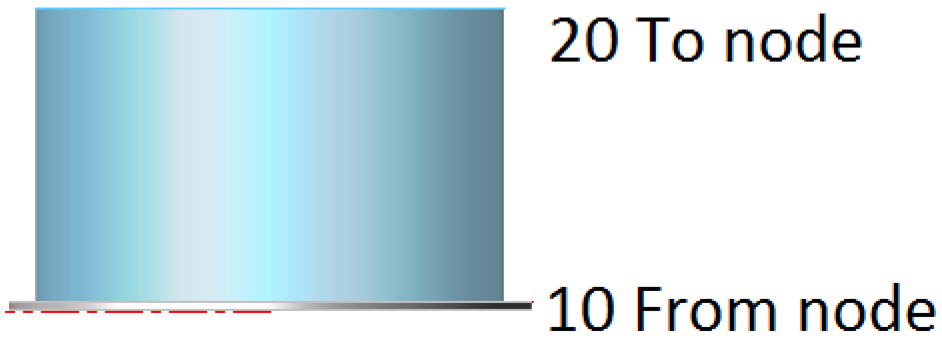


Figure 14 Example of from and to node

Element Type:

Displays the type of the element. You can not create a new element by using this but you can change the type by selecting one of the following from the list.

If the element type is changed, all detail data, such as nozzles, are lost.

- **Cylinder** - A cylindrical shell
- **Elliptical** - An elliptical head
- **Torispherical** - A Torispherical 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

Diameter Basis:

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

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.

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

"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.

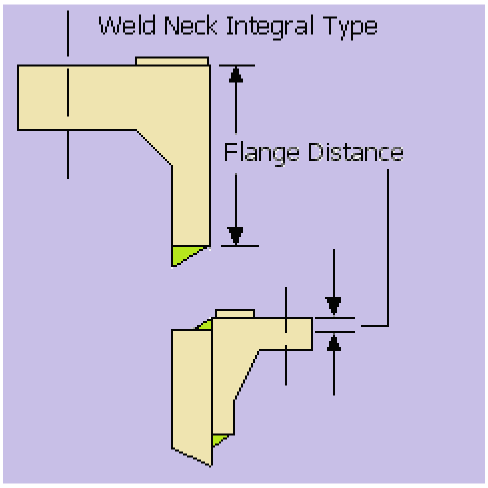
This option is only available when **Elliptical**, **Torispherical**, or **Spherical** are selected for Element Type.

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.

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.

Internal Corrosion Allowance:

If the vessel having internal corrosion allowance, have to enter the value.

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.

External Corrosion Allowance:

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.

Wind 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, 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.

Material Name:

PV Elite is a global package with international code rules plus extensive region-specific content.

We can enter the name of the material for a component. 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.

Alternatively, you can click **Select Material** on the **Utilities** toolbar to select a material directly from the Material Database Dialog Box.

To modify the material properties of the selected element, click to open the Material Properties Dialog Box (refer figure 15). Doing so only changes the properties of this element for this analysis. It does not modify the database.

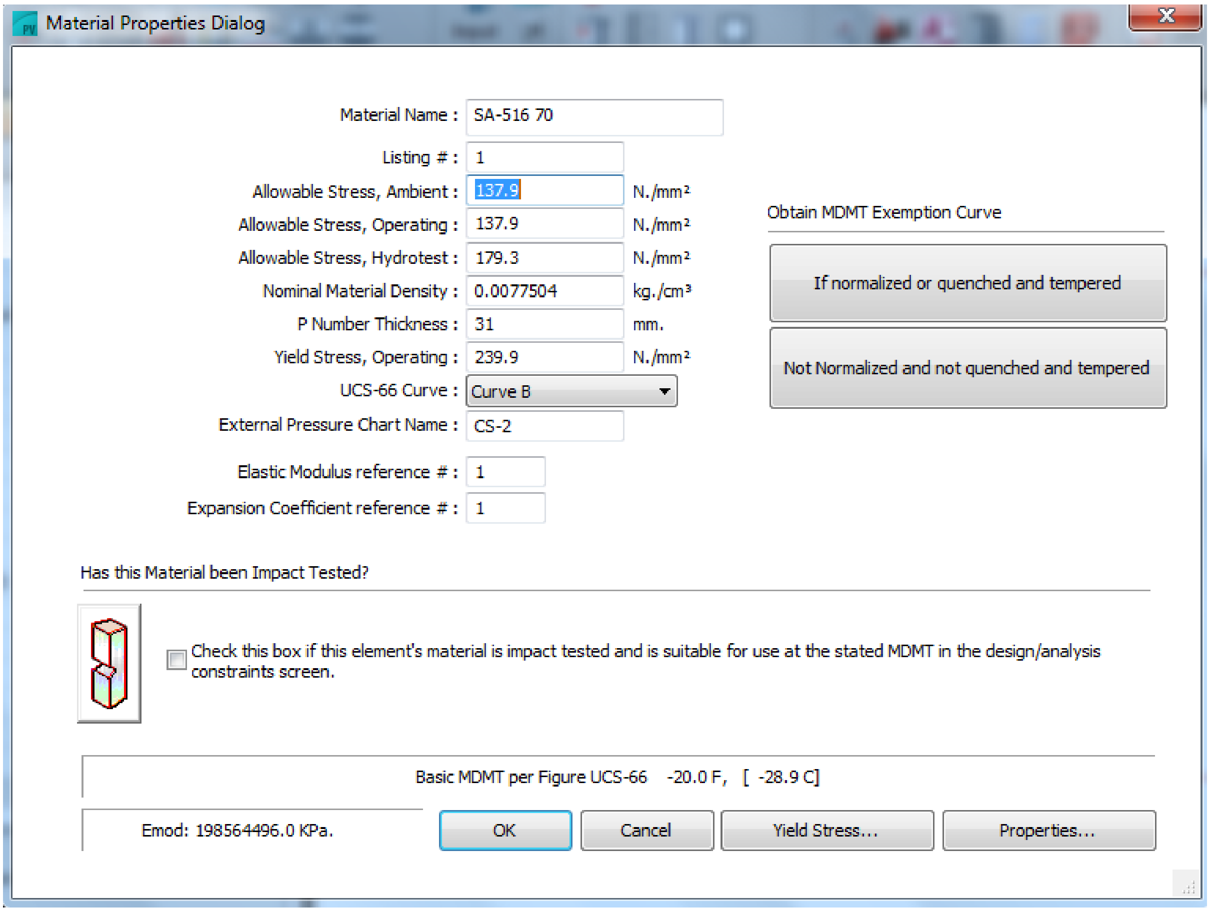


Figure 15 Material Properties Dialog Box

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** for determining this value.

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** for determining this value.

3.6.1 ADDITIONAL ELEMENT DATA

Here we are going to see some important additional element details for model a component of an equipment.

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. This option is only available when Elliptical is selected for Element Type.

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.

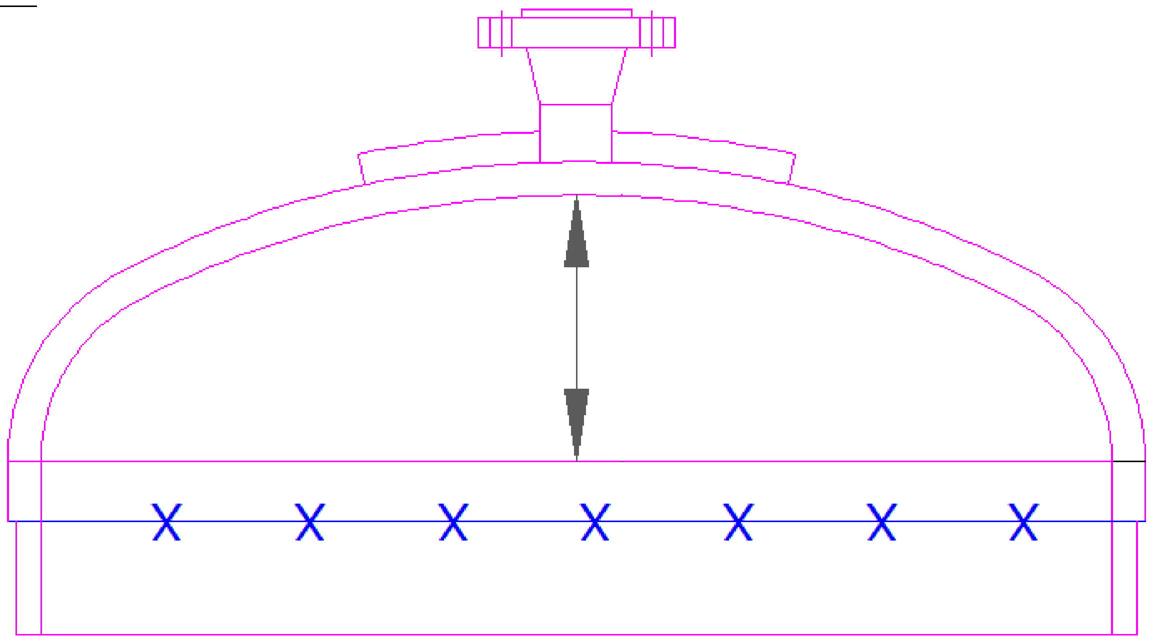
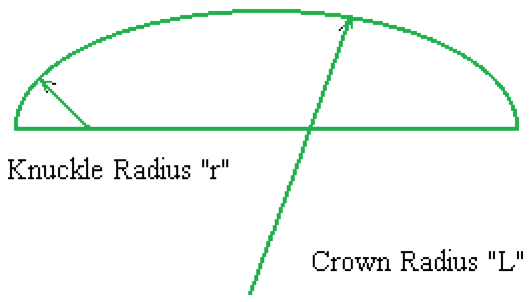


Figure 16 Elliptical head

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, and **Elliptical**, **Torispherical**, or **Spherical** are selected for Element Type.

'To' End Inside 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. This option is only available when **Conical** is selected for Element Type.

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.

This option is only available when **Conical** is selected for Element Type.

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.

This option is only available when **Conical** is selected for Element Type**.**

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.

This option is only available when **Conical** is selected for Element Type.

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** 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. This option is selected by default and is only available when you select **Conical** for the Element Type.

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 and is only available when **Conical** is selected for Element Type.

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. This option is only available when **Welded Flat** is selected for Element Type.

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)

Consult Paragraph UG-34 before using these values.

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. This option is only available when **Welded Flat** is selected for Element Type.

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. 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. This option is only available when **Welded Flat** is selected for Element Type.

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** (refer figure 17) opens. This option is only available when **Body Flange** is selected for Element Type.

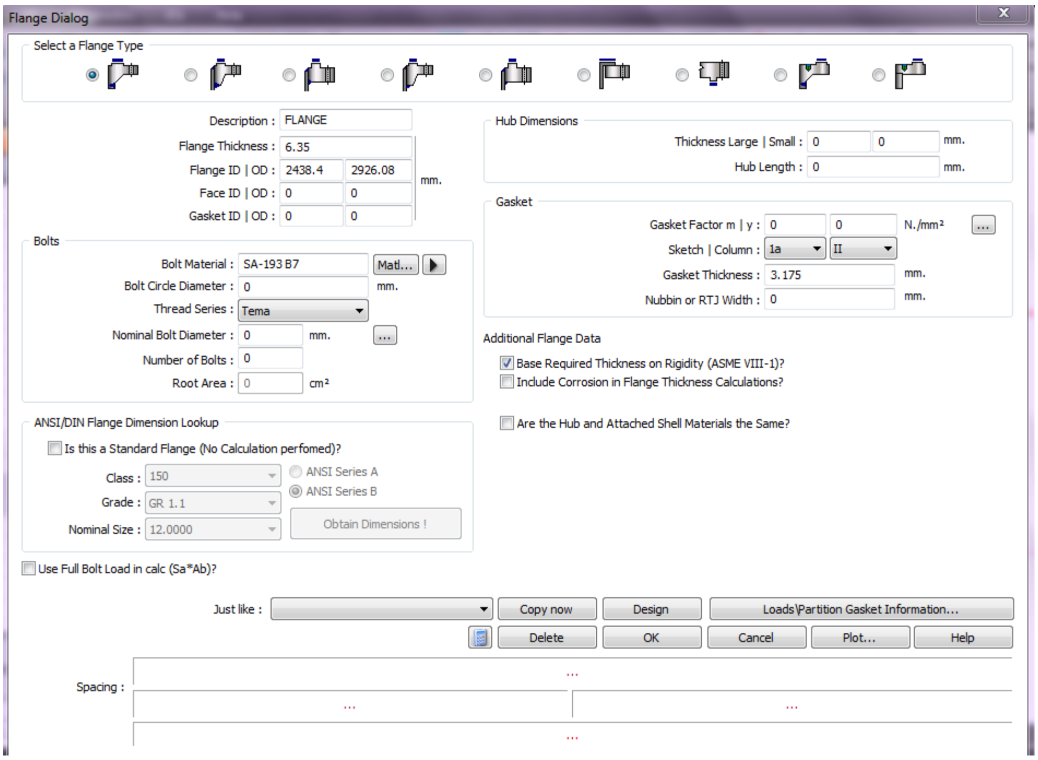


Figure 17 Flange Dialog Box

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** opens. This option is only available when **Skirt** is selected for **Element Type**.

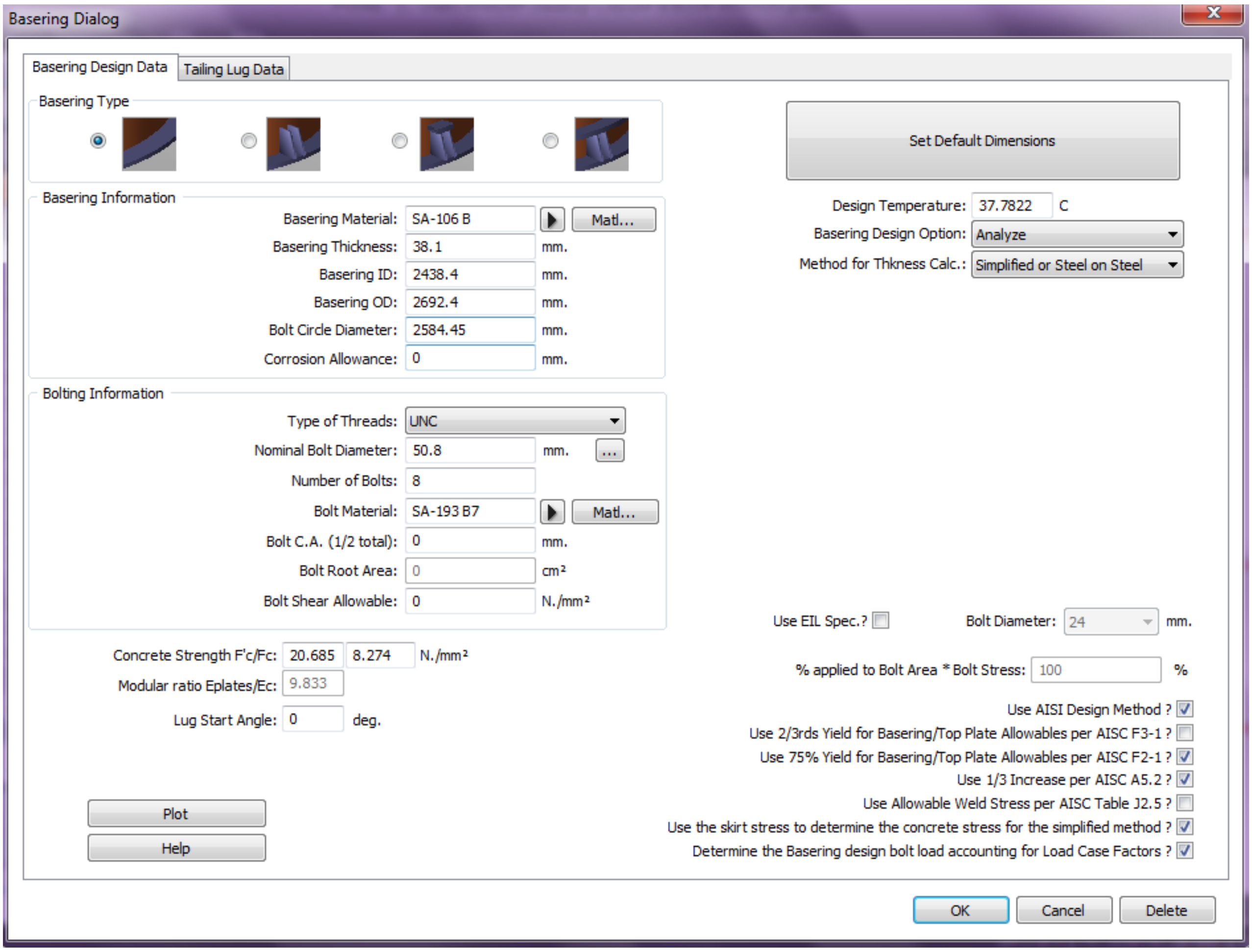
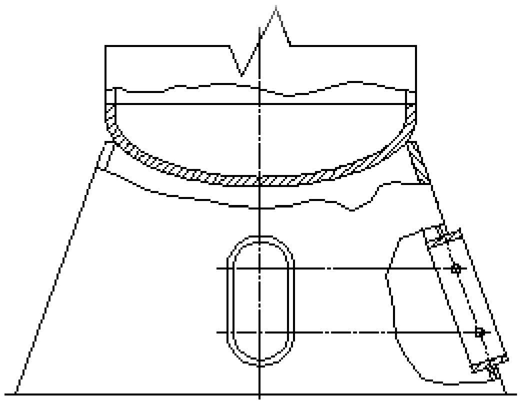


Figure 18 Basering Dialog Box

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**. Enter the following parameters for each opening: Number of Skirt Openings, Width, Height, Center Spacing, Frame Thk, Frame Width, and Layout Angle.

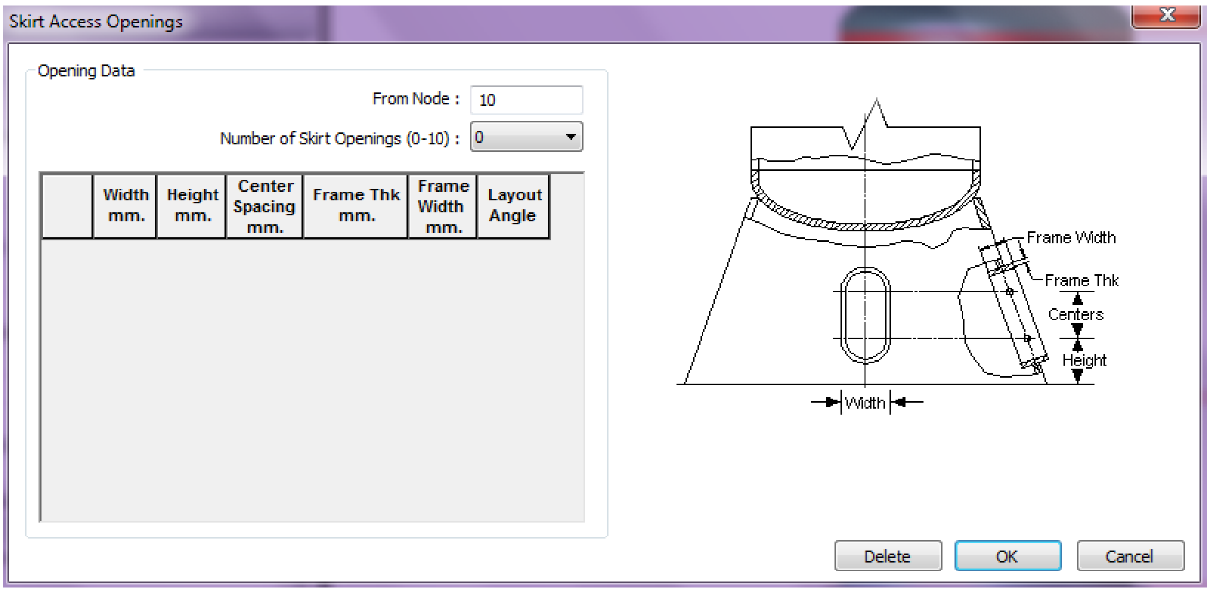


Figure 19 Skirt Access Openings Dialog Box

4. MODELING & DESIGNING OPTIONS

The following options are used for modeling an equipment.Here we have some toolbars in PV Elite which is used for model and design an equipment.

4.1 Elements panel

In PV Elite home tab, the Element tool bar(panel) is placed top left corner.using the options in the Element tool bar, we can model a component of an equipment.

Figure 20 Elements tool bar

Ellipse:

Using this  option and general input data tab we can model an elliptical head of an eqipment.

In general input data tab we need to give diameter basis, diameter & length of the straght flange and thickness of the head. For more information refer chapter 3.6 of this article.

Torisherical:

Using this  option and general input data tab we can model a torispherical head of an eqipment.

In general input data tab we need to give diameter basis, diameter & length of the straight flange and thickness and material of the head.For more information refer chapter 3.6 of this article.

Sherical:

Using this  option and general input data tab we can model a spherical head of an eqipment.

In general input data tab we need to give diameter basis, diameter & length of the straight flange and thickness and material of the head.For more information refer chapter 3.6 of this article.

Welded Flat head:

Using this  option and general input data tab we can model a flat head of an eqipment. In general input data tab we need to give diameter basis, diameter, finished thickness and additional element data like **Welded Flat Head Attachment Sketch** and **Attachment Factor** etc.

For more information refer chapter 3.6 of this article.

Ansi/Bolted flanges (Body flange):

Using this  option and general input data tab we can model a body flange of an eqipment.

Cylinder:

Using this  option and general input data tab we can model a cylindrical shell of an eqipment.

In general input data tab we need to give diameter basis, diameter & length of the shell and thickness and material of the shell.

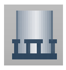
Conical shell or Head:

Using this  option and general input data tab we can model a conical shell or head of an eqipment.

In general input data tab we need to give diameter basis, diameter & length of the cone and thickness and material of the cone and additional element data like **'To' End Inside Diameter, Line of Support Options** etc.

For more information refer chapter 3.6 of this article.

Skirt:

Skirt is one type of support used for vertical vessels. Using this  option and general input data tab we can model a skirt of an eqipment.

In general input data tab we need to give diameter basis, diameter & length of the skirt and thickness and material of the skirt and additional element data like **Skirt Diameter at Base, Perform Basering Analysis** and **Evaluate Holes in Skirt.**

4.2 Details panel

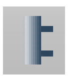
This tool bar(panel) (refer figure 21) is used to add additional components details like nozzles, stiffening ring, platforms, vessel packing details, saddle support details , lug support details ,leg support details ,vessel lining details, insulation details, force/moment input , weight and piping input , tray input ,tube sheet , lifting lug , clips ,working fluid detail and jacket or vapour distribution belt , half pipe jacket or limpet coil.

Figure 21 Details tool bar

Stiffening rings:

Stiffening rings can prevent the pressure vessel from buckling or increase the buckling pressure.

Stiffening rings are attached either internally or externally to the vessel or heat exchanger.

Using this  option we can get the stiffening ring dialouge box (refer figure 22). Enter the required details in the dialouge box to add the stiffener.

Some main inputs are ring location i.e OD or ID, distance from ‘from node’, ring type, ID & OD of the ring.

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.

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

Options in the Stiffening ring dialouge box,

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.

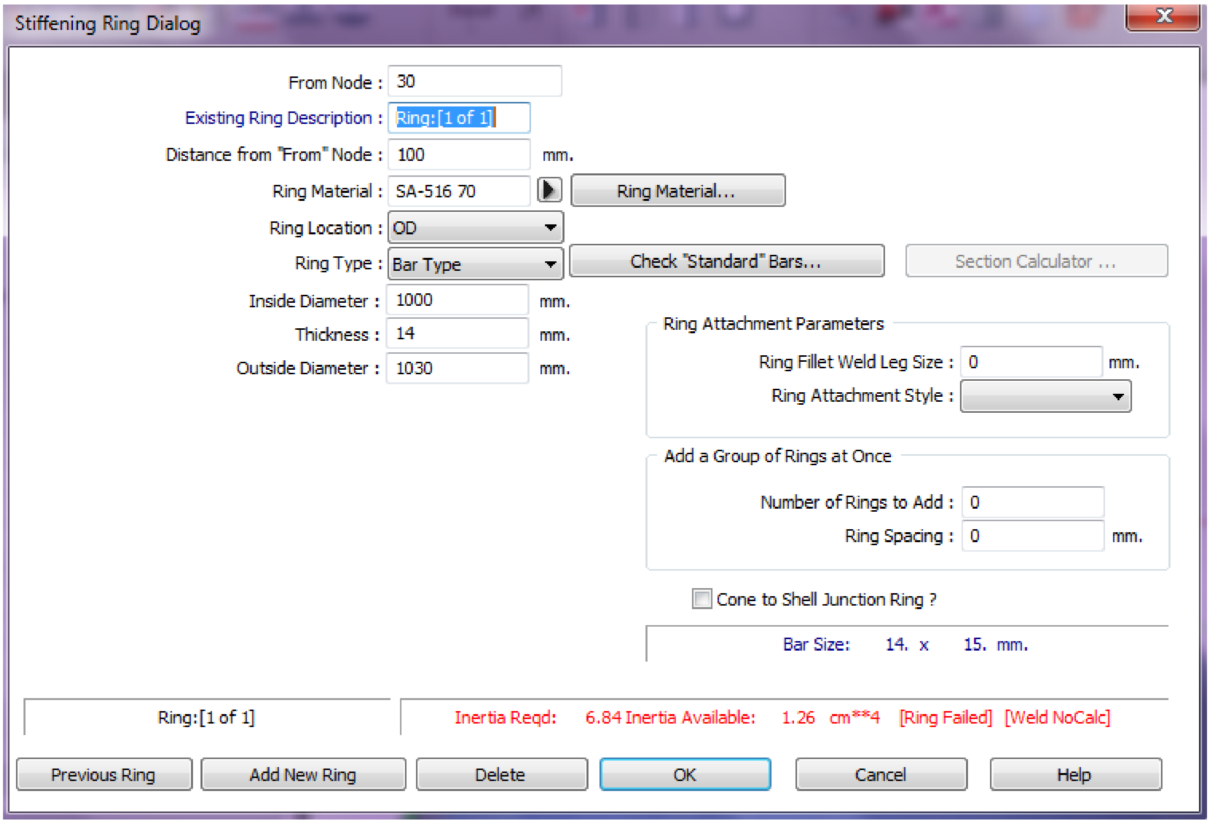


Figure 22 Stiffening ring dialouge box

Nozzle:

A **nozzle** is a pipe or tube or coupling with or without a flange and it can be used to allow the flow of a fluid (liquid or gas) into vessel or from the vessel and also for instrumentation installation purpose.

Using this  option we can get the stiffening ring dialouge box (refer figure 23). Enter the required details in the dialouge box to add the Nozzles.

The Nozzle stress analysis dialouge box also having the option for local stress analysis refer figure 25 (external loads).Using this option we can perform WRC-107, WRC-297 and PD5500 for a nozzle.

In nozzle main dialouge box option only, we have to enter the nozzle size and schedule (thickness) and which type of nozzle (like with/without pad, radial/hillside, heavy barrel…) and layout angle, outside projection, flange type/rating.

Previous

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

Add Platform

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

Delete

Deletes all data for the current Nozzle.

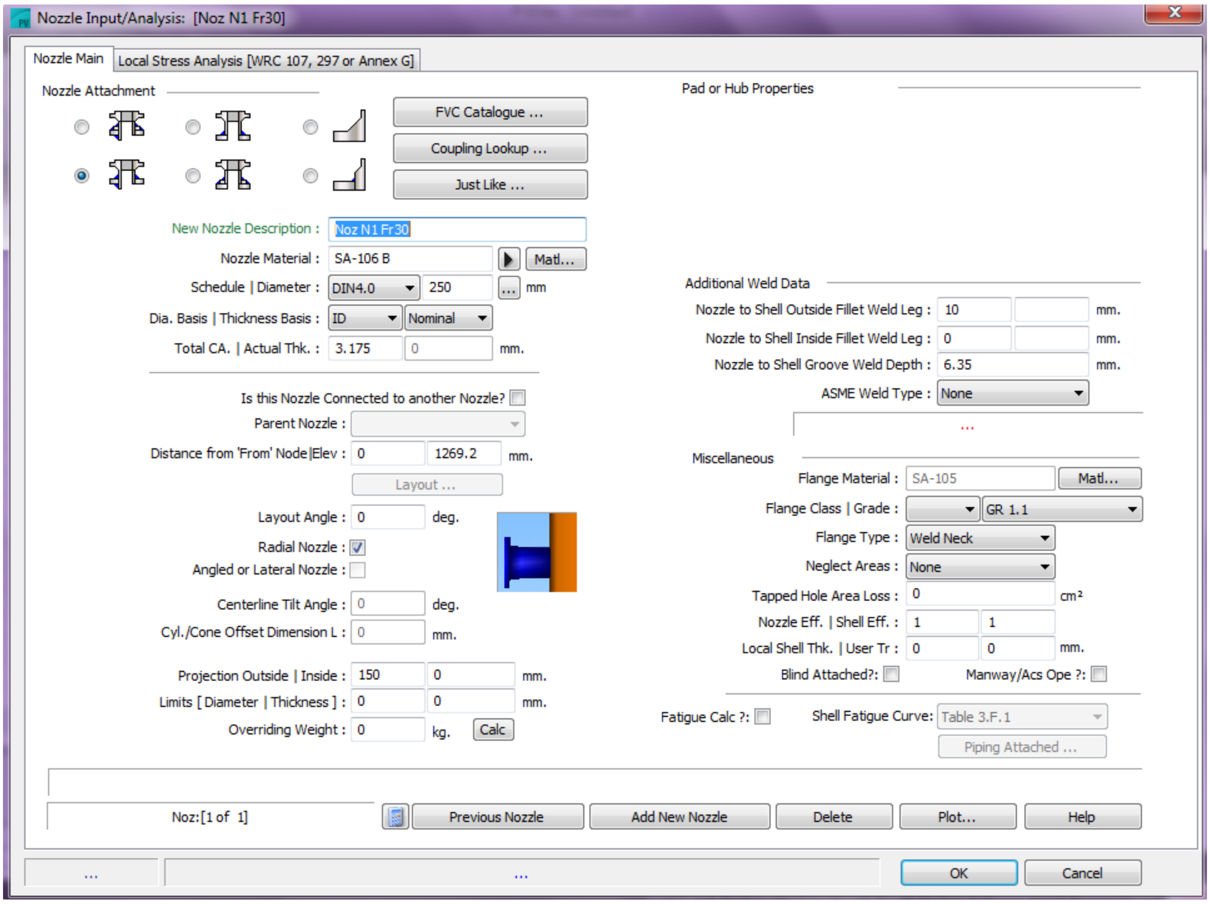


Figure 24 Nozzle stress Analysis dialouge box

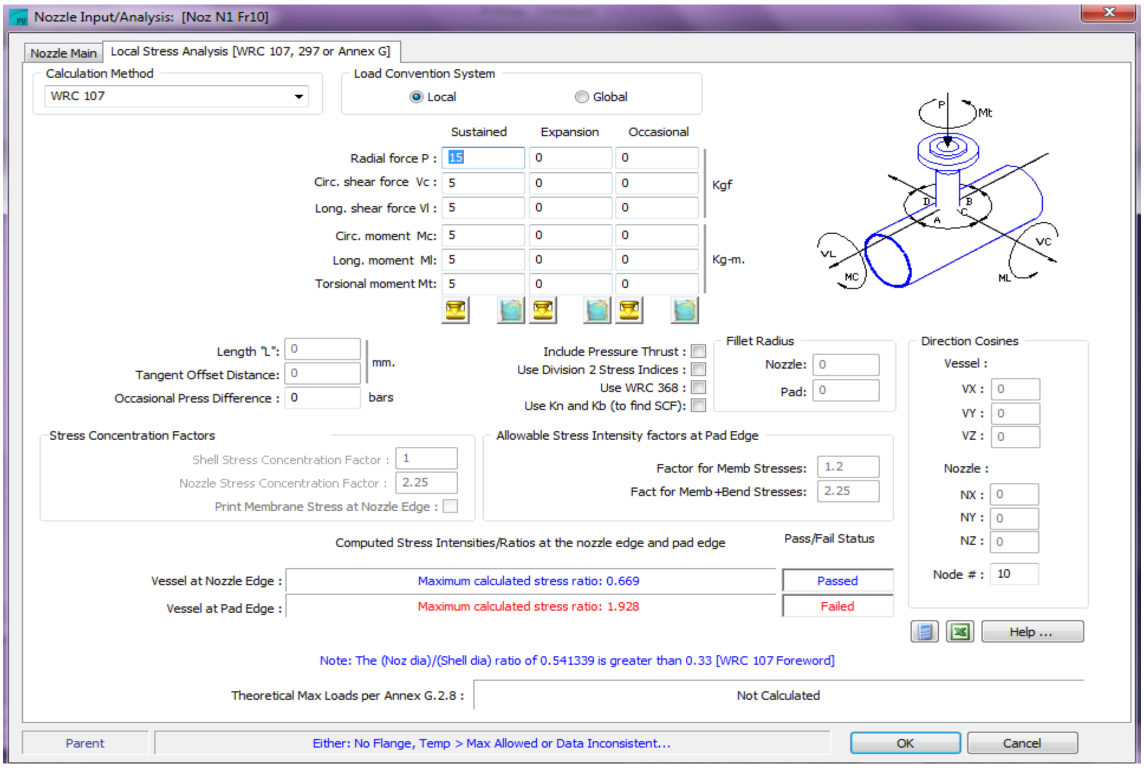
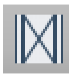


Figure 23 Local stress analysis dialouge box

Packing:

If a vessel having internals like packing and ceramic balls, we need to add the weight of that packing to the vessel.

Using this option  we can get the packing input dialouge box (refer figure 24). Enter the required details in the dialouge box to add the Packing detail.

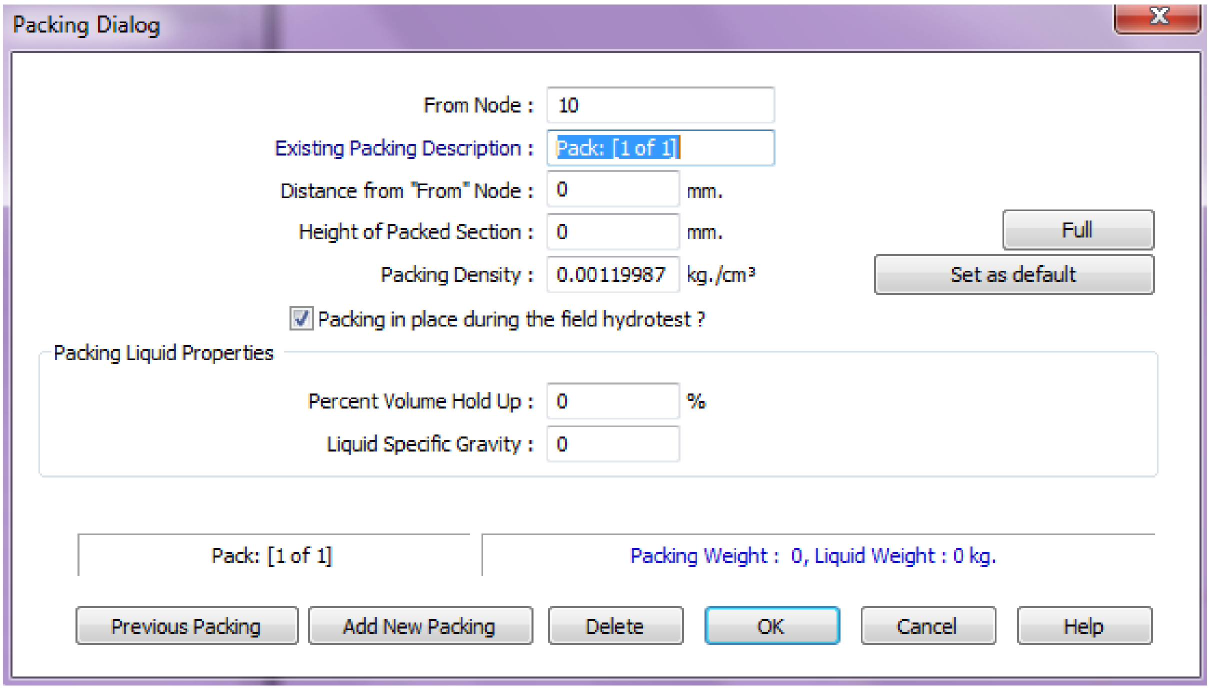


Figure 25 packing input dialouge box

The main inputs for packing in a vessel are Distance from ‘from node’, height of the packed section or Full option (eg- filling ceramic balls in a head or shell entirely we can use full option instead of height of packed section), packing density.

Previous

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

Next Packing

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

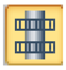
Add Packing

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

Delete

Deletes all data for the current packing.

Platform:

Using this option  we can get the platform input dialouge box (refer figure 26). Enter the required details in the dialouge box to add the platform.

Options in the platform dialouge box,

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.

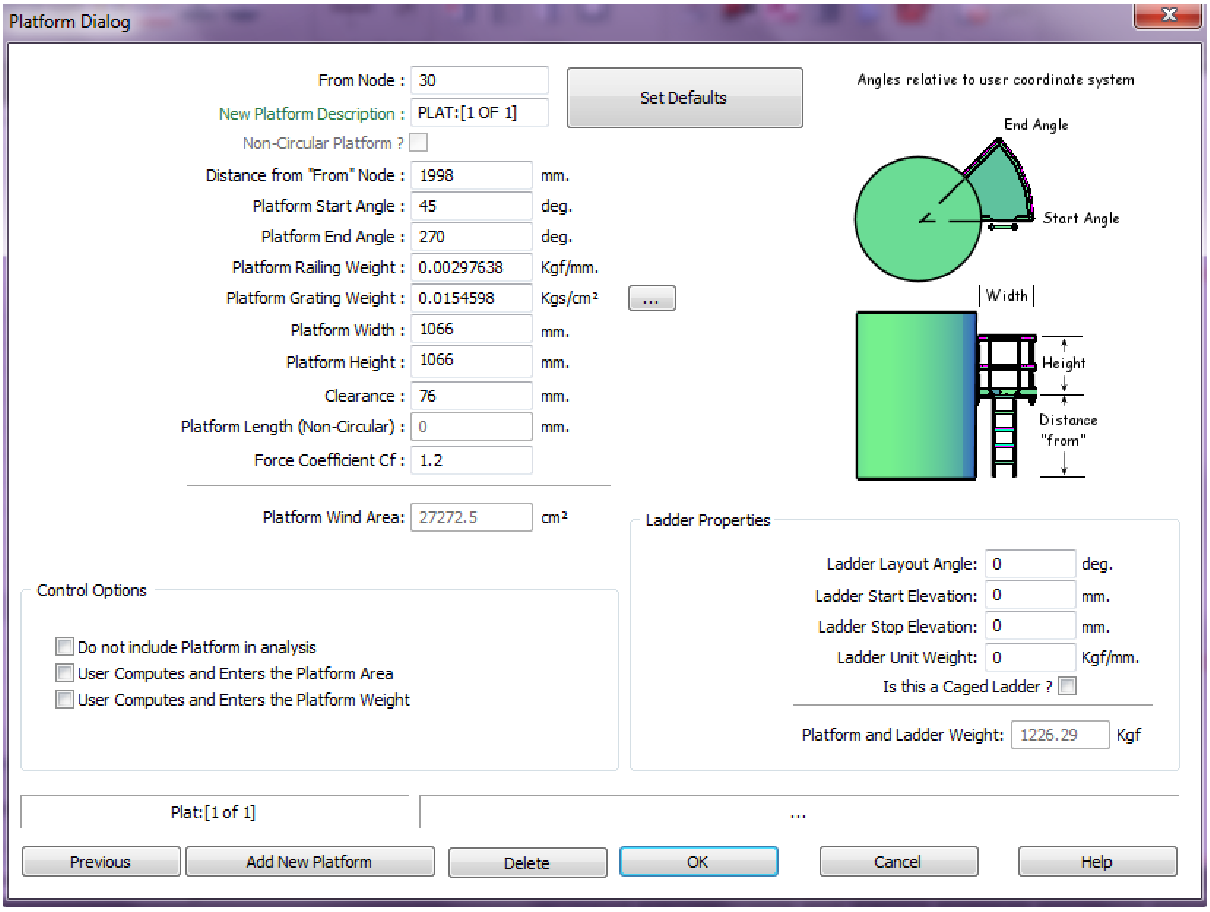


Figure 26 Platform dialouge box

Weight /piping input:

Weight and piping inputs are used to add the external weight/overhead piping weight to the vessel for analysis.

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.

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.

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.

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.

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.

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

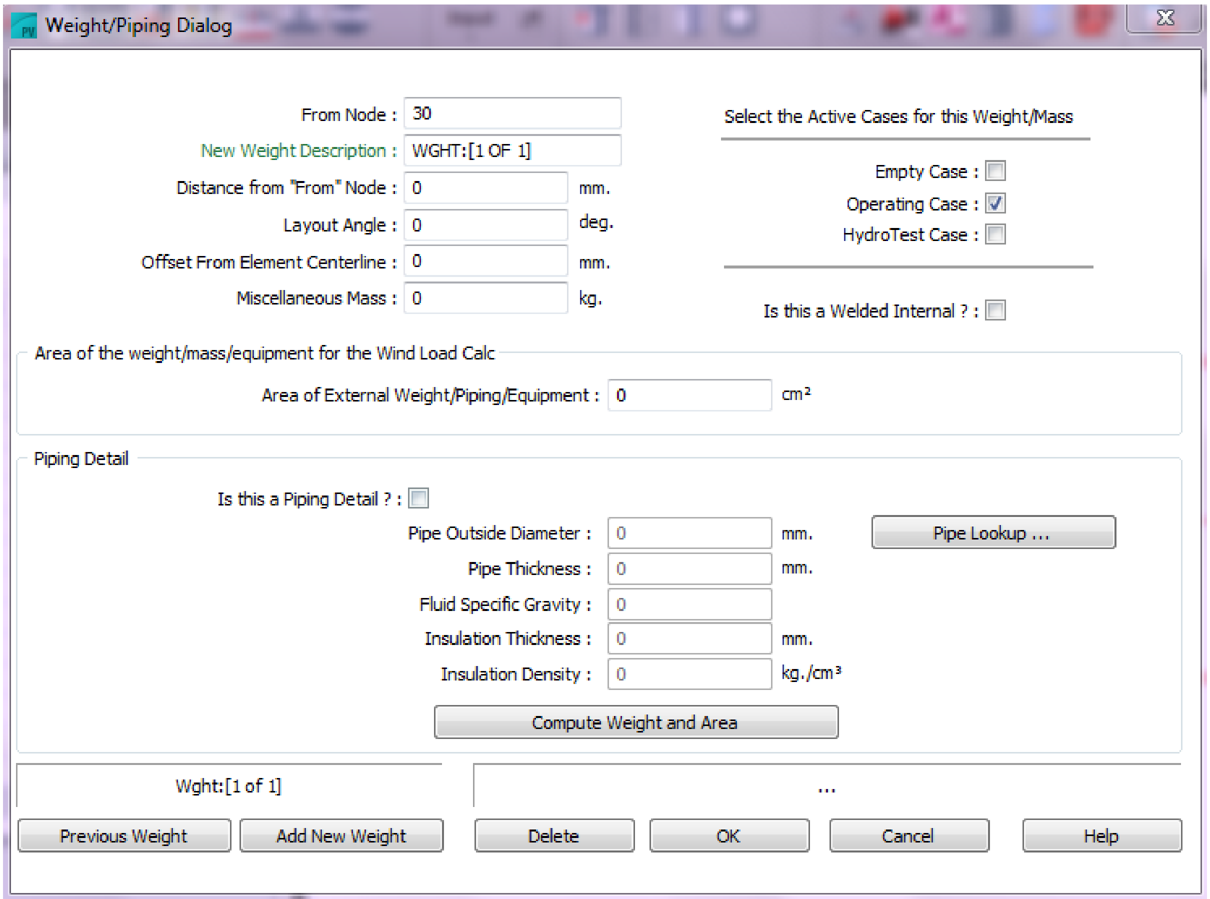


Figure 27 Weight/piping dialouge box

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.

Applied Forces – X

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.

<u>Note:</u>

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

Applied Forces - Y

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.

<u>Note:</u>

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

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.

Applied Moments - X

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.

Applied Moments - Y

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.

Applied Moments - Z

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 L*o/ ads

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.

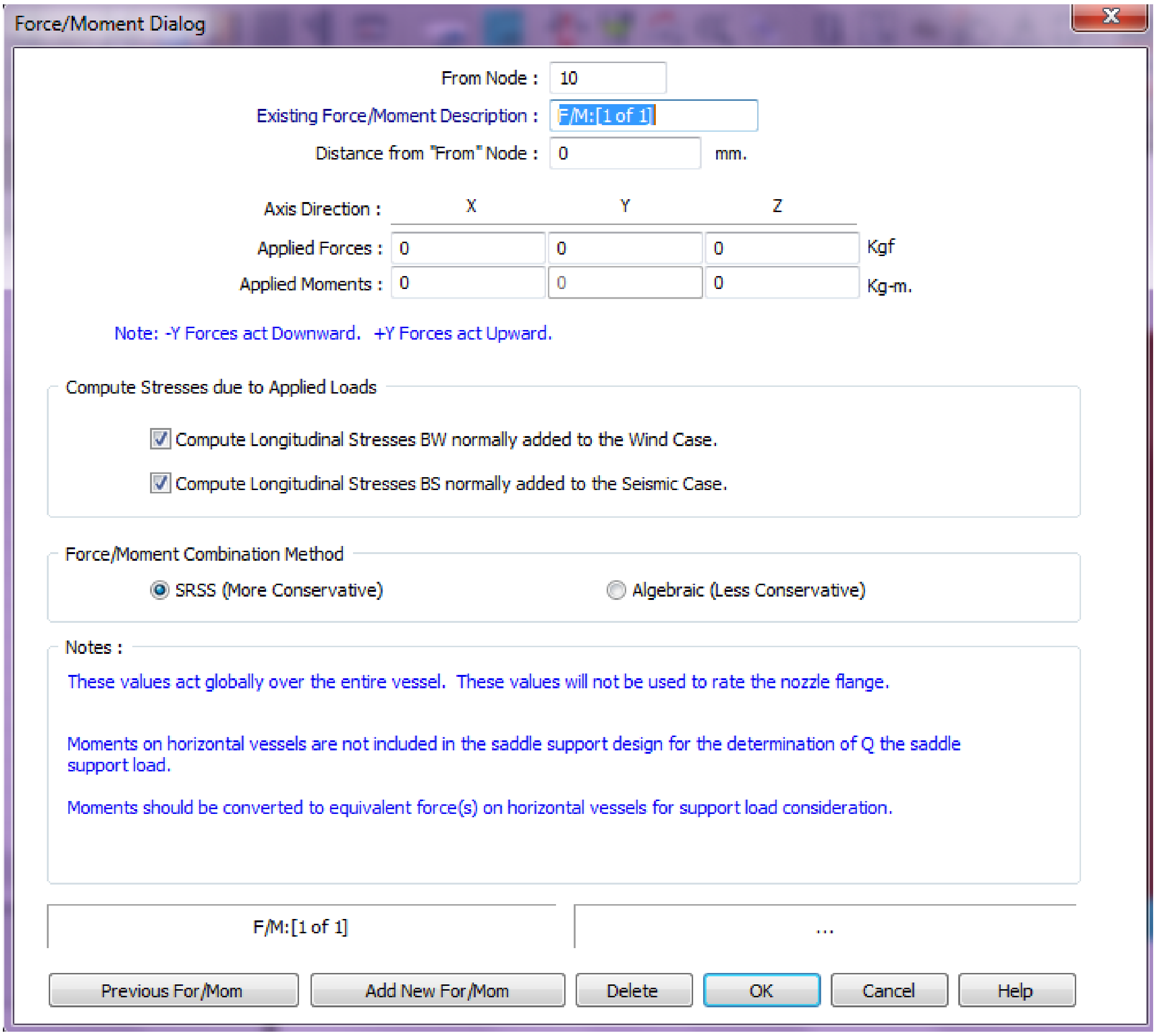


Figure 28 Force/Moment dialouge box

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.

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 Density.

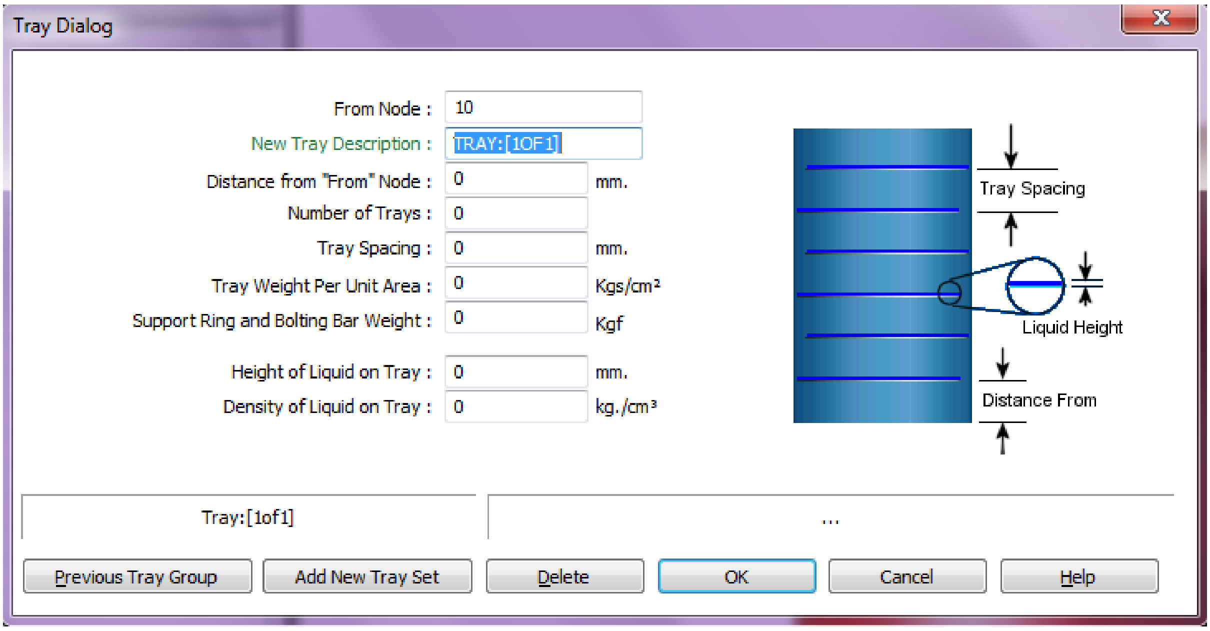
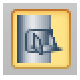


Figure 29 Tray dialouge box

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.

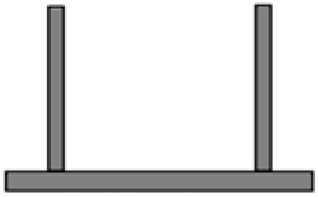
Delete

Deletes all data for the lug.

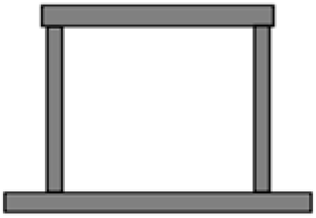
Lug Type

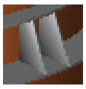
We have the following types of geometry for the support lug,

Simple geometry with gussets

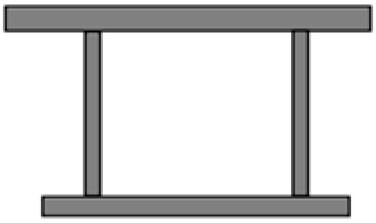


Gusseted geometry with top plate





Gusseted geometry with continuous top encirclement ring



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.

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.

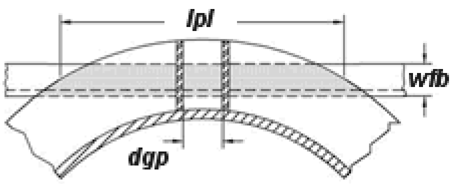
Radial Width of Bottom Ring (wpl)

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):

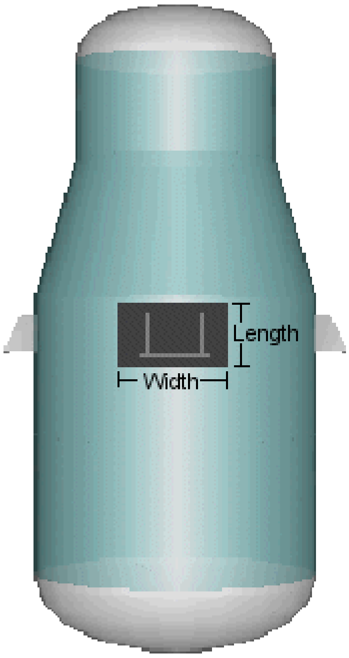
Enter the mean gusset width, defined as the gusset width at the top plus the gusset width at the bottom divided by two. The software uses this value to calculate the actual stresses in the gusset plates.

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).

Perform WRC 107 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.



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:

Thickness - Enter the thickness of the pad. Account for any allowances for external corrosion in the pad thickness.

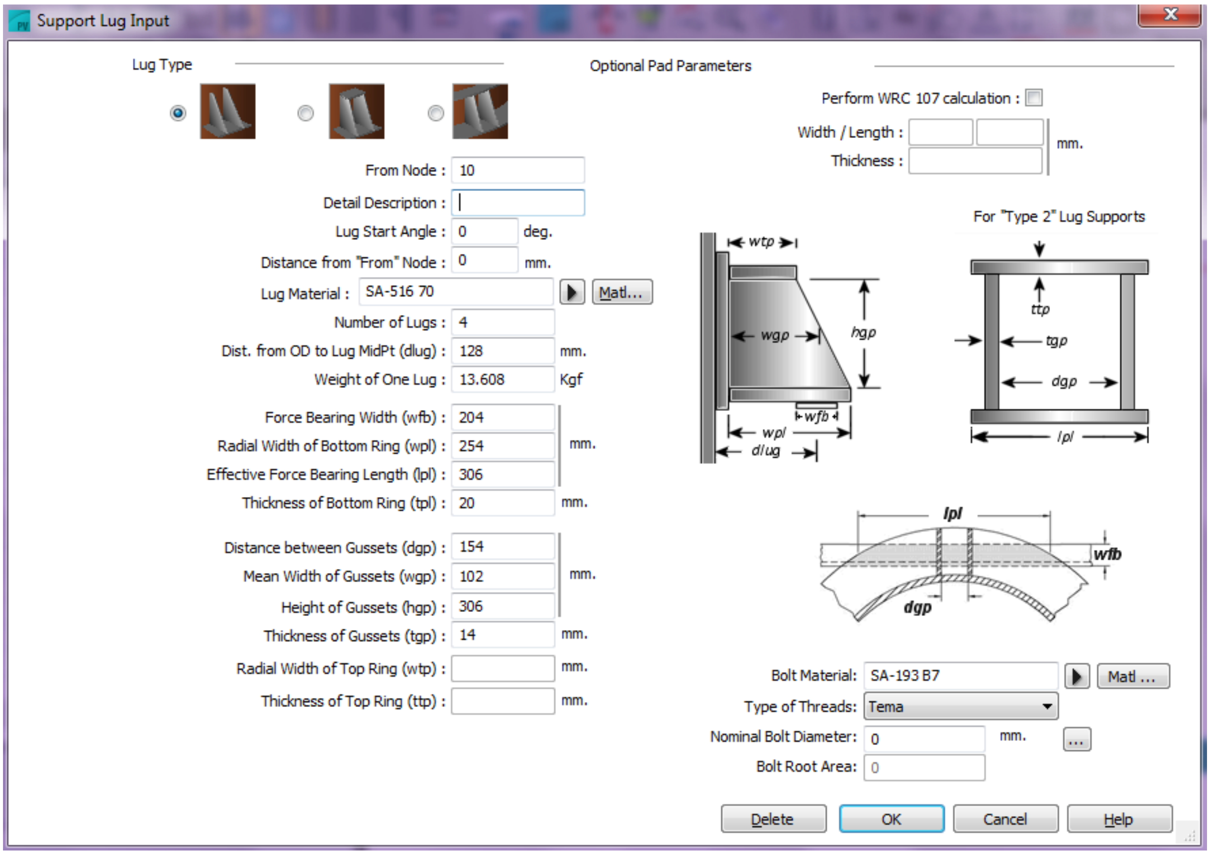


Figure 30 Lug support dialouge box

Legs:

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

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

Delete

Deletes all data for the legs.

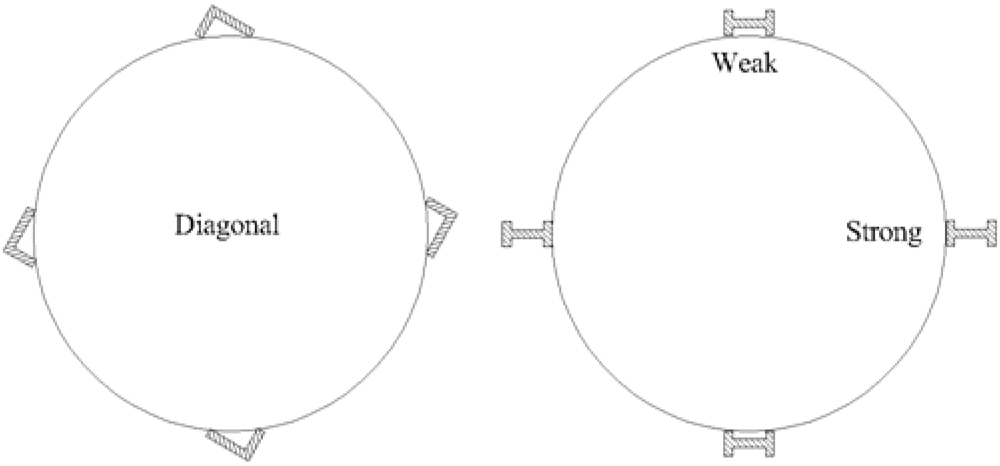
Leg Centerline Diameter

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.

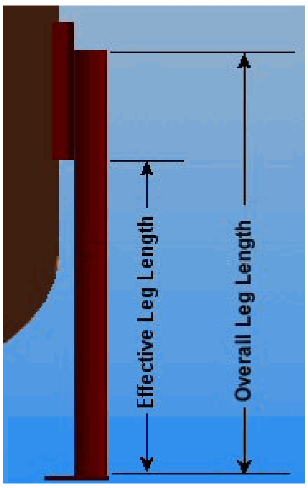


Overall Length of Legs

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.



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.

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.

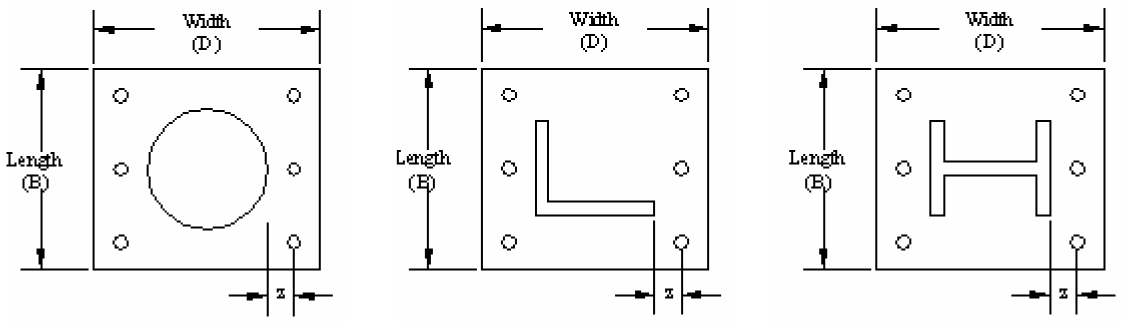
End Condition	Theoretical K	Recommended K
Fixed - Fixed	0.5	**0.65**
Fixed - Pinned	0.7	**0.80**
Fixed - Trans	1.0	**1.20**
Pinned - Pinned	1.0	**1.00**
Fixed - Rotates	2.0	**2.10**
Pinned - Rotates	2.0	**2.00**

Occasional Load Factor

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

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.



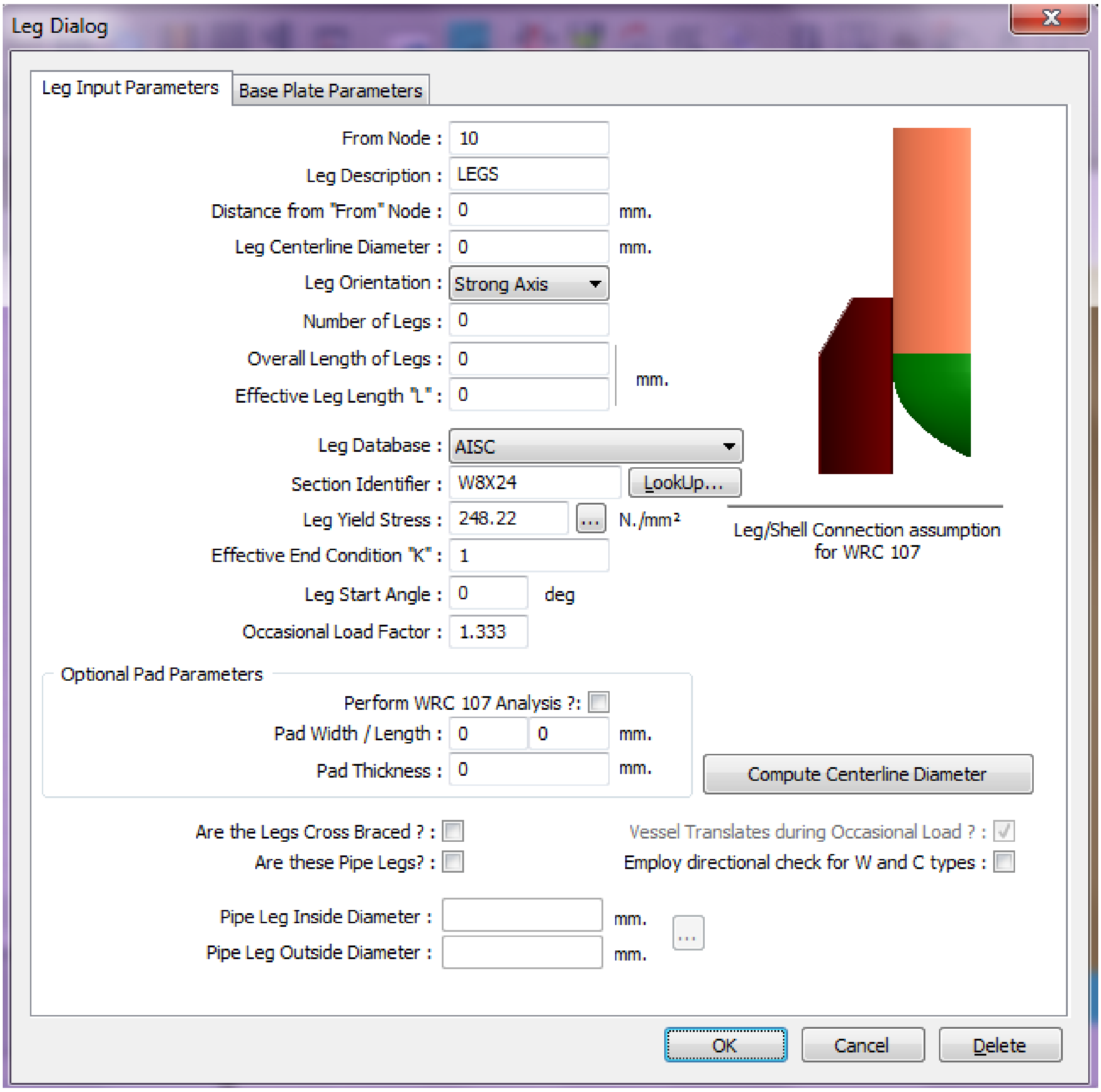
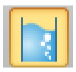


Figure 31 Leg dialouge box

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 Density. To use this command effectively, all elements in the vessel model should be created first

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.

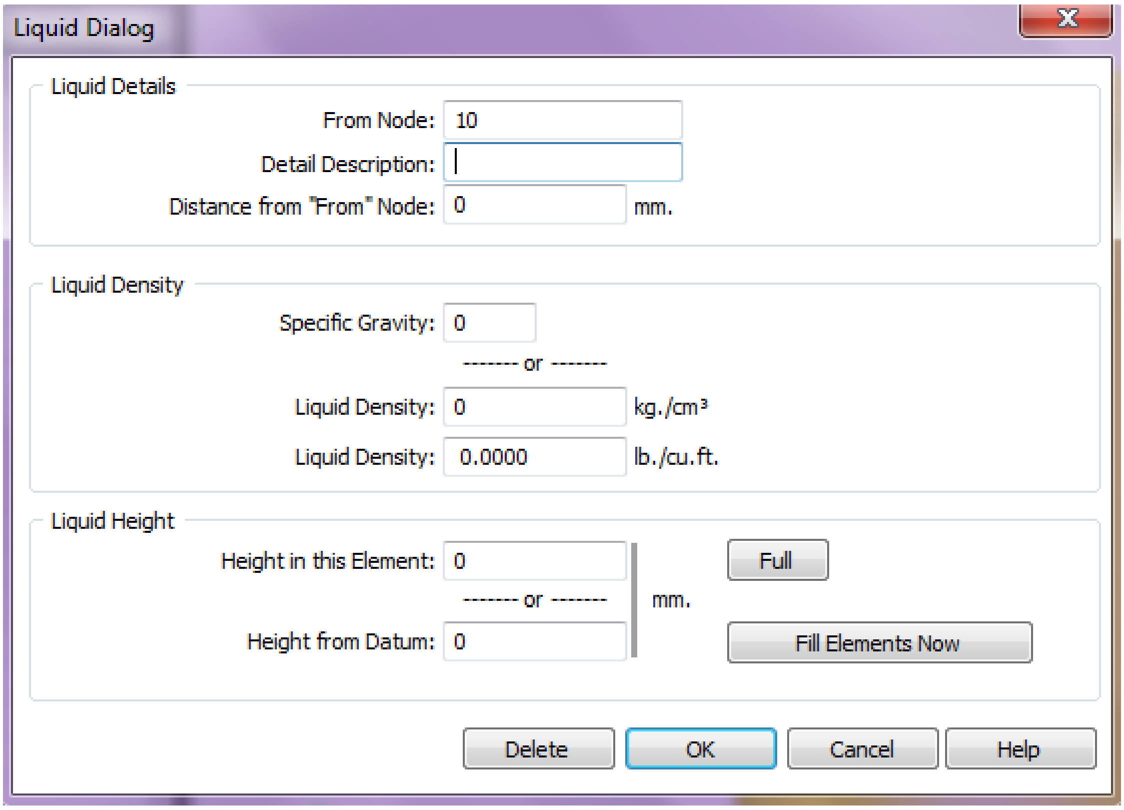


Figure 32 Liquid dialouge box

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.

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.

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

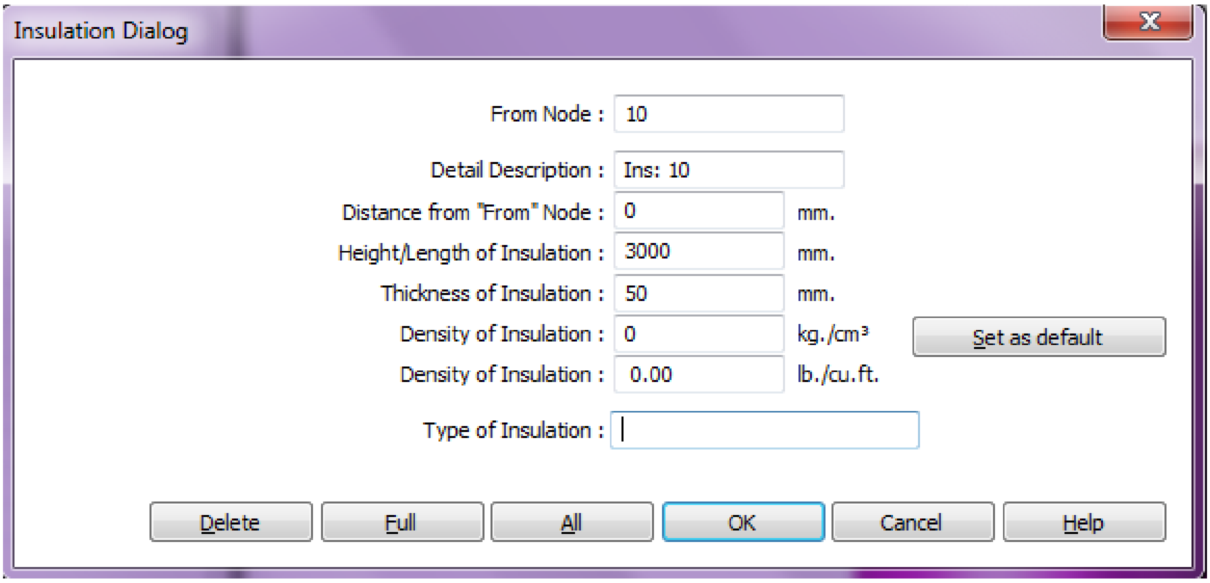


Figure 33 Insulation Dialouge box

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.

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.

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

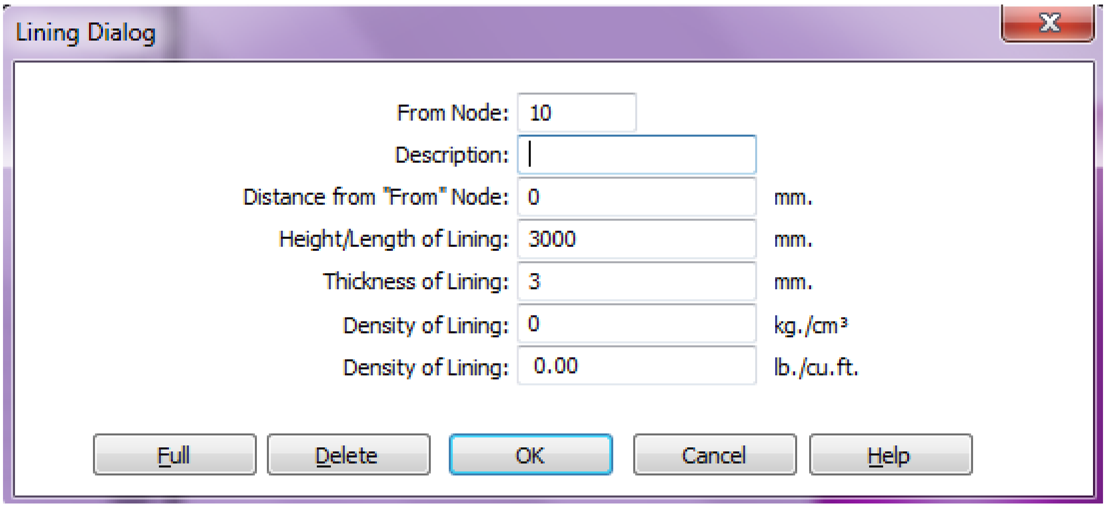


Figure 34 Lining dialouge box

Saddle:

Saddle is one type of support used for horizontal vessels.

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

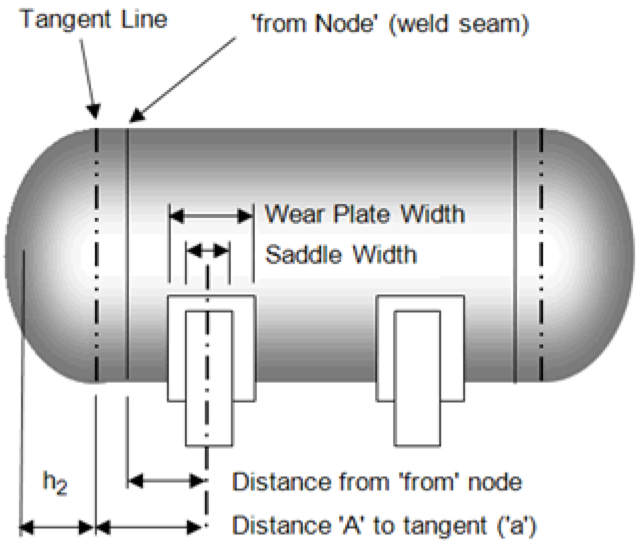
Click to open the **Saddle Properties dialog box**, and select a standard saddle (from *SaddleData.txt*, found in the PV Elite System folder).

Same as First

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

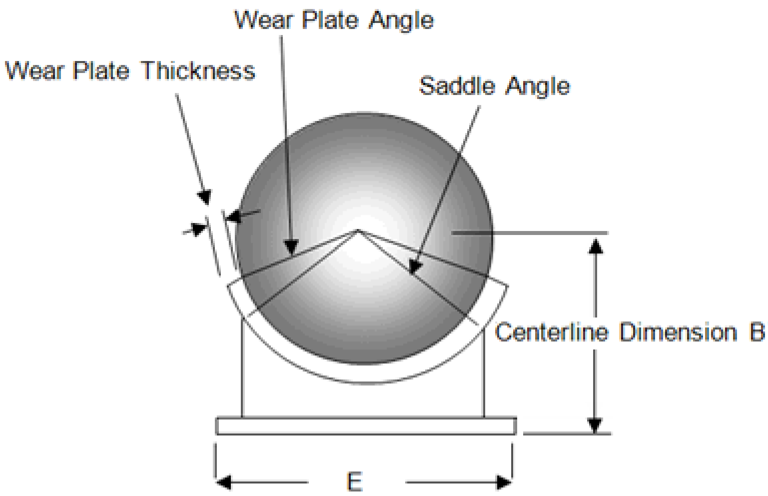
Saddle Width

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.

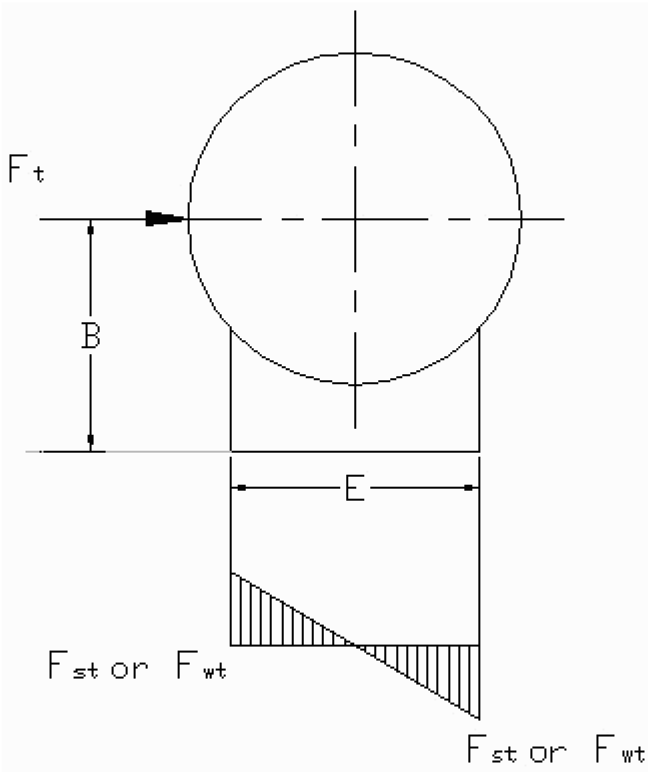
Friction Coefficient Mu

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:

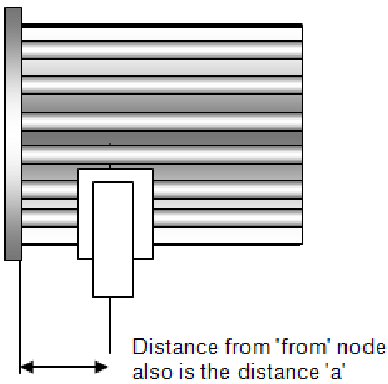


The saddle reaction load Fst (or Fwt for wind) due to the transverse load Ft is:

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.

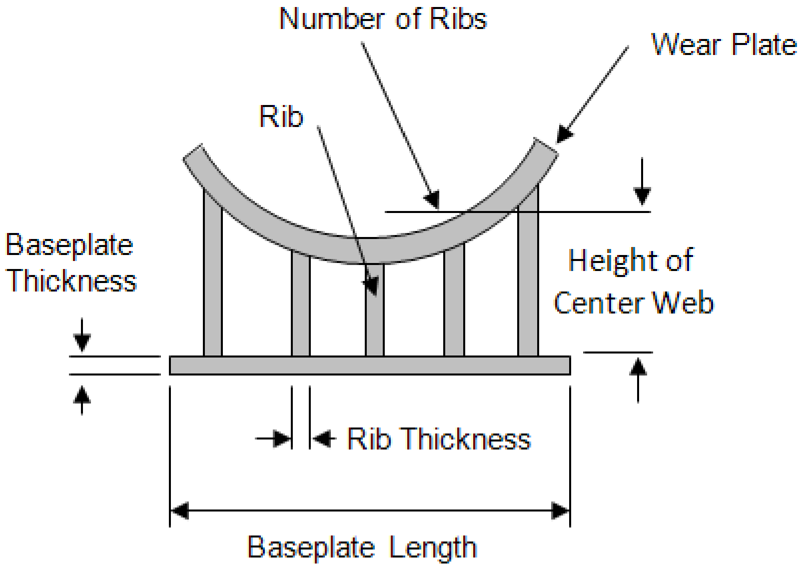
When you create a shell by welding the ends together, there is a longitudinal weld. If that weld is between saddles, there are bending moment stress.

Perform Saddle Check?

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

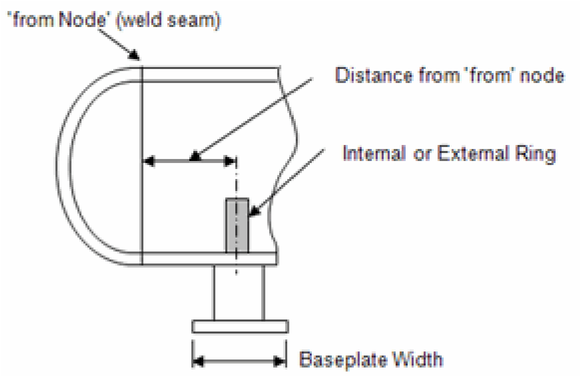
Baseplate Length

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



Baseplate Width

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

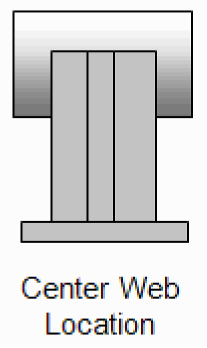


Height of Center Web

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**.



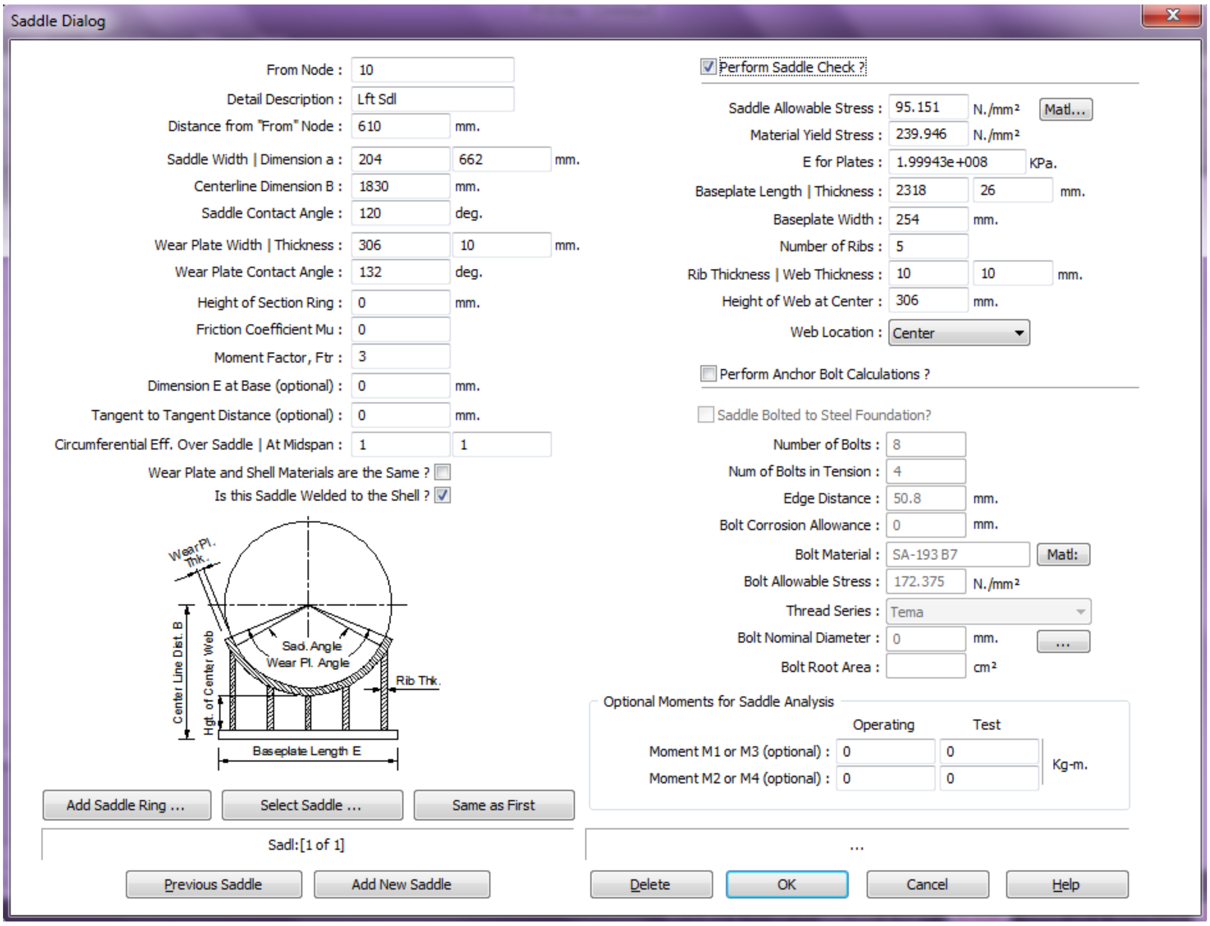


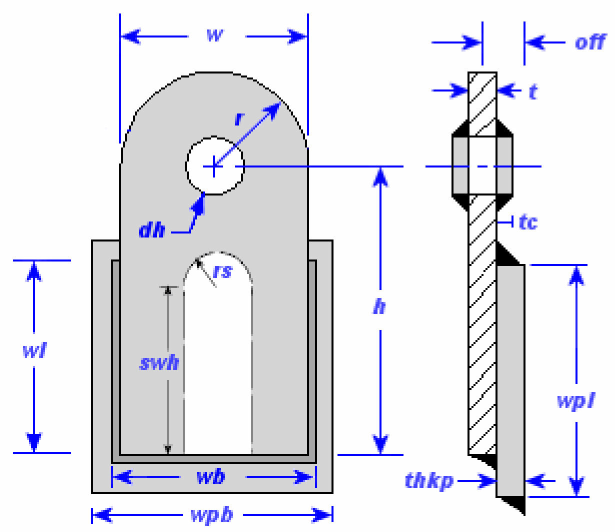
Figure 35 Saddle dialouge box

Lifting Lug Data:

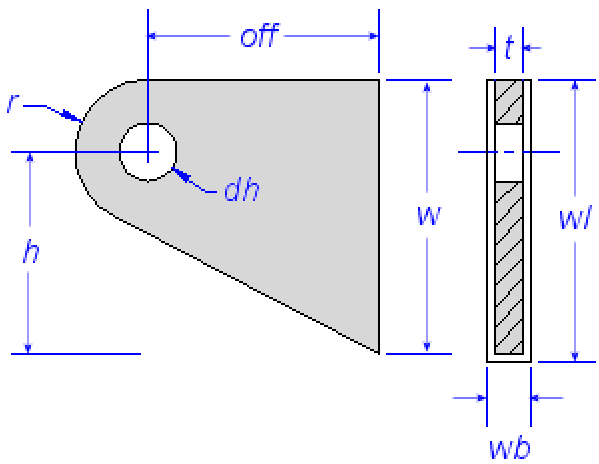
Home tab: **Details** > **Lifting Lug Data**

Adds lugs to the selected element. You can create:

1. *Flat-type lugs on vertical vessels, generally place near the top head.*



2. *Perpendicular, ear-type lugs for horizontal vessels.*



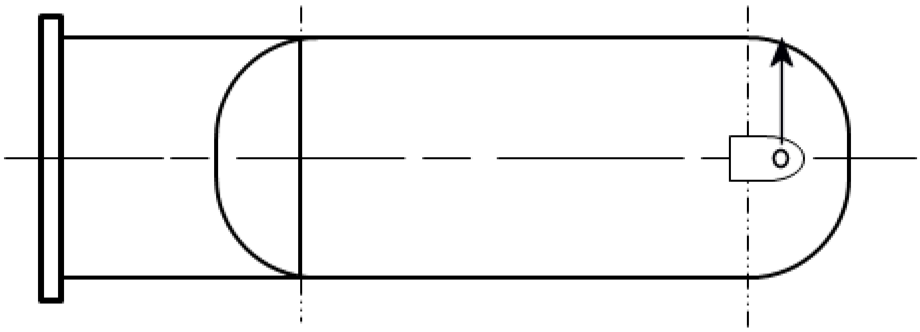
The software calculates the reactions on each lug, the stresses in each lug, and the stress in the welds.

Delete

Deletes all data for the lug.

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.



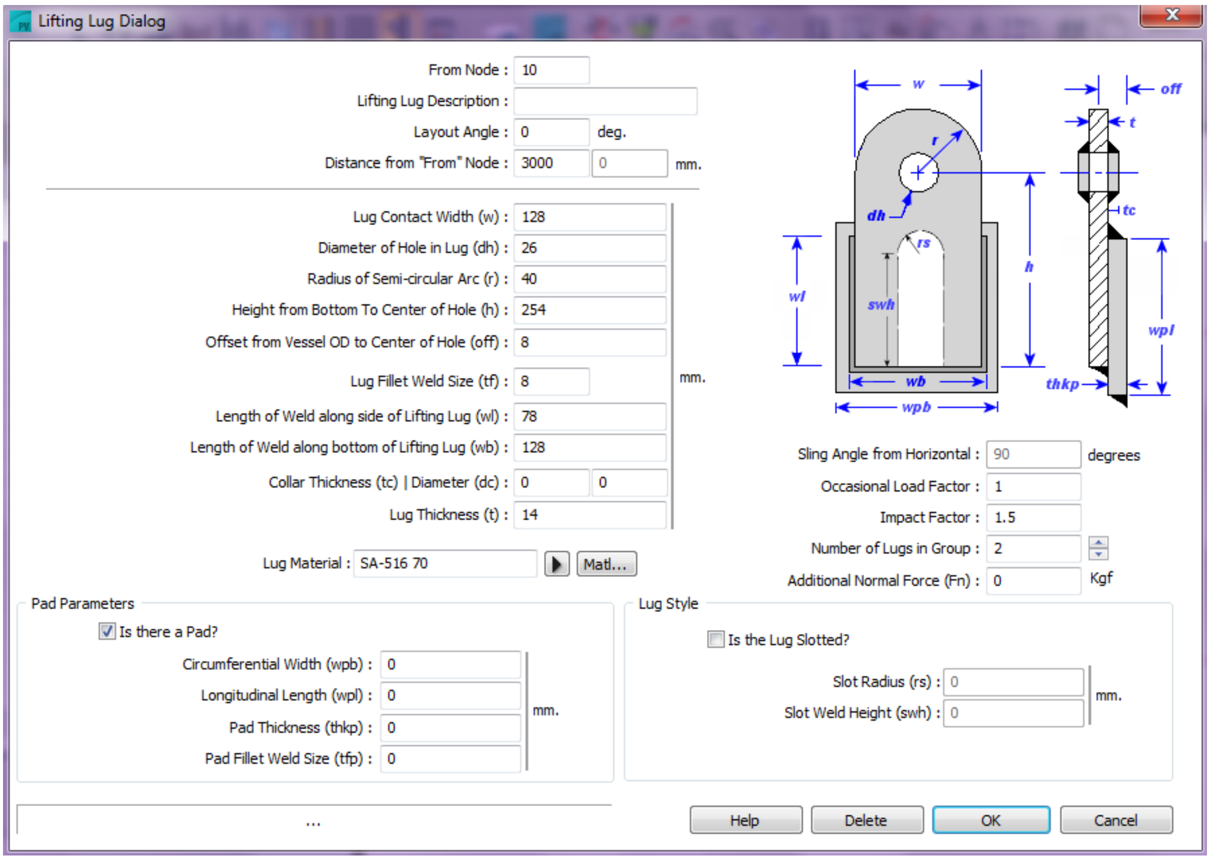


Figure 36 Lifting lug dialouge box

Generic 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.

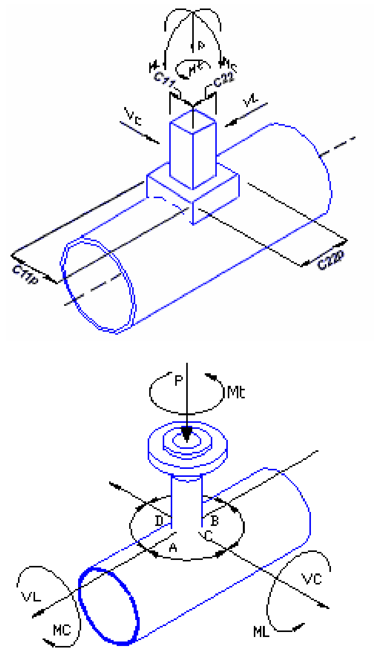
Sustained, expansion, and occasional loads on the clip must be determined from a separate analysis and entered. Clips supporting piping generally have loads in all categories, while platform clips generally have only sustained and occasional loads.

Local Attachment Loads at the Shell Surface

You can enter values in the following load sets:

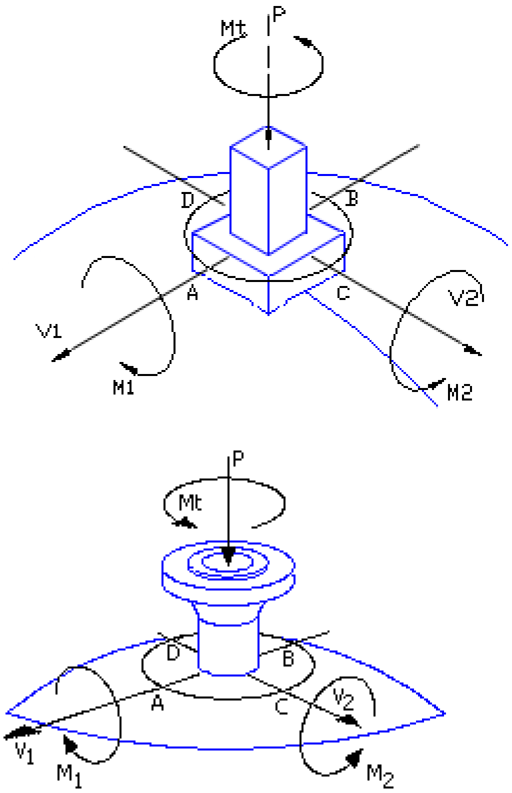
- Sustained - (SUS) Primary loads, typically weight + pressure + forces.
- Expansion - (EXP) Secondary, self-limiting thermal expansion loads.
- Occasional - (OCC) Irregularly occurring, short-term loads, such as wind loads, seismic loads, and water hammer.

*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

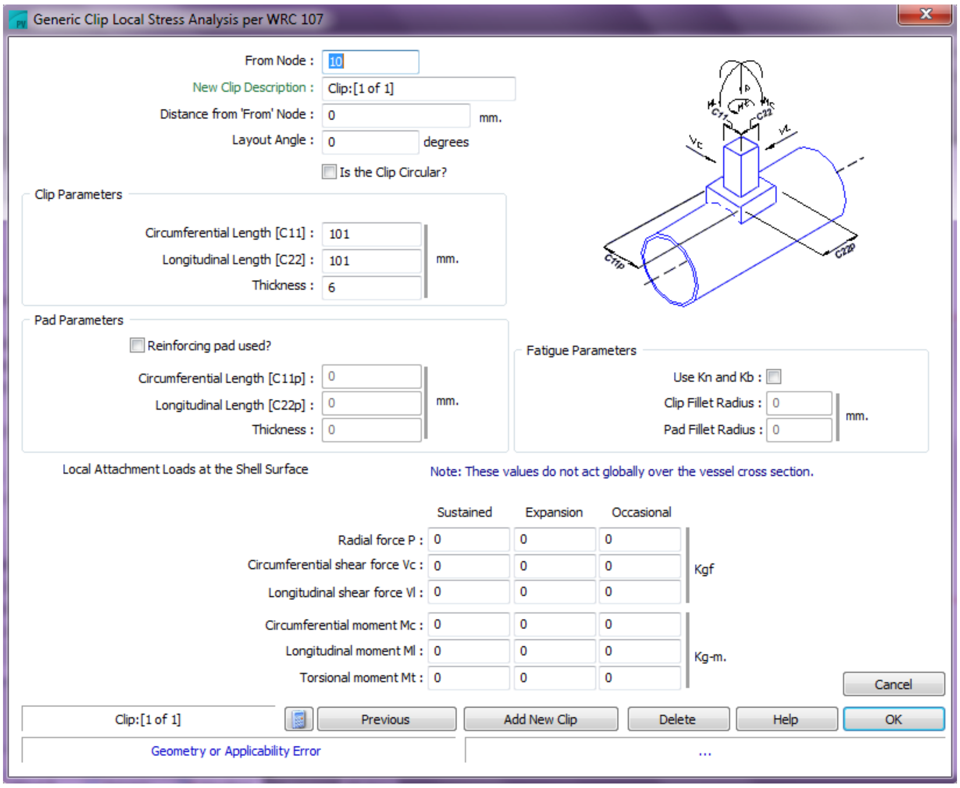


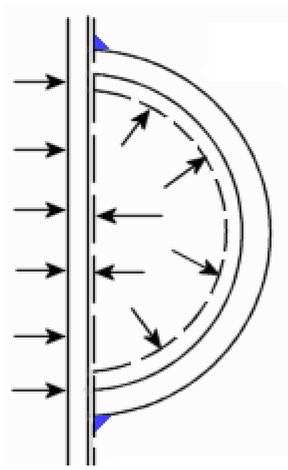
Figure 37 Clip dialouge box

Halfpipe Jacket:

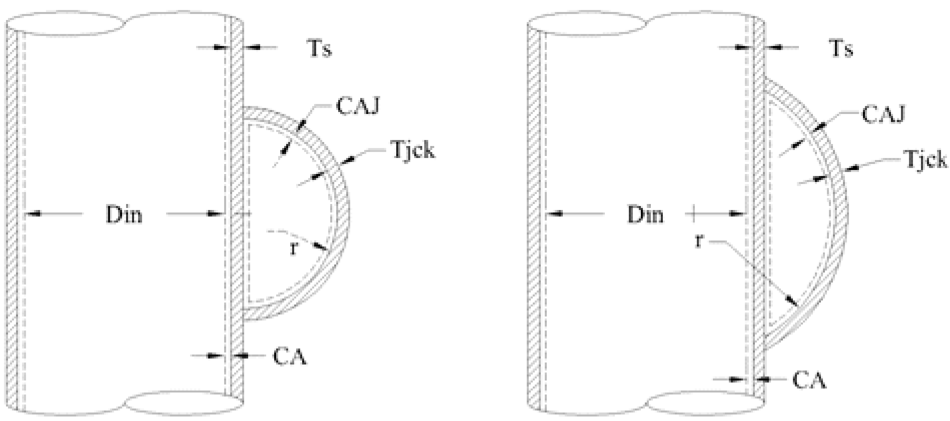
Home tab: **Details** > **Halfpipe Jacket** 

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

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 the ASME Boiler and Pressure Vessel Code, Section VIII, Division 1, Paragraph EE-1, Appendix EE. The analysis is only valid for the cylindrical geometries shown in Figure EE-4.

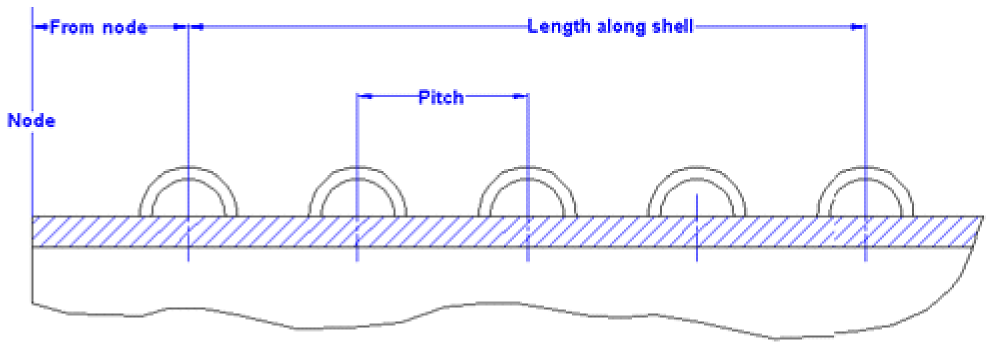


Additionally, only nominal pipe sizes from 2 to 4 can be used. Although there are no charts for sizes 2.5 and 3.5, the software accepts these sizes and performs iterations between the given charts. If the half-pipe is a nonstandard pipe size, or has a formed radius, the actual radius is used in the calculations.

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.

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.



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**.

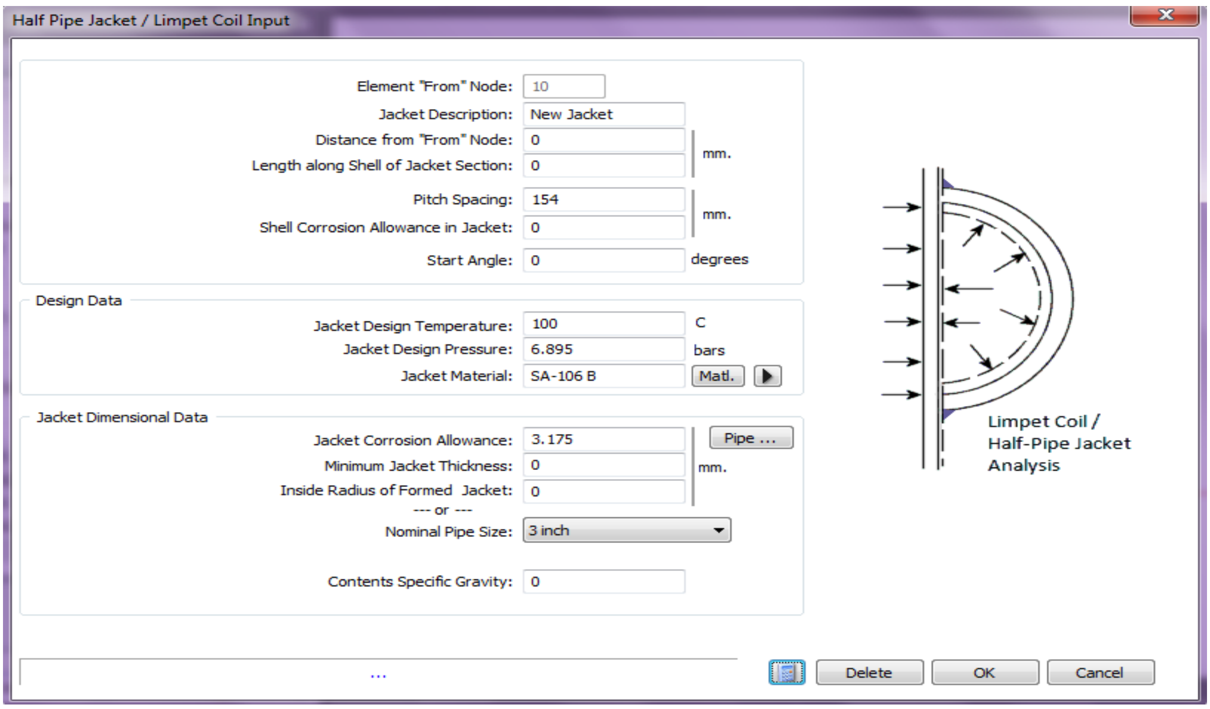


Figure 38 Half pie jacket/Limpetcoil dialouge box

Vessel Jacket, Vapour/Distribution Belt:

Home tab**: Details > Jacket or Vapor/Distribution Belt**

Adds ASME Appendix 9. Type 1 cylindrical jackets to the shell on the selected cylinder element, according to ASME Code, Section VIII, Division 1, Appendix 9.

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, 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.

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

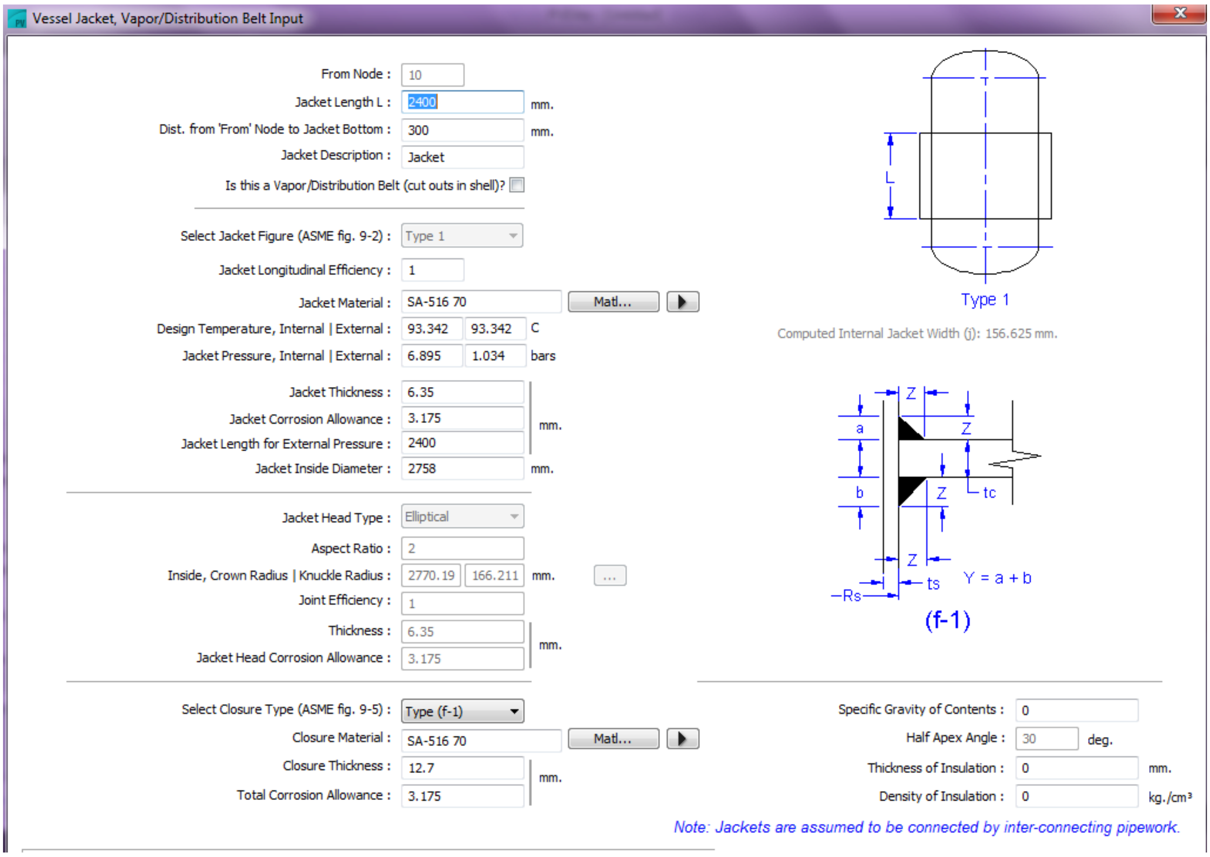


Figure 39 Vessel Jacket, Vapour/Distribution Belt dialouge box

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.

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 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.
  - ***Gouge* –** A local, elongated, mechanical removal or relocation of material from the component surface resulting is 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 in a region of widely scattered pitting,
  - localized regions of pitting, and
  - pitting confined within a region of an LTA.

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 these 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.

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 these 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.
- 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.
- 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
- 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.

- 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 pares 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 6.15)
  - 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.

Remaining Strength Factor (RSFa)

Enter the allowable remaining strength factor (RSFa). The recommended RSFa for all major design codes per the 2007 edition of API 579 is 0.90. See Table 2.3.

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 character 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 6.3 to 6.10) 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 6.11). 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 6.11). 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 6.12). If the pit has an irregular shape, a diameter and depth that encompasses the entire shape should be used in the assessment.

MAWP Approach

Select this option to calculate the remaining life of Type A, B, and C components using a systematic method. You must select this option for Type B and C components. This approach also ensures that the design pressure is not exceeded during normal operation if the future corrosion rate is accurately established.

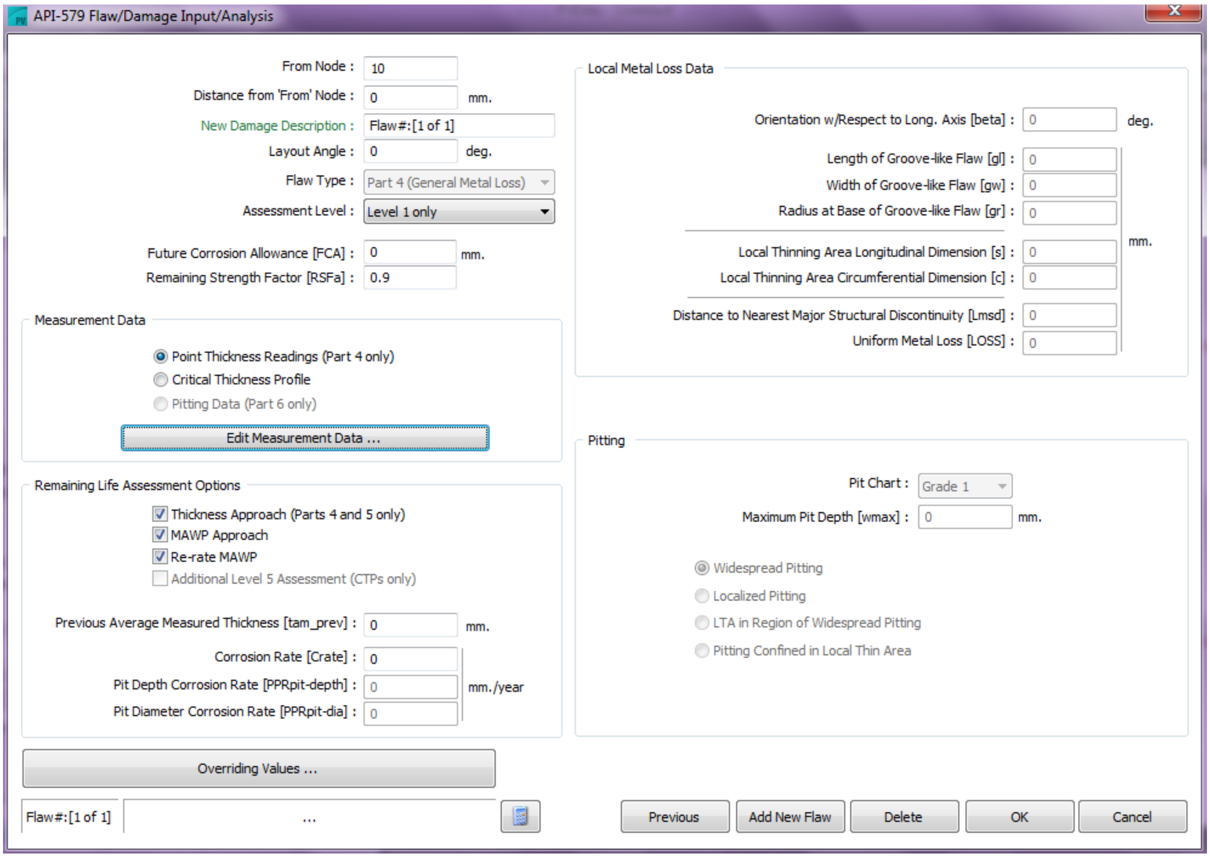


Figure 40 API-579 Flaw/Damage Input/Analysis dialouge box

4.3 Utility panel

This tool bar(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. 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.

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:

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.

4.4 Auxilliary 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.
	List Dialog - Open the Detail Listing dialog box. For more information, see List Dialog.
	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.
	Rigging Results - Display rigging results.
	Create Database - Create database of input files. For more information, see Create Database.
	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.
	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.

5. ANALYSIS

Once the modeling and designing is over, we need to analyse the model (equipment) to check status of the equipment.

Using the Analyze panel we can analyse the model.

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.
	Error Check - Error check the vessel input. For more information, see Error Check Only.
	Review - Review the analysis data output for the vessel from the last analysis. For more information, see Review Reports.
	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.

Analyze:

Home tab: **Analyze > Analyze** 

Analyzes the current model and creates the output files. Click reports in the **Report List** to see results of the 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.

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.

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.

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.

6. CONCLUSION

This book gives some idea about the PV-Elite pressure vessel design software.