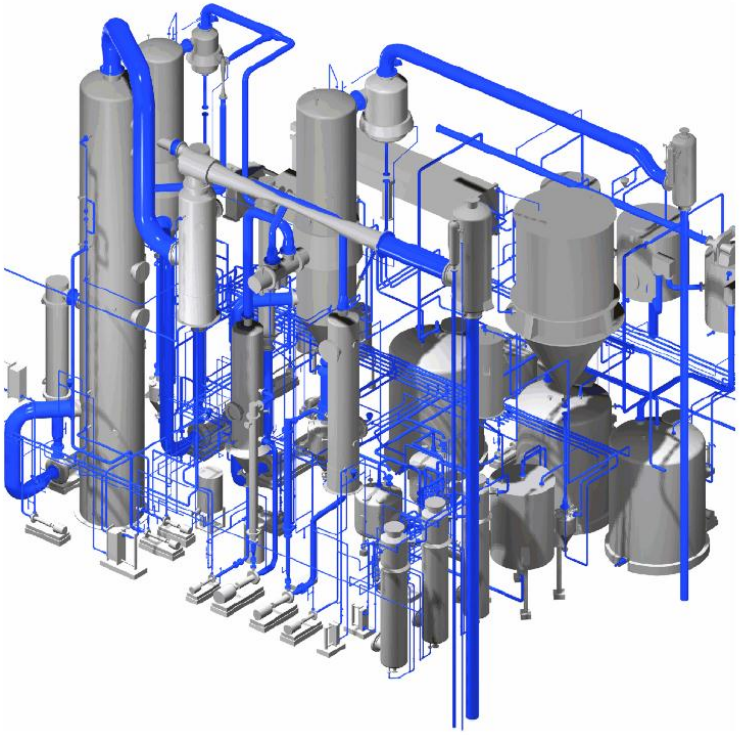
PV ELITE 2007



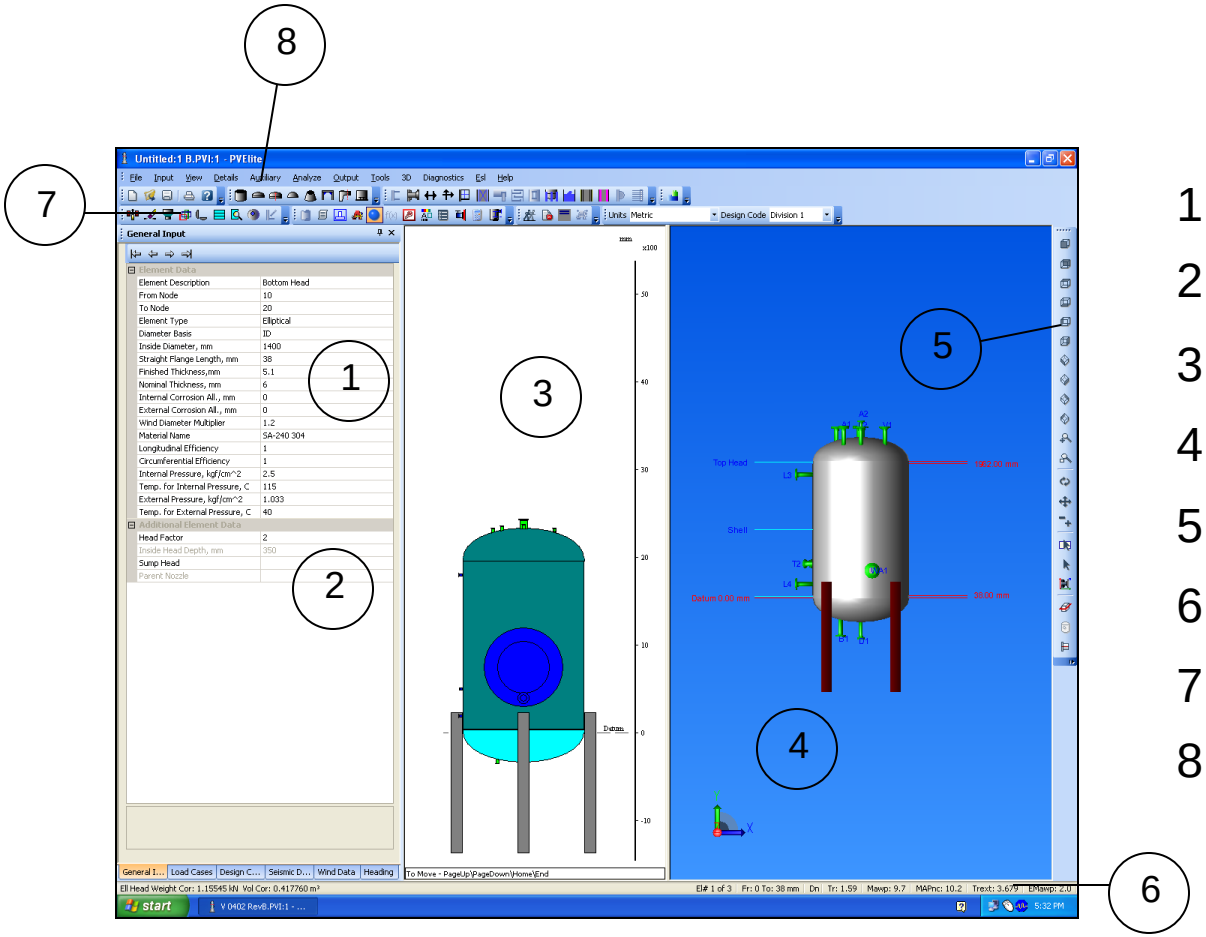
TRAINING PROGRAM

WHAT IS PV ELITE 2007?

1. International Program for calculation Detail of Pressure.

2. Design Code available in PV-ELITE ;
   - ASME sect. VIII division 1.
   - ASME sect. VIII division 2 .
   - PD 5500
   - EN 13445

INTERFACE Of PV-ELITE PROGRAM.



1. Element Basic Data Area.
2. Element Additional Data.
3. 2D model view.
4. 3D model view.
5. Oraphic Control.
6. Quick result.
7. Tool bar.
8. Main menu.

TOOL BAR

Toolbars—Displays a variety of toolbars users can access.

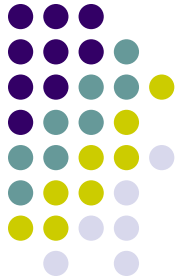
Name	Description
Standard Bar 	Allows the user to open, save and print.
Element Bar 	Allows users to create a vessel with default data by clicking an icon on the **Element** Toolbar.
Detail Bar 	Allows users to add certain details i.e., stiffeners, nozzles, forces, moments, lining, half-pipe jackets and tubesheets etc. to the current element when applicable.
Utility Bar 	Enables users to insert, delete, update, share and flip elements.
Auxiliary Bar 	Allows manipulation of the model, create drawings and databases and others.

BASIC CONTROL VIEW

3D View Icons

Option	Description
Predefined Views	Changes the current view to front, back, top, bottom, left, right view or a standard isometric view.
Zoom Extents	Resizes the model so that it fits in the current window.
Zoom Window	Allows users to use the mouse to draw a window around the portion of the model that you want to zoom in on. This is a rubber band zoom. Alternately, spin the mouse wheel to zoom in and out.
Orbit	Allows users to rotate the model using the mouse. Click the right mouse button and move the mouse to rotate the model.
Pan	Allows users to translate the model in the direction the mouse is dragged. Pressing the mouse wheel and holding it down while moving the mouse will also pan the model.
Zoom Camera	Enables users to zoom in or out. Click this button then press the left mouse button and move the mouse diagonally across the screen to zoom in or out. Alternately, spin the mouse wheel to zoom in and out.
Select by Click	Allows the selection of a detail for further manipulation. This is the cursor icon.

Basic step to create vessel.



1. Input Data.
2. Create 1st Element.
3. Select unit and design code.
4. Create all main element.
5. Create support.
6. Create accessory.
7. Run and solve the problem.
8. Convert to word and Issue!!!

1. INPUT DATA

1.1 Load Cases

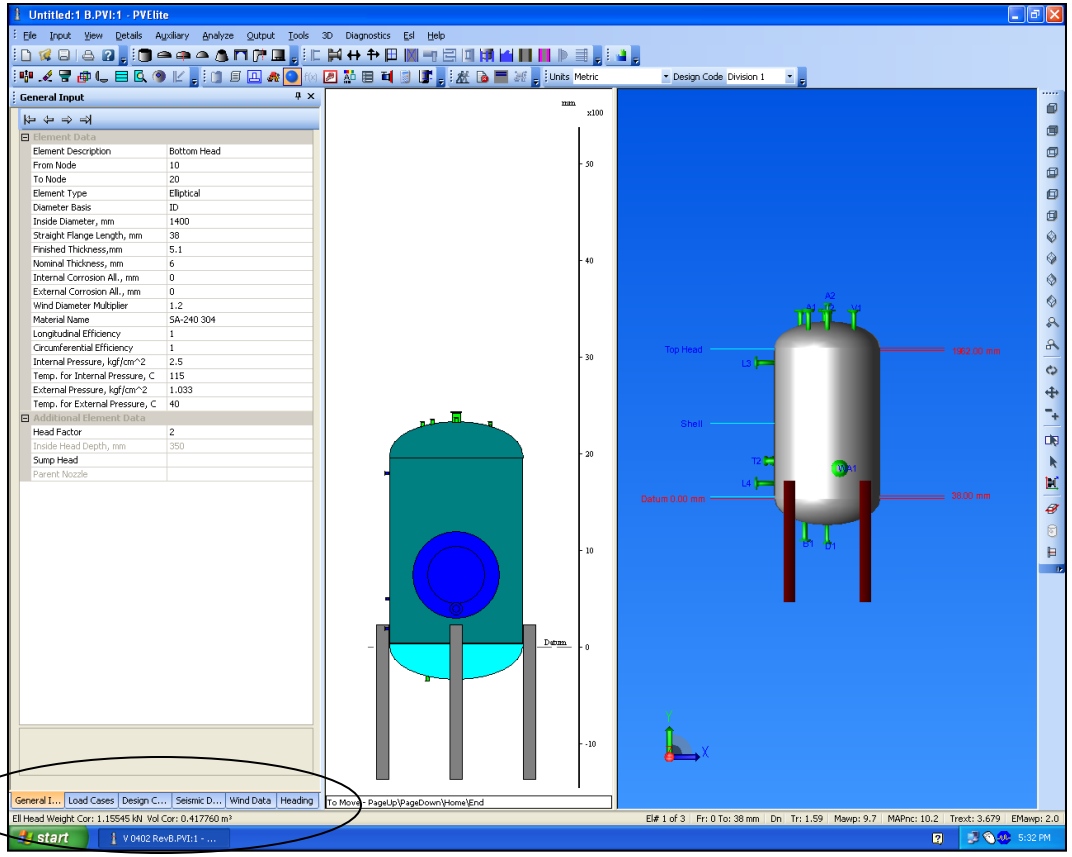
1.2 Wind Data

1.3 Seismic Data

1.4 Heading

1.5 Design Constrains

1.6 General Input



1. INPUT DATA

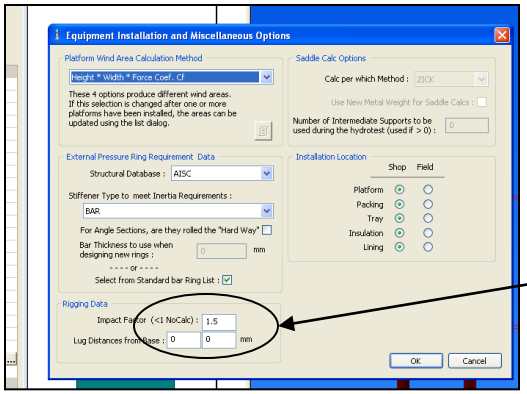
1.1 Load Case

Calculation for any case of load force effect to your vessel.

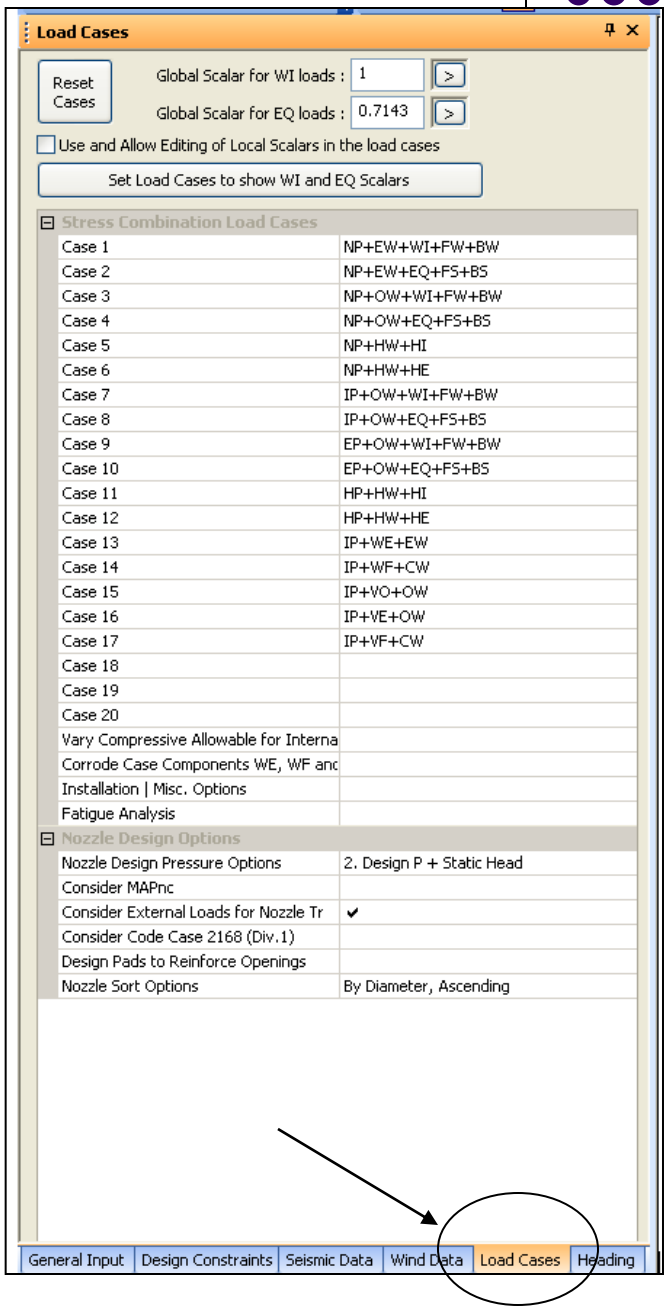
1. Study load case by press “F1”.
2. Click at “Fatigue Analysis”

   to perform fatigue calculation.
3. Click at “Installation | Misc.Option”

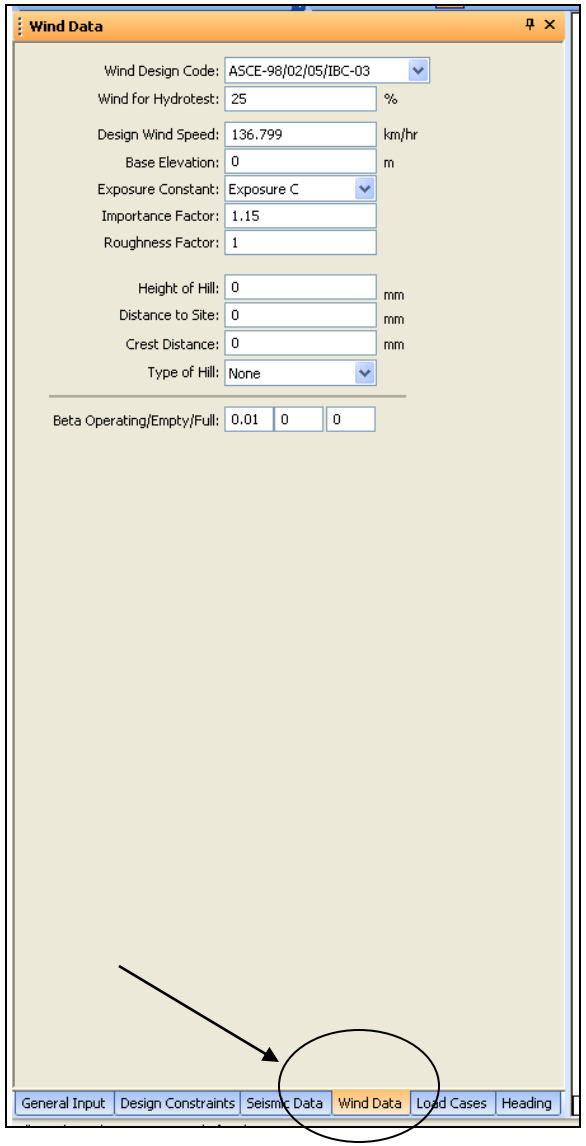
   to check rigidity when lifting.



Input this value to perform rigidity.



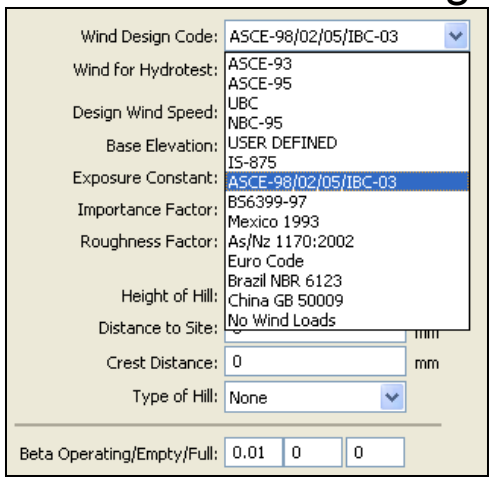
1. INPUT DATA



1.2 Wind Data

Calculation for effect and force from wind.

1. Select “Wind Design Code”

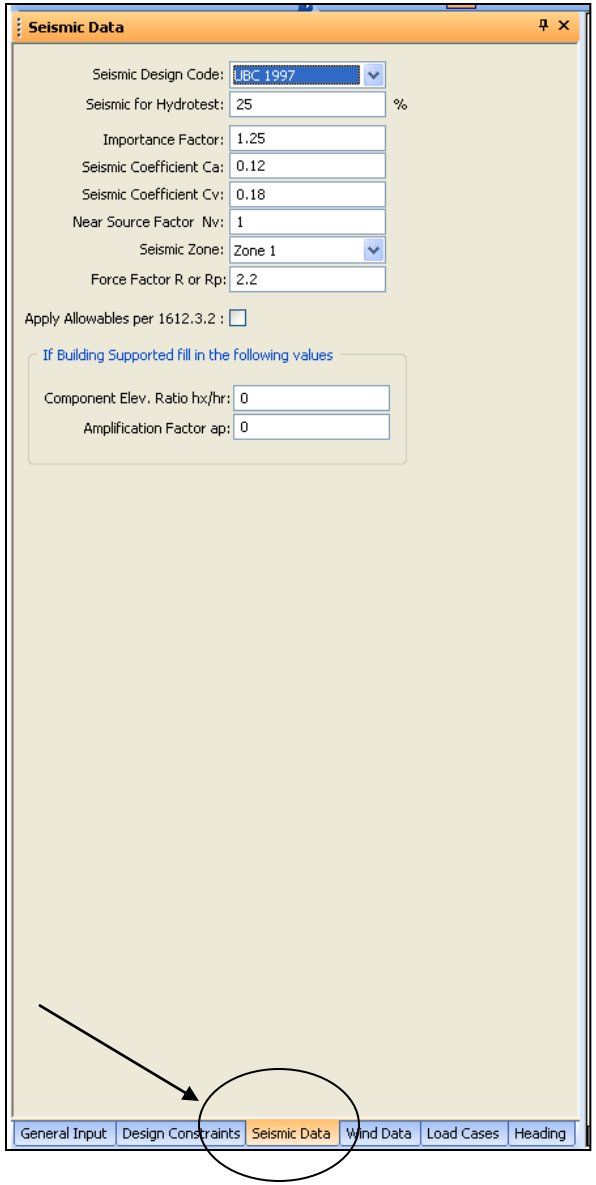


each code will have different data to input.

2. Fill remain box by press “F1” to select value.

3. This sample picture is input for plant at Rayong which you may use if have not data.

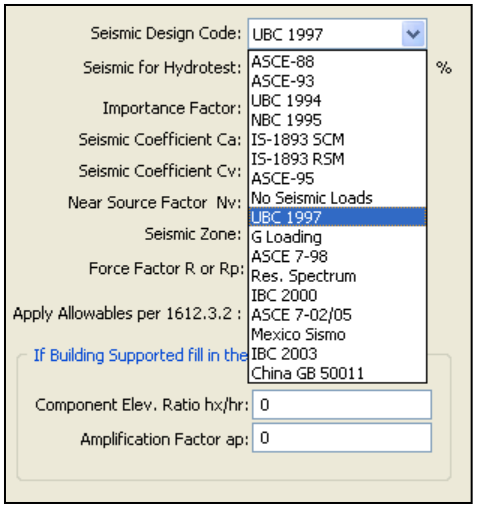
1. INPUT DATA



1.3 Seismic Data

Calculation for effect and force from Seismic.

1. Select “Seismic Design Code”



each code will have different data to input.

2. Fill remain box by press “F1” to select value.

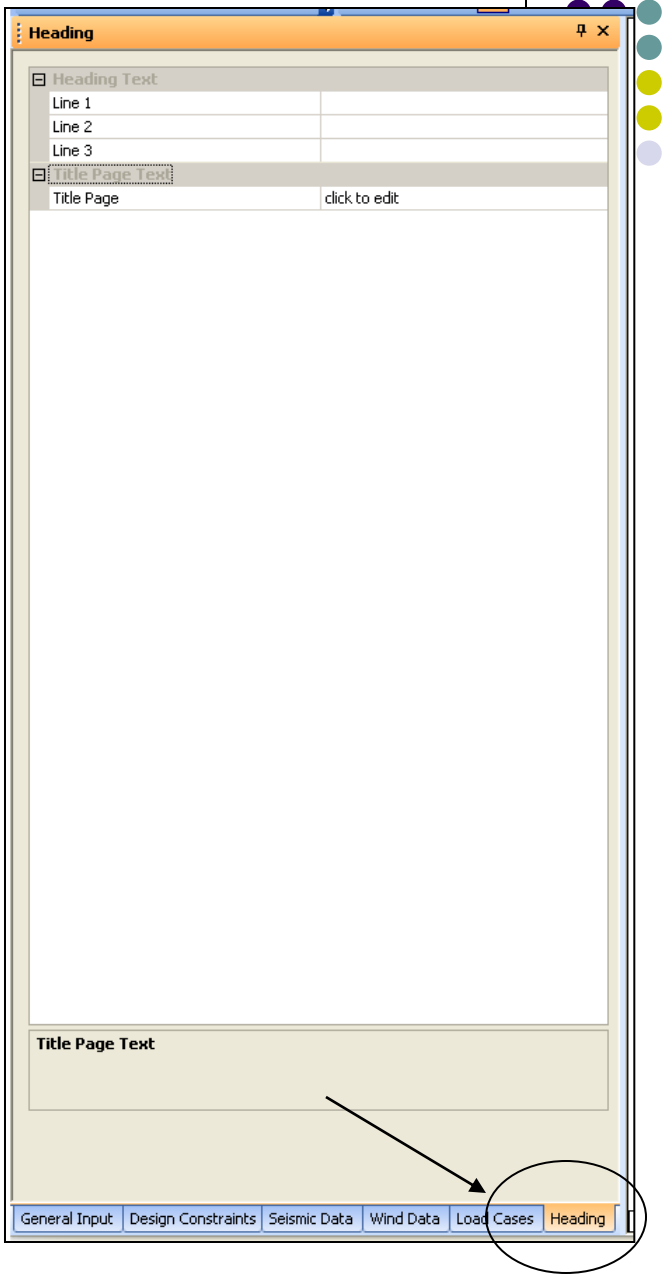
3. This sample picture is input for plant at Rayong which you may use if have not data.

1. INPUT DATA

1.4 Heading

Type wording which you want PV-Elite to show into your Title page of calculation report.

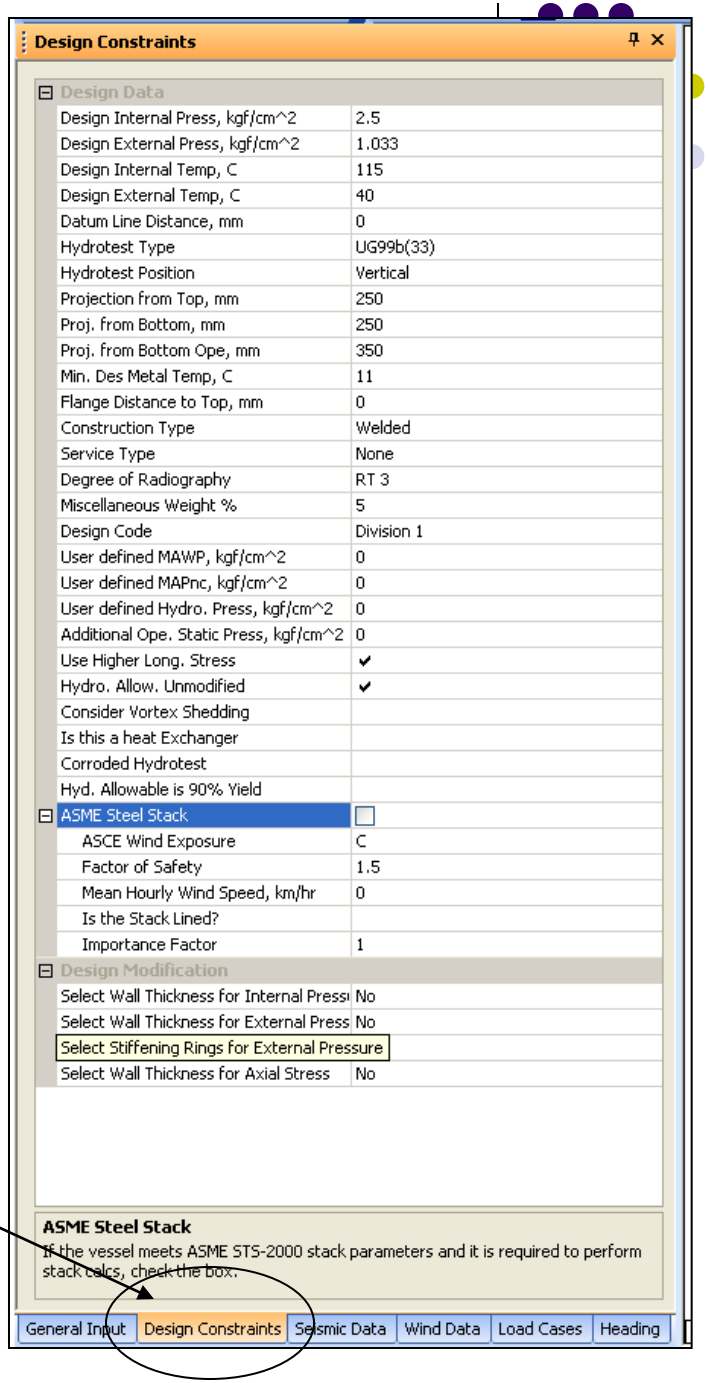
**** NOT IMPORTANT, NO NEED TO FILL ANYTHING AT THIS PAGE ***



1. INPUT DATA

1.5 Design Constrain

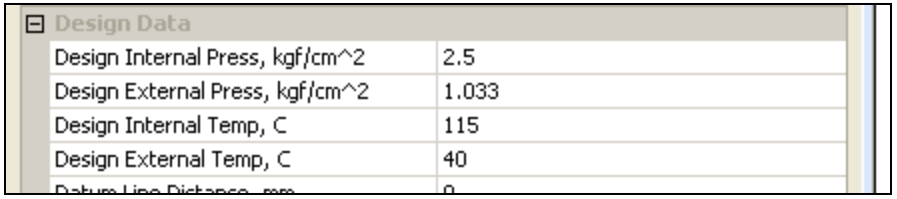
Input design data for equipment, this value filled will effect to all part of your equipment.



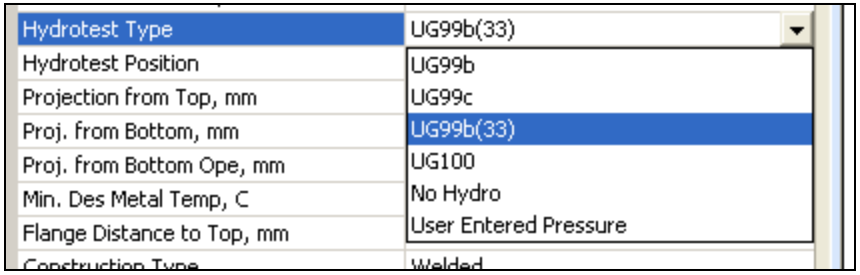
1. INPUT DATA

1.5 Design Constrain

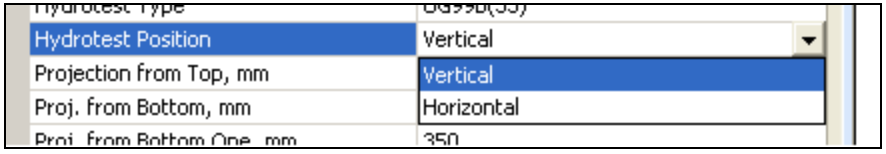
Main Input



1. Design temp and pressure : input from your information.



2. Hydro test Type : select **“UG99(33)”** for hydro test pressure at 1.3 time of design pressure (normal practice).

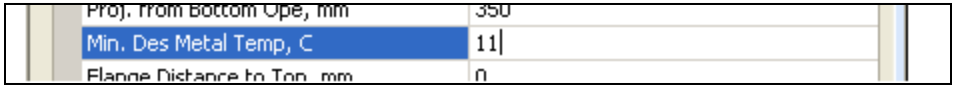


3. Hydro test Position : select “Vertical” or “Horizontal”.

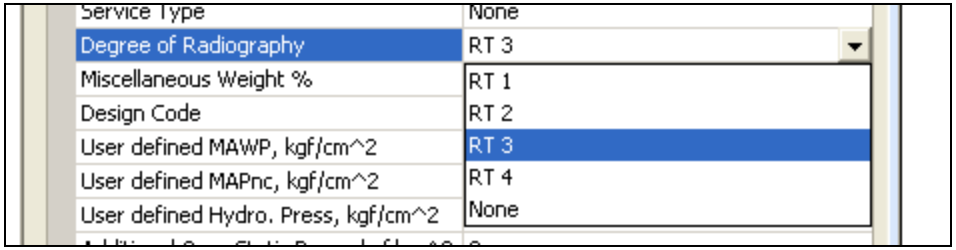
1. INPUT DATA

1.5 Design Constrain

Main Input



4. Minimum design metal temp. : input from your information, this value concern to requirement of Impact Test. (Normally input = 11º C)



5. RT1 for Full RT

RT3 for Spot RT

None for No RT

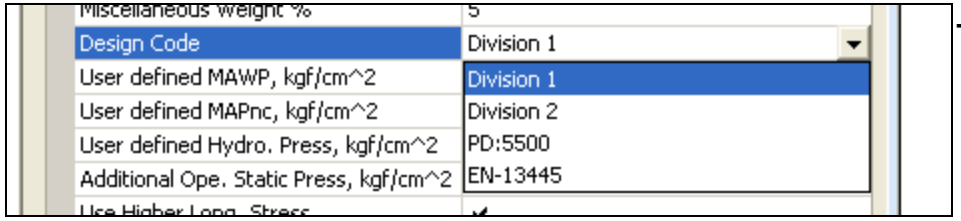
6. Miscellaneous Weight : additional weight to cover overall weight of equipment.

this additional will include in support calculation.

1. INPUT DATA

1.5 Design Constrain

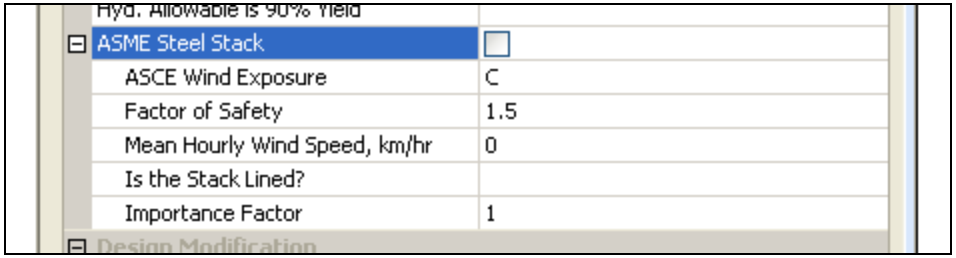
Main Input



7. Select your design code.

8. Include force from vortex when outlet flow at bottom head.

9. Select when calculate shell and tube heat exchanger.



10. Select when calculation Stack.

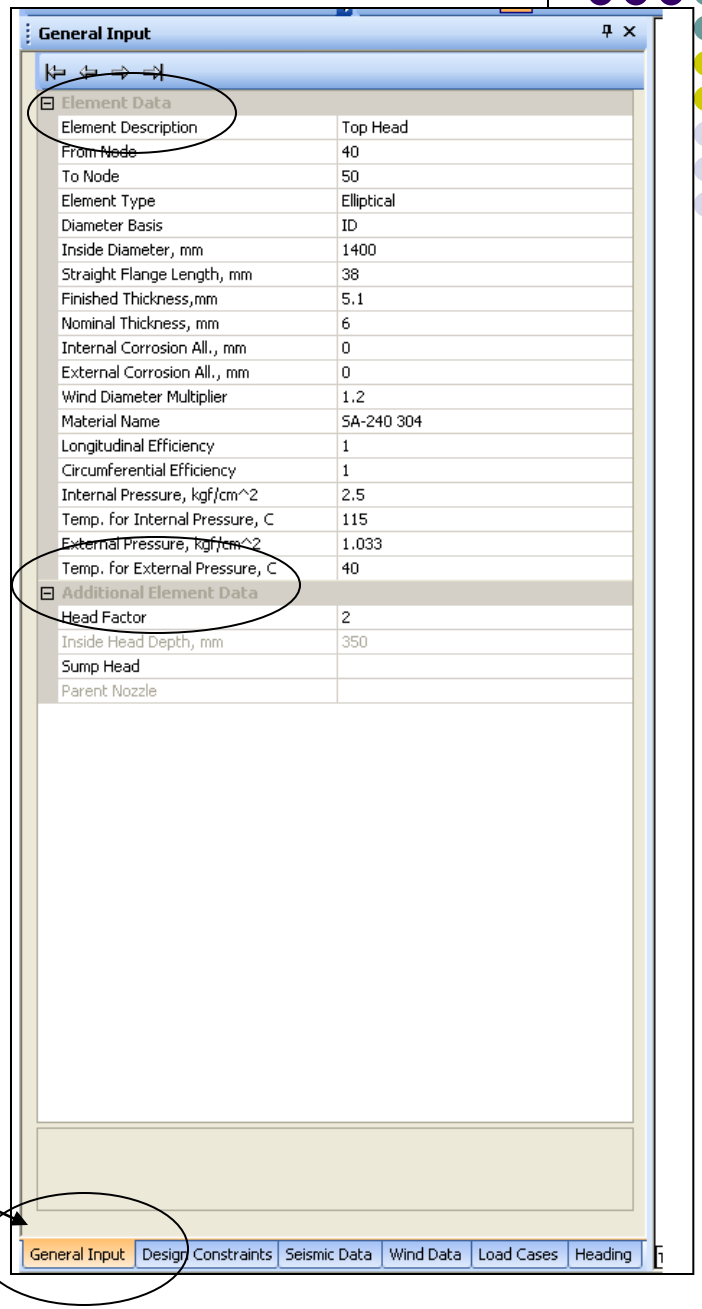
1. INPUT DATA

1.6 General Input

Input detail data for each part separate in two kind of data ;

1. Element Data : Fill all of detail design from your information.

2. Additional Data : will have different to input for each type of element.

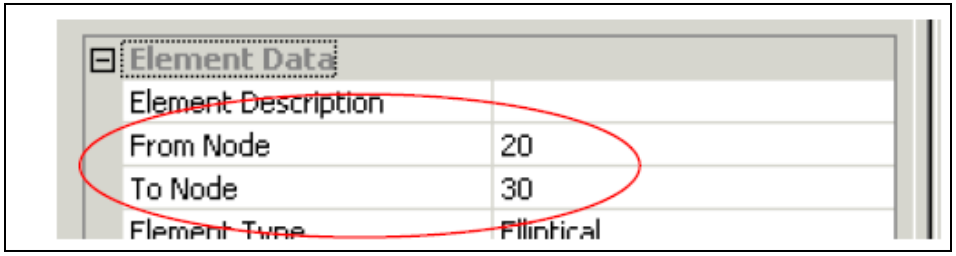


1. INPUT DATA

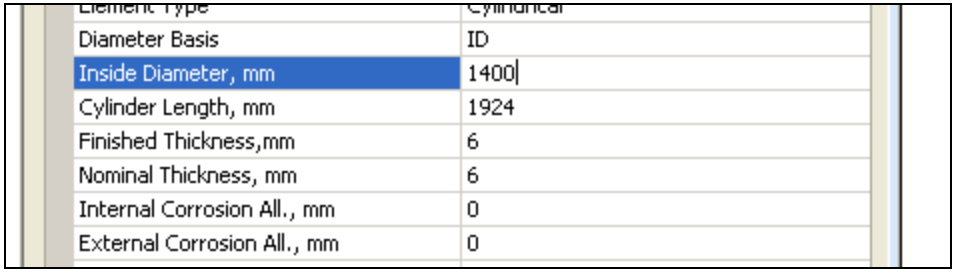
1.6 General Input

Element Data

1. Name of Element.



2. Node of Element.

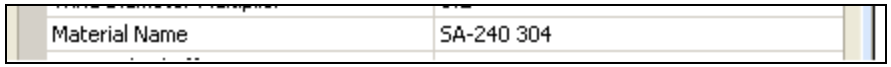


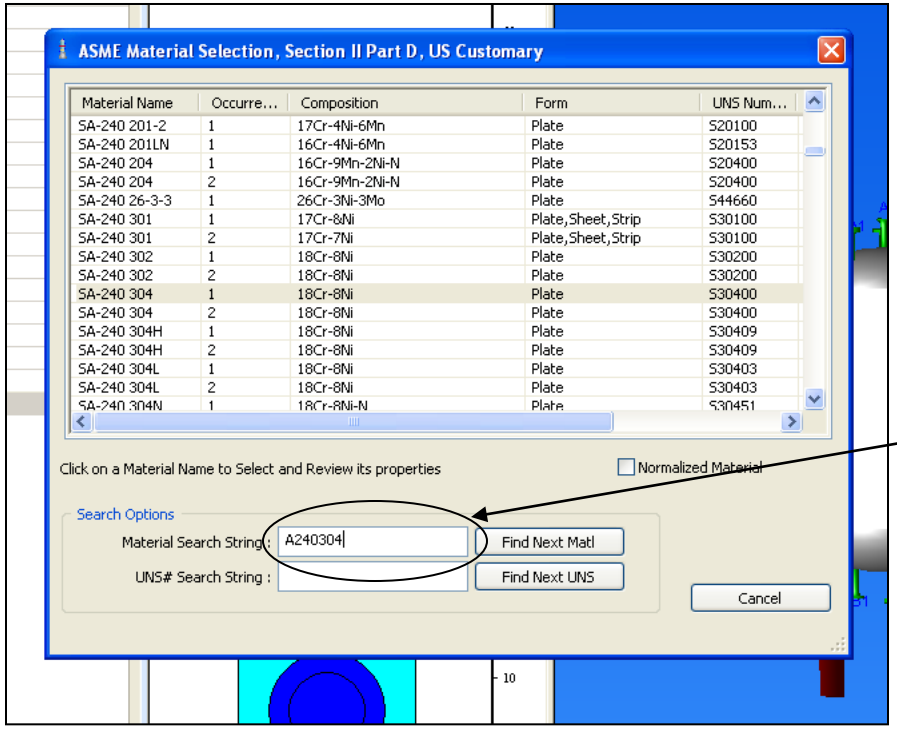
3. General input data (eq. diameter thickness)

1. INPUT DATA

1.6 General Input

Element Data





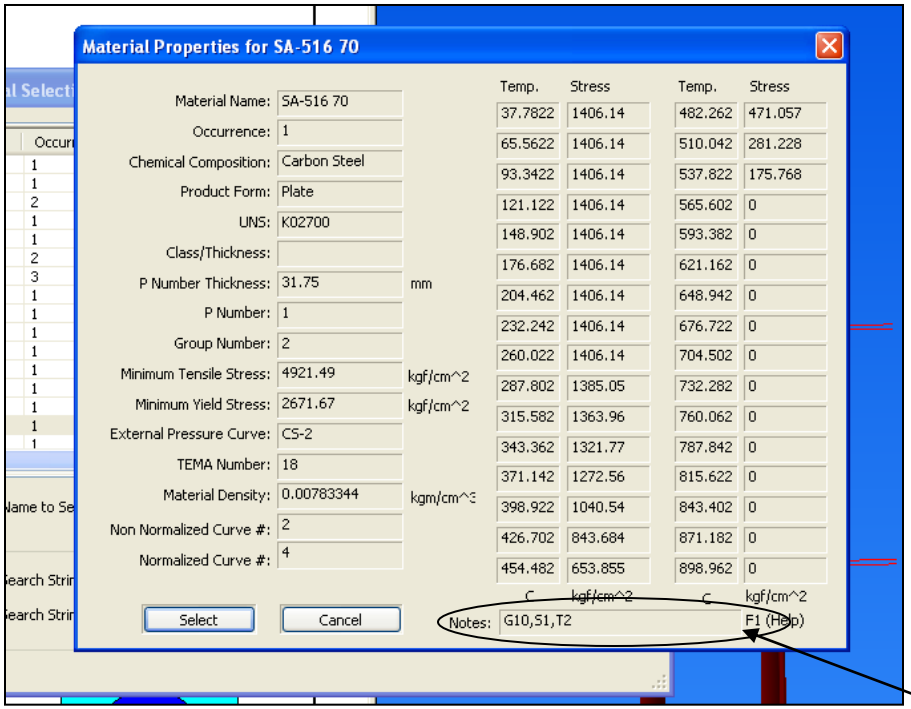
4. Material Name : to select material, click at  button at top of window.

The selection material window will appear, type your material code in white box below.

1. INPUT DATA

1.6 General Input

Element Data



- Select material you want, material properties will shown then click “Select”.

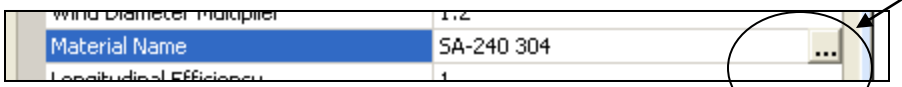
- Note : for stainless steel will have 2 or more row which show same grade material, be careful to select, it will have difference strength & stress, you can read condition of material at “Notes” in material properties window (press “F1”).

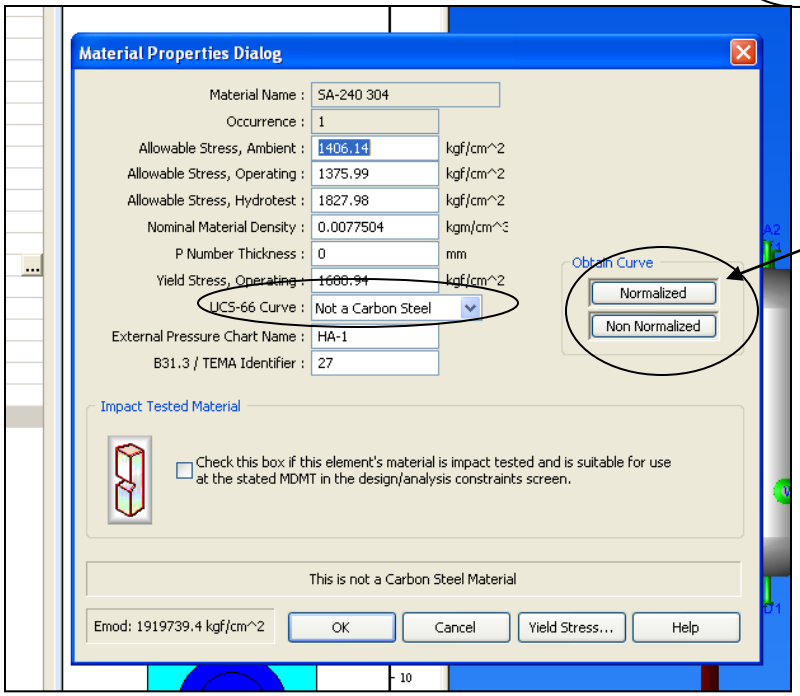
1. INPUT DATA

1.6 General Input

Element Data

Furthermore for material properties





- Click at right button, “Material Properties Dialog” will shown.

- You can select your material to be normalized here.

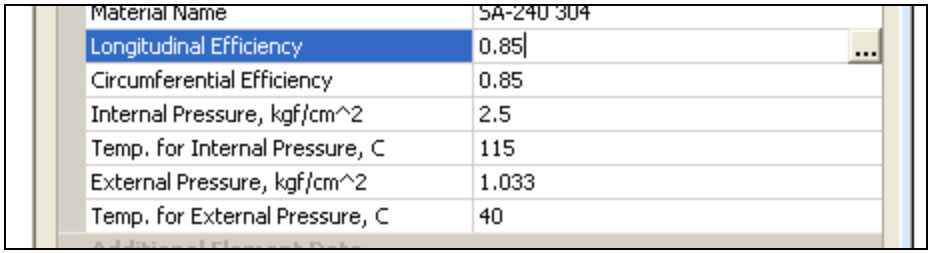
- Note : If you select nomalized for carbon steel you must change “UCS-66 Curve” by yourself

** This will concern to requirement of Impact Test **

1. INPUT DATA

1.6 General Input

Element Data

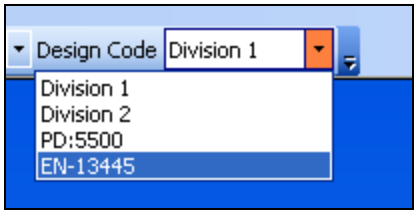


5. Other data : Fill from your information.

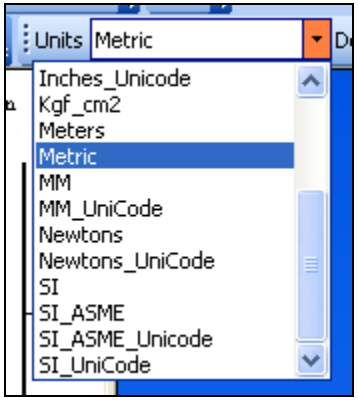
- Note : Efficiency = 1 for Full RT

  Efficiency = 0.85 for Spot RT

  Efficiency = 0.7 for No RT



6. Design Code : select design code (Division 1)



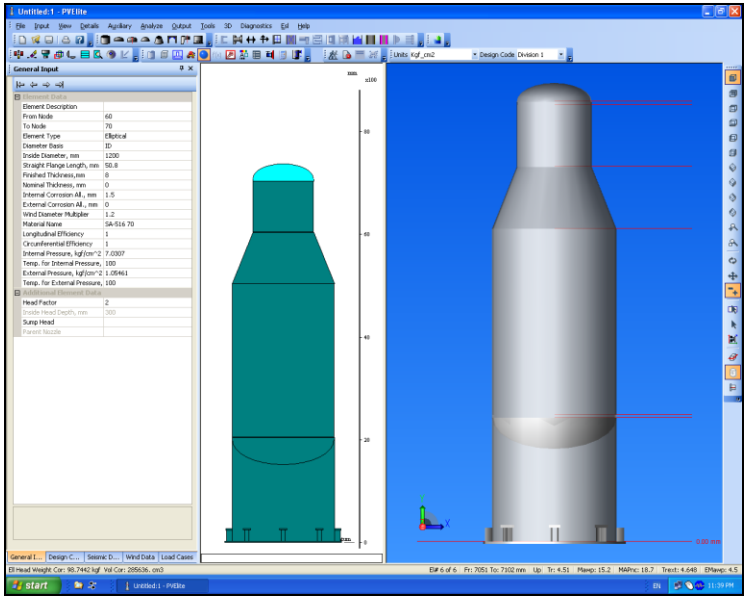
7. Units : select unit to be apply for your calculation.

2. CREATE ELEMENT

What is Element ?

- PV Elite will count Main body of vessel (eq. shell, head, skirt) to be Element and specified serial by Node (from 10 to 20 , 20 to 30 etc.)

- Create Element by click at component button



Cylindrical Shell

Ellipse Head (Ellipsoidal Head)

Torispherical Head (F & D Head)

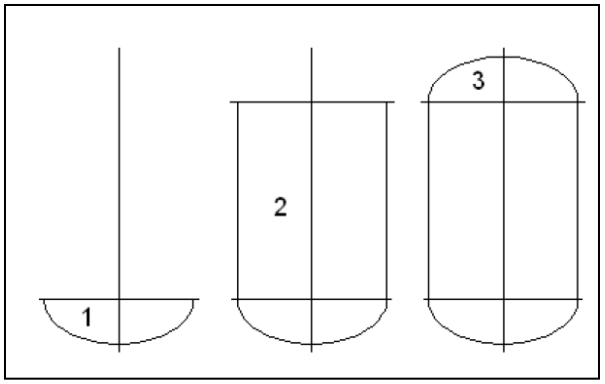
Hemispherical Head

Conical Head or Transition

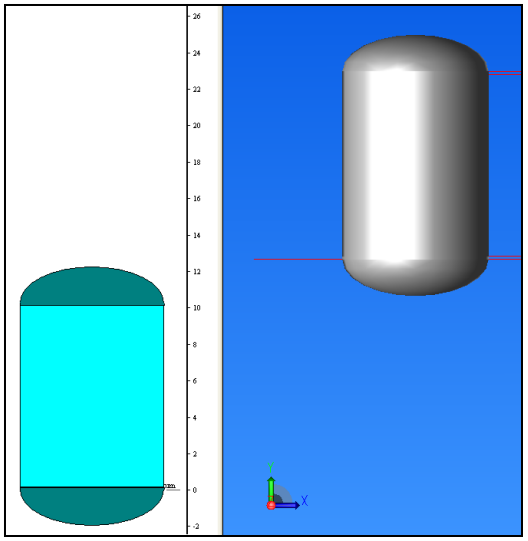
Welded Flat Head

2. CREATE ELEMENT

- Start to create your vessel by click at component button to create head then step by step to create your vessel from bottom head to shell to top head.

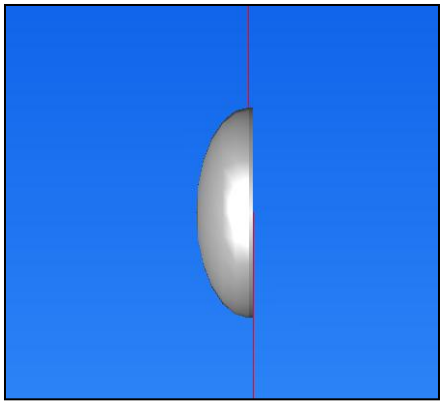


- 2D and 3D model window will show your vessel accordingly.



2. CREATE ELEMENT

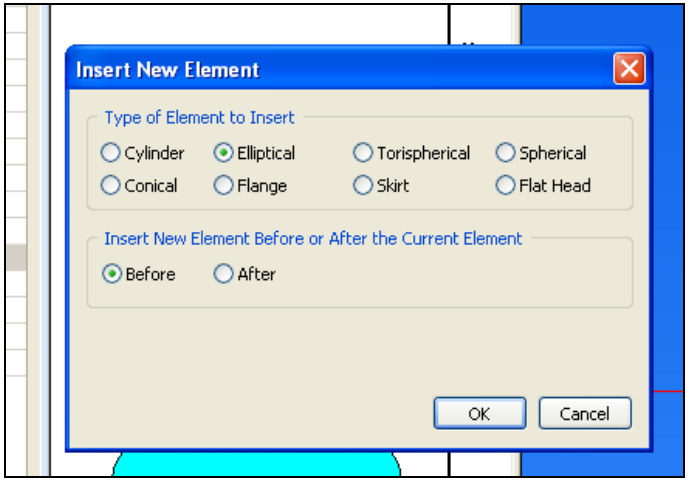
- For Horizontal vessel click at  button when create first element



- Note : you can not change vertical vessel to horizontal vessel if you have more than 1 element.

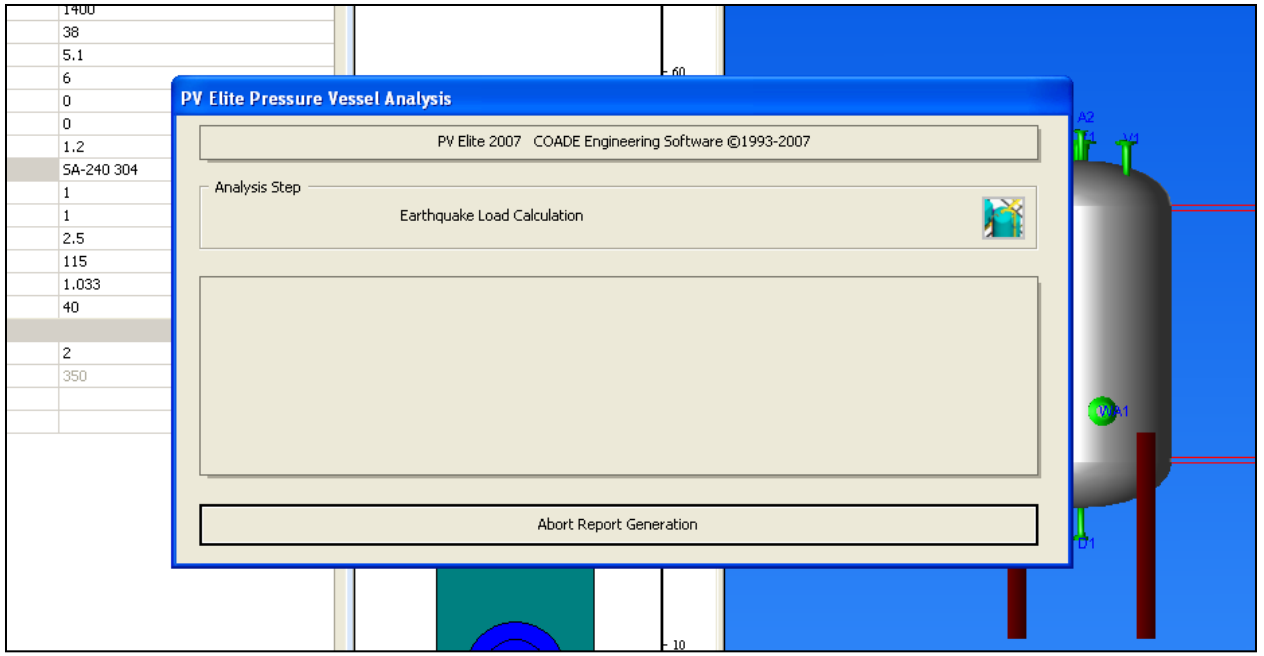
- To delete element, click  .

- To insert element, click  , “Insert New Element” will shown.



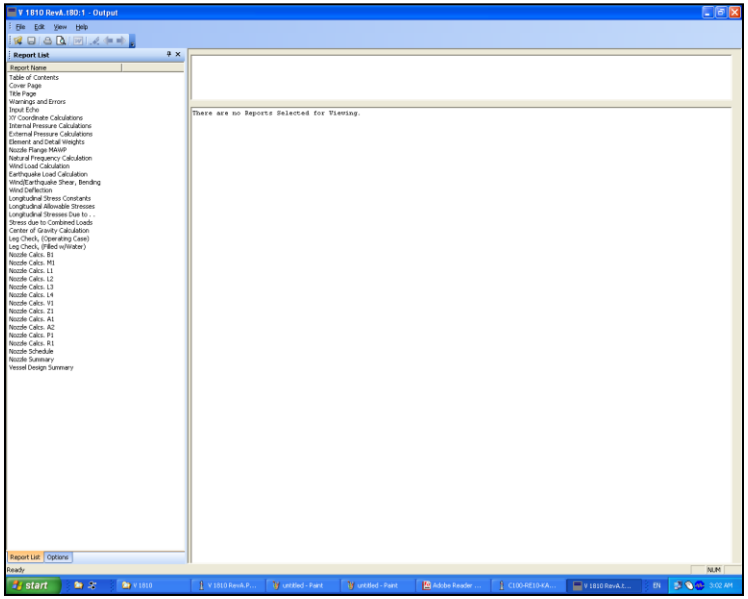
2. CREATE ELEMENT

- To calculate your vessel click  button, PV Elite will start analysis your vessel.



2. CREATE ELEMENT

- The result will shown in report window.



- If your model have failed part, PV Elite will alarm you, let adjust it as per PV Elite alarm.

3. CREATE SUPPORT

Support for your Vessel are separated in 4 kind ;

- Skirt 

- Leg 

- Bracket 

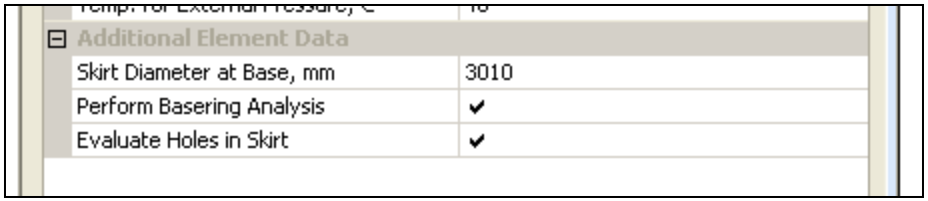
- Saddle 

3. CREATE SUPPORT

3.1 Skirt

- Skirt should be created as first because PV Elite count Skirt as one Element

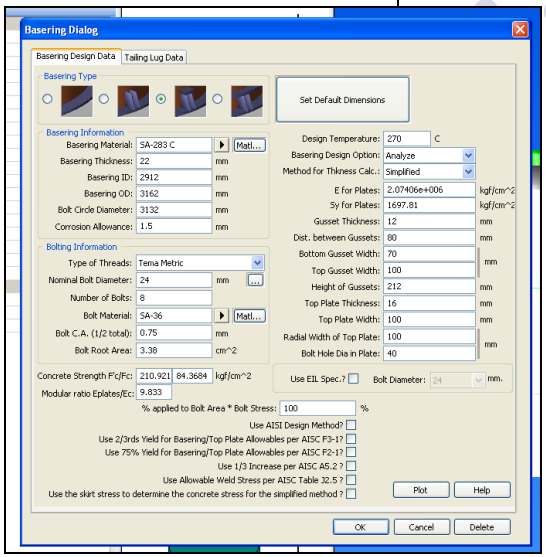
- Click at  button, Skirt will be generated as element with additional data



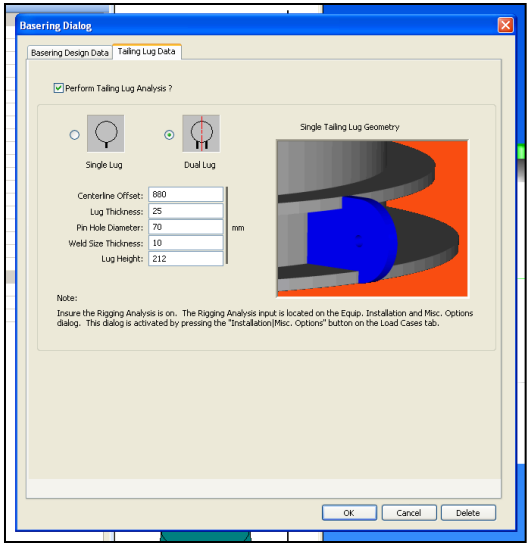
3. CREATE SUPPORT

3.1 Skirt

- Click at “Perform Basering Analysis”, Basering Dialog will shown.



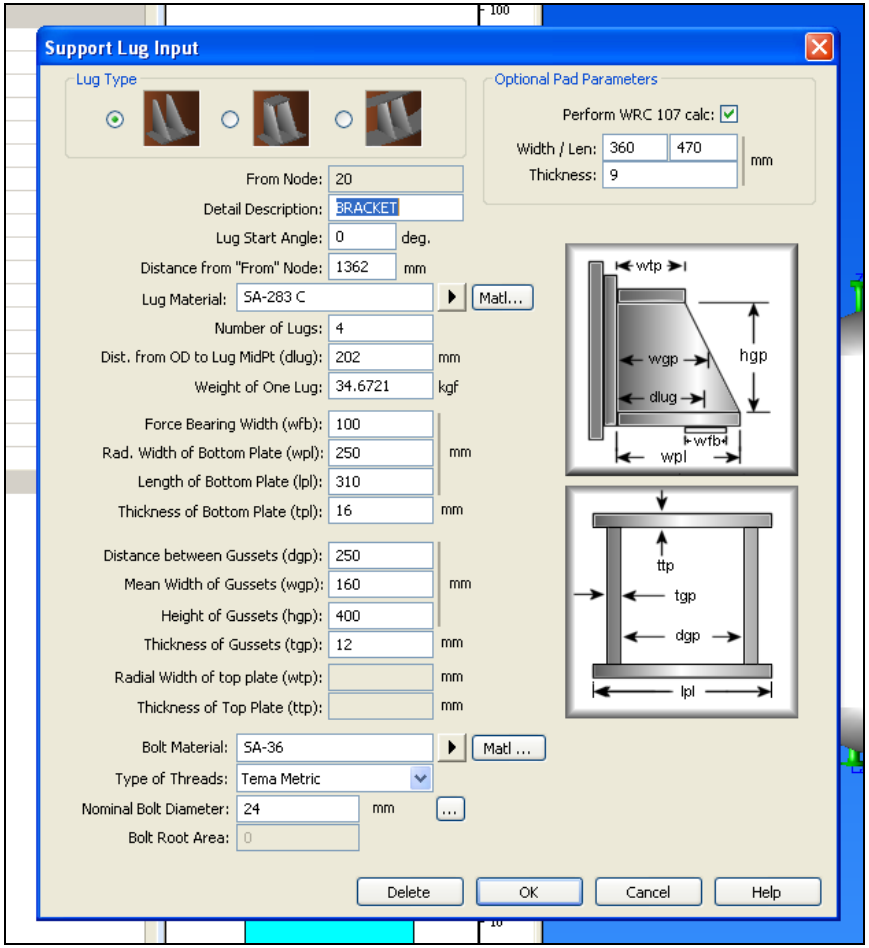
- This window will have two tab “Basering Design Data”, use for calculate anchor chair and “Tailing Lug Data”, use for calculate tailing lug.



- Input all data for your anchor chair and tailing lug note that you should input and adjust it later when perform analysis.

3. CREATE SUPPORT

3.2 Bracket



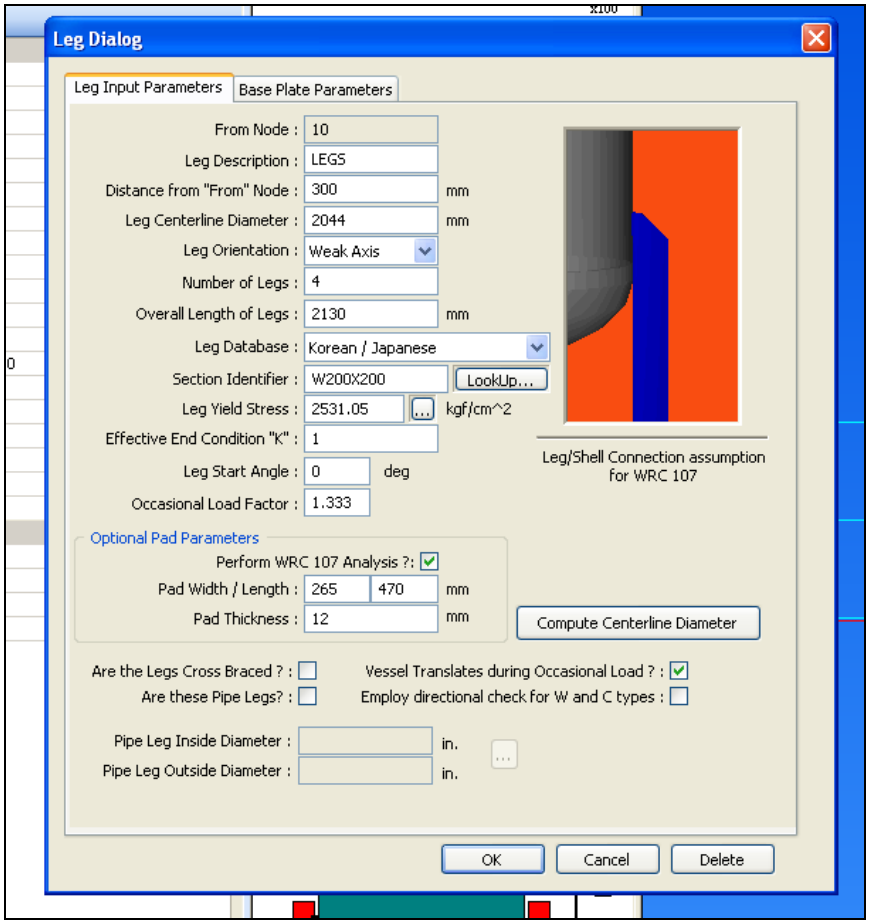
- Select shell element and click at 

- Select lug type and input dimension data as see in this window.

- This input will simply, press ”F1” for more information.

3. CREATE SUPPORT

3.3 Leg



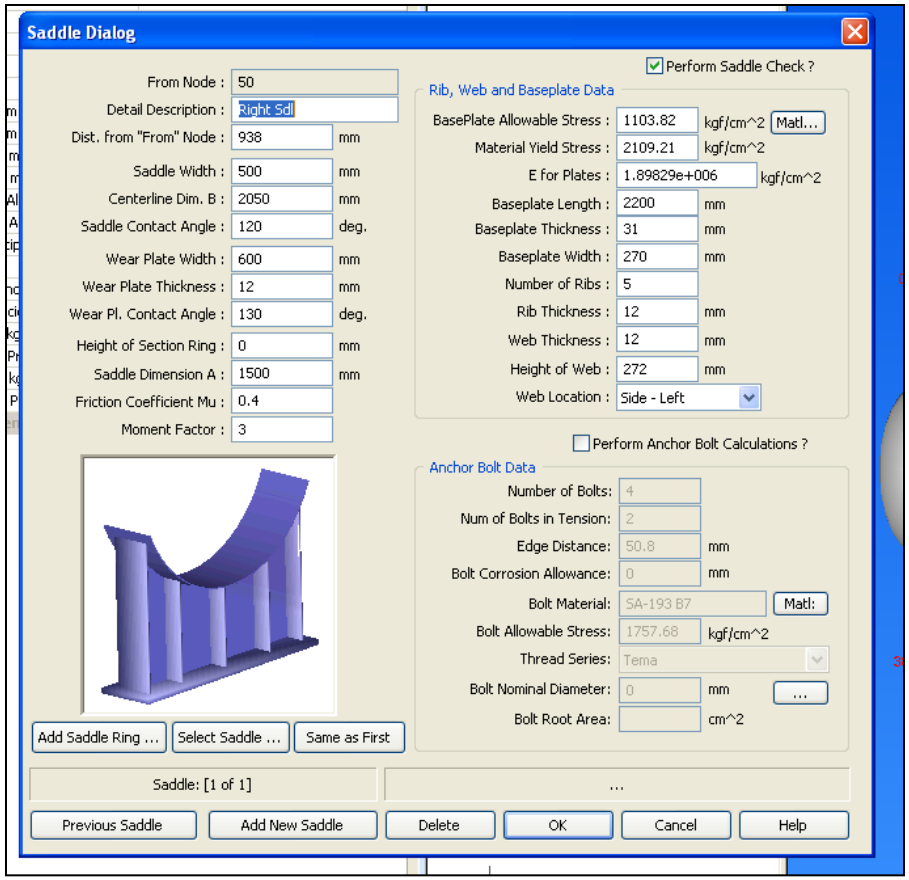
- Select shell element and click at 

- Select lug type and input dimension data as see in this window.

- This input will simply, press ”F1” for more information.

3. CREATE SUPPORT

3.3 Saddle



- Select shell element and click at

- Select lug type and input dimension data as see in this window.

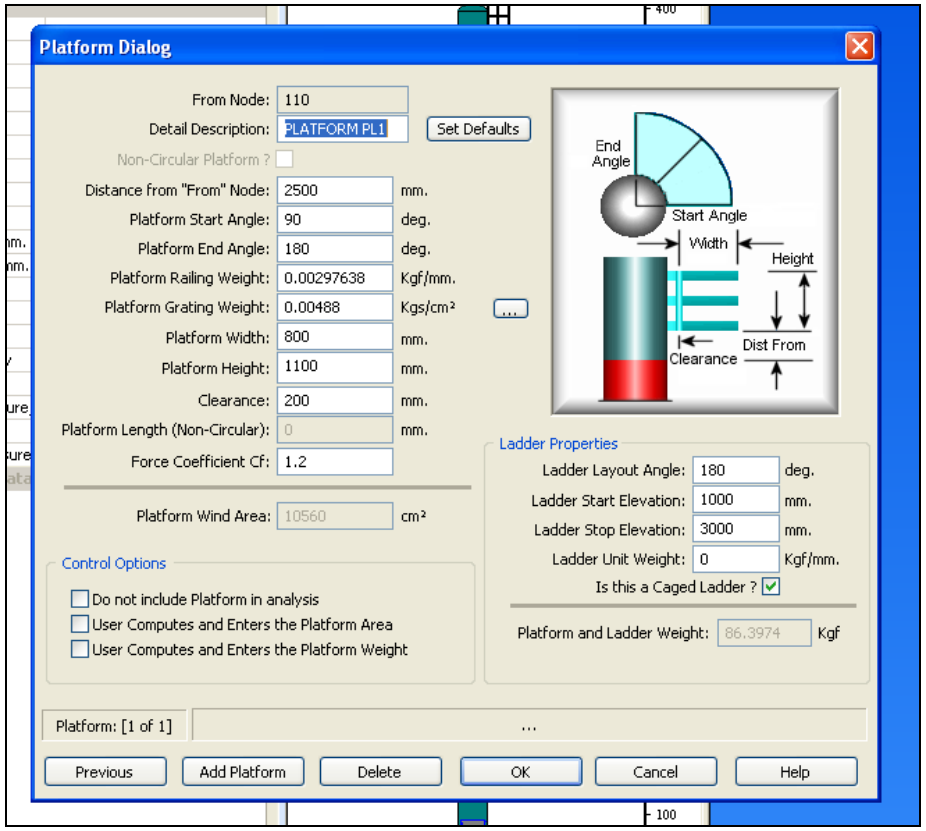
- This input will simply, press ”F1” for more information.

4. OPTIONAL INPUT

- Platform 
- Liquid 
- Insulation 
- Stiffener ring 
- Packing 
- Tray 
- Lining 
- Half-Pipe Jacket 

4. OPTIONAL INPUT

4.1 Platform



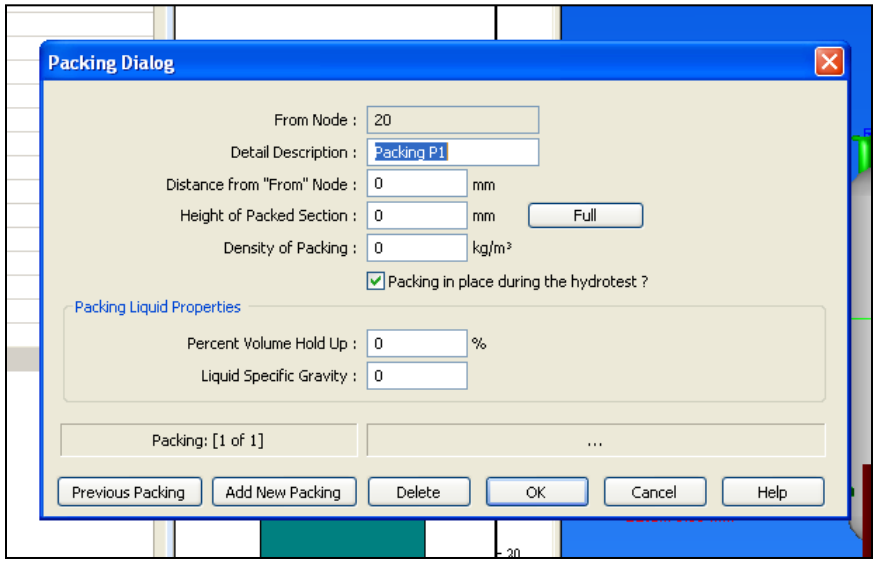
- Select shell element and click at 

- Input dimension data as see in this window.

- This input will simply, press ”F1” for more information.

4. OPTIONAL INPUT

4.2 Packing



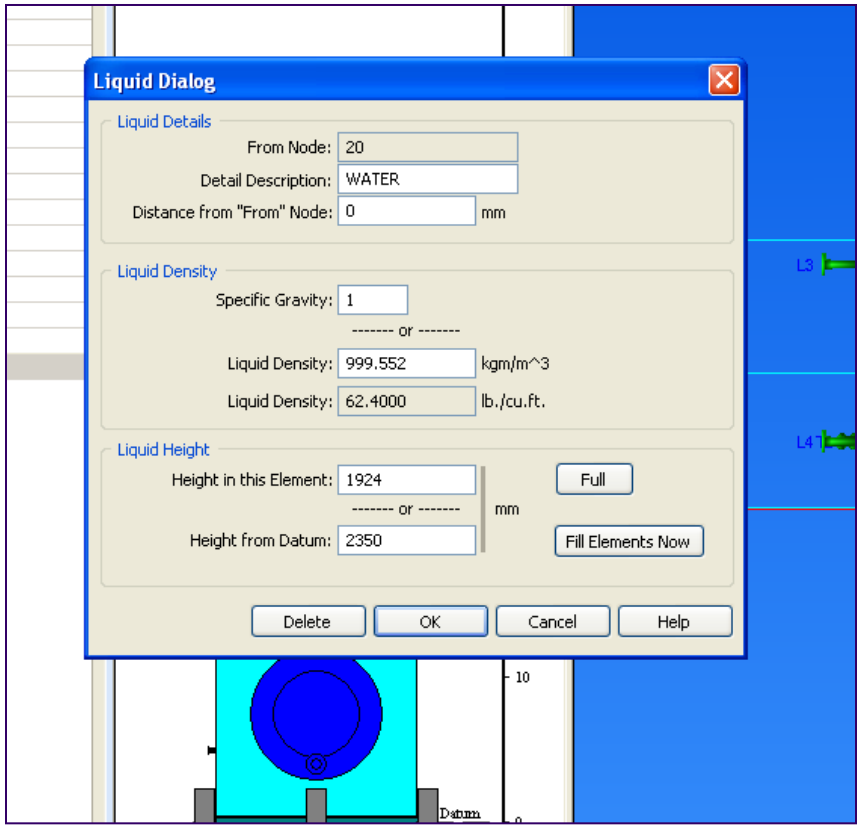
- Select shell element and click at

- Input Packing data.

- This input will simply, press ”F1” for more information.

4. OPTIONAL INPUT

4.3 Liquid



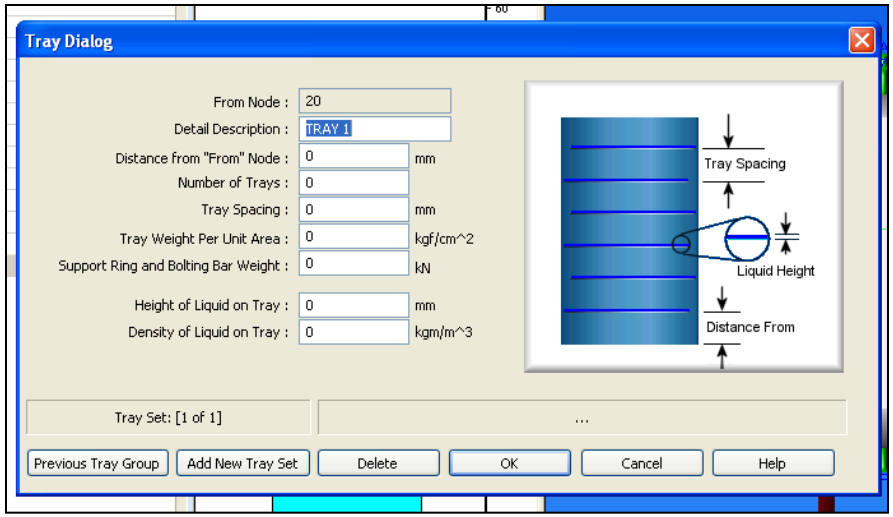
- Select element to be fill fluid and click at 

- Input liquid data and how much you fill liquid in your element.

- This input will simply, press ”F1” for more information.

4. OPTIONAL INPUT

4.4 Tray



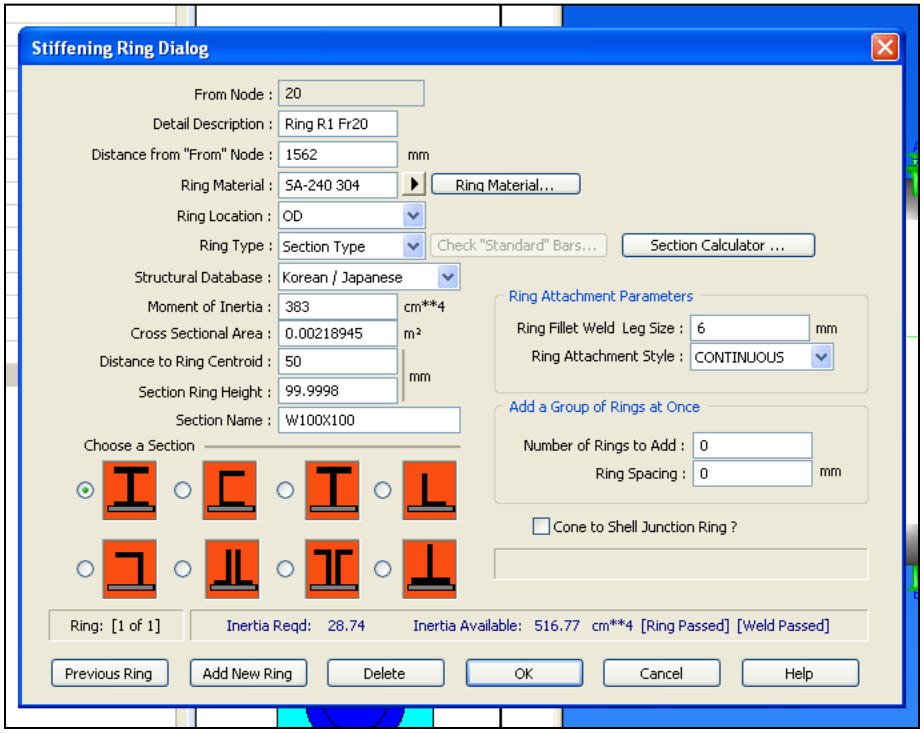
- Select shell element and click at 

- Input Tray data.

- This input will simply, press ”F1” for more information.

4. OPTIONAL INPUT

4.5 Stiffener Ring



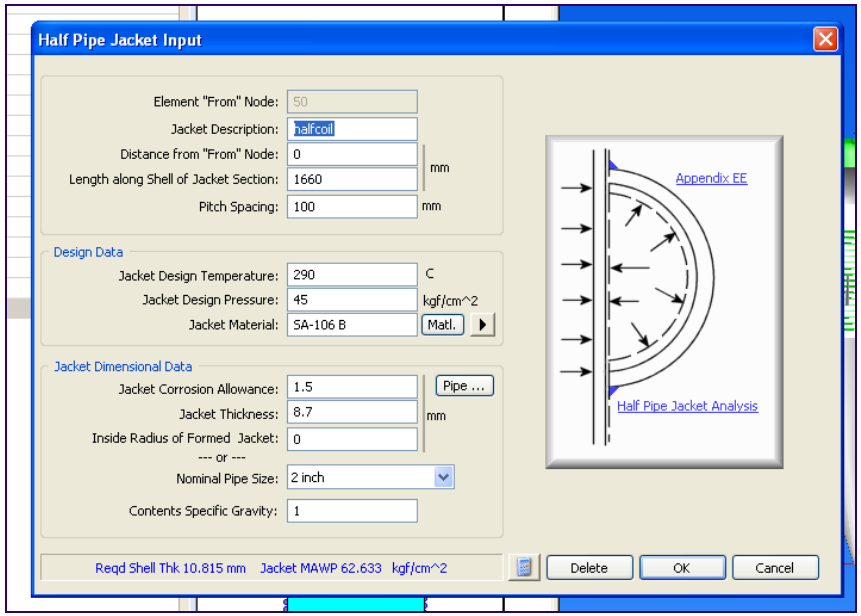
- Select shell element and click at 

- Select and input dimension data as see in this window.

- This input will simply, press ”F1” for more information.

4. OPTIONAL INPUT

4.6 Half-Pipe Jacket



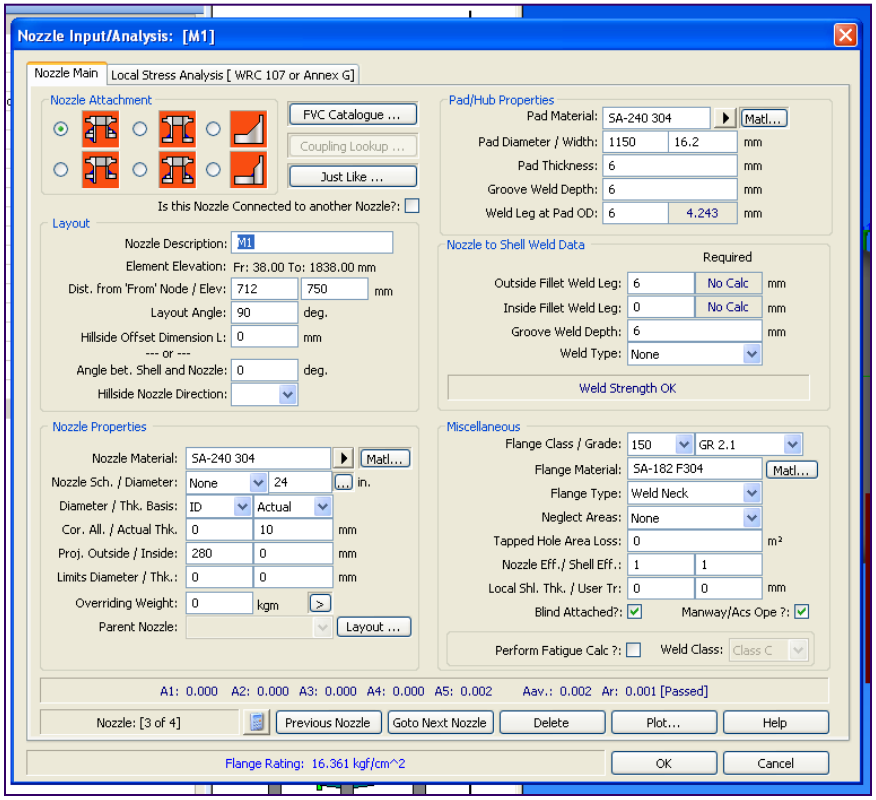
- Select shell element and click at 

- Input dimension data as see in this window.

- This input will simply, press ”F1” for more information.

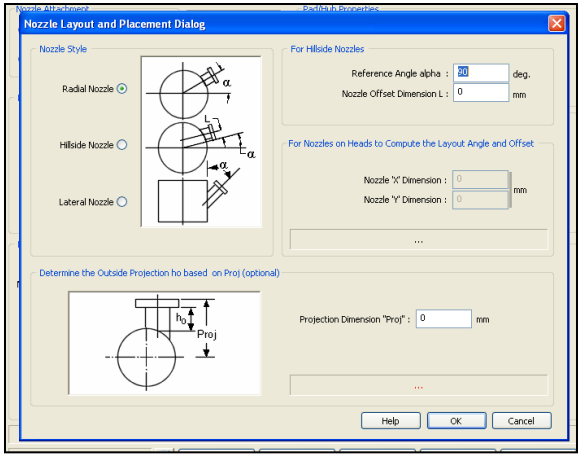
5. Nozzle

- Nozzle can be created at every element you want by click at  button.
- Nozzle input window will appear.



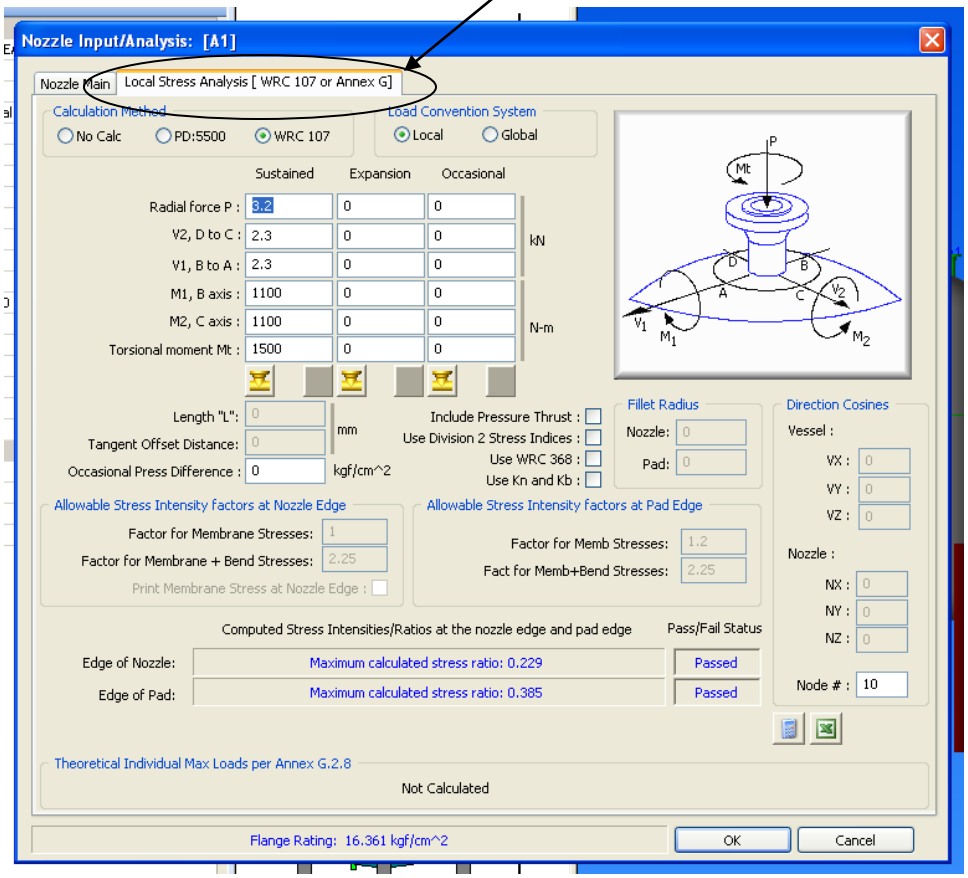
- Select attach type of your nozzle
- Input all data, but for layout of nozzle you can click at 

It will easy to input your orientation.



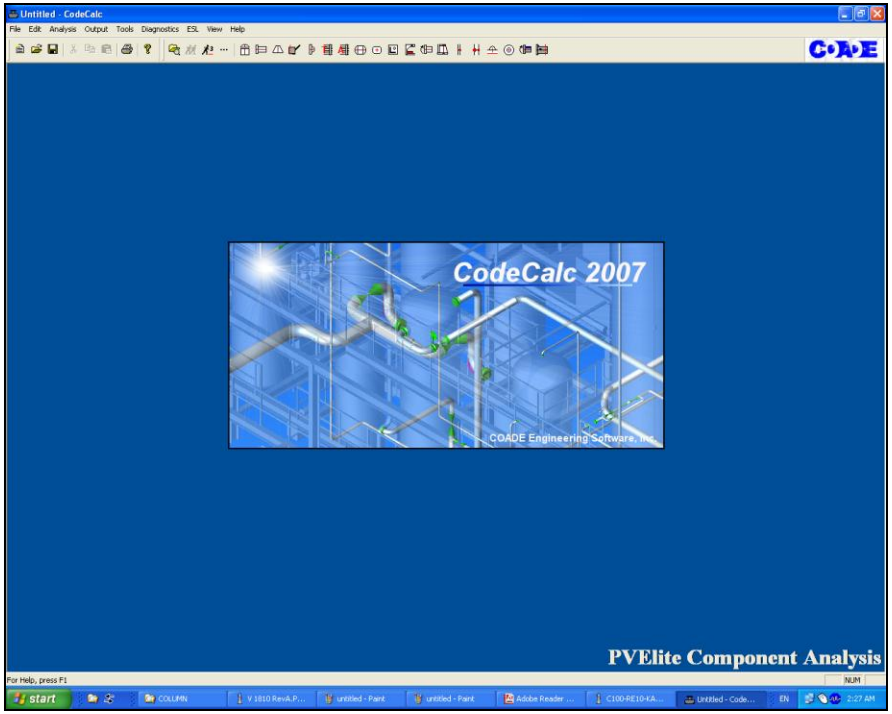
5. Nozzle

- Input external load for your nozzle by select WRC 107



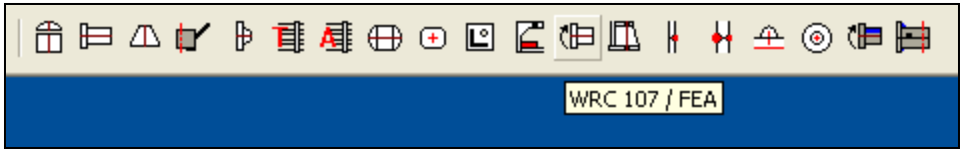
6. Code Cal

- Code Cal is support program for PV Elite with analyze each part individually.
- Useful for part which PV Elite can not create such as Lifting Lug.
- Click at  button, Code Cal program will start.

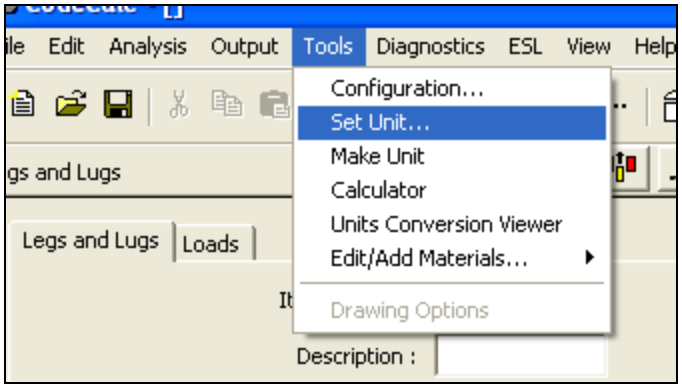


6. Code Cal

- All feature of Code Cal are shown at top of window, stop your cursor to see what each feature is.

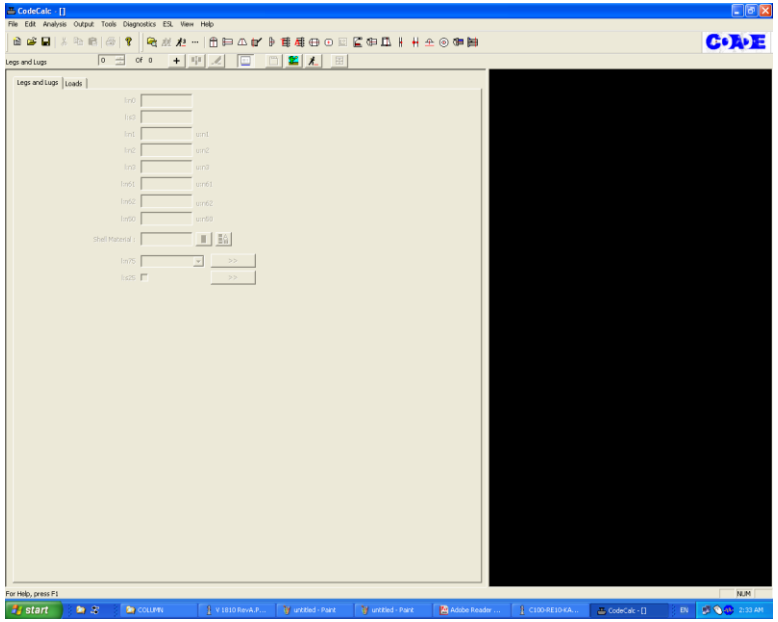


- To set unit, click at tool bar



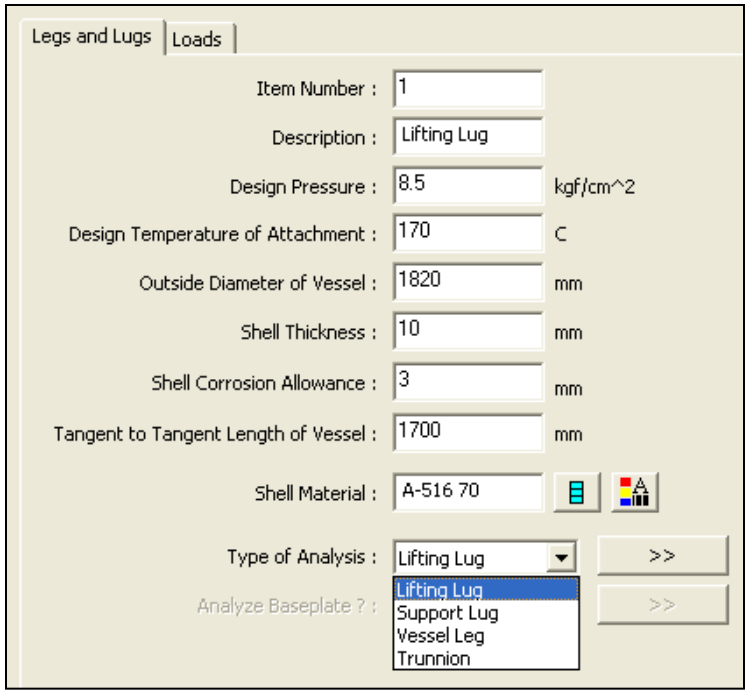
6. Code Cal

- Important feature is Legs and Lugs  (use to calculate lifting lug), input window will appear (take a few second)

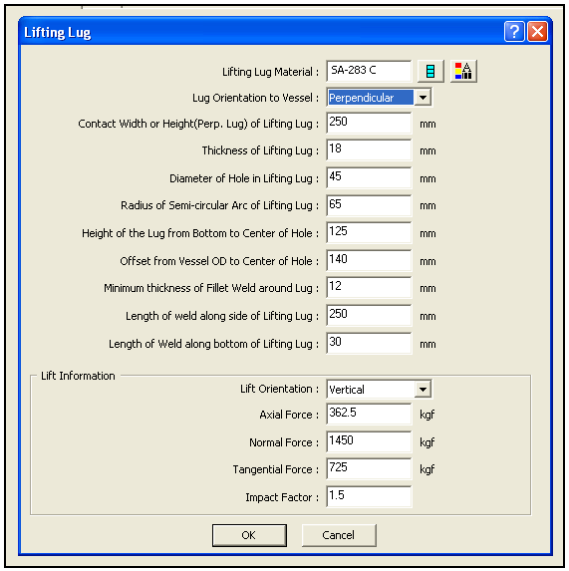


6. Code Cal

- Click at  button to create analysis file

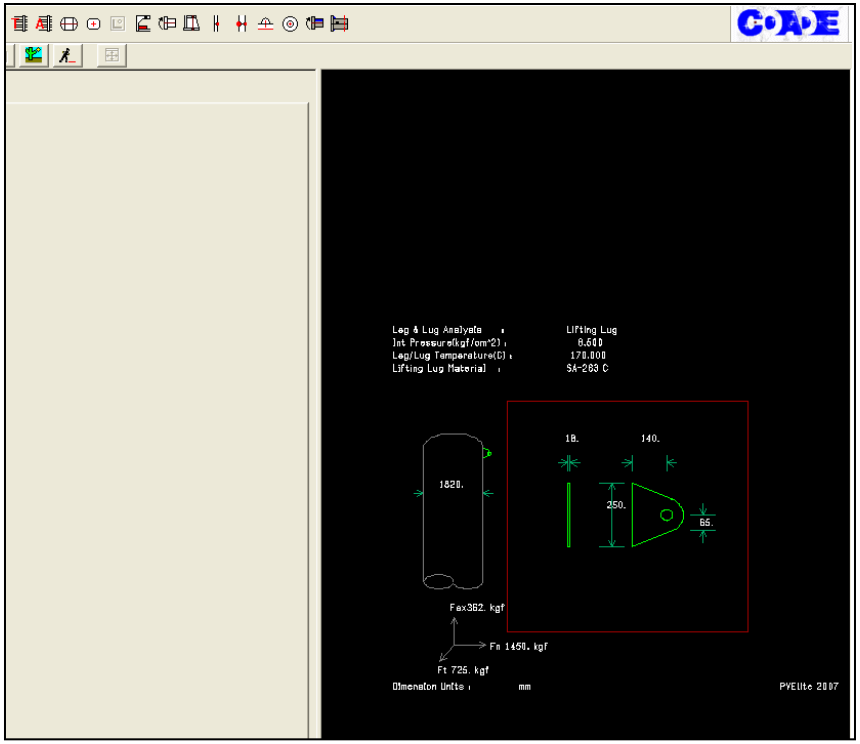


- Input general data and select type of analysis, select “Lifting Lug”
- Input general data and select type of analysis, select “Lifting Lug”
- Lifting Lug input window will appear.



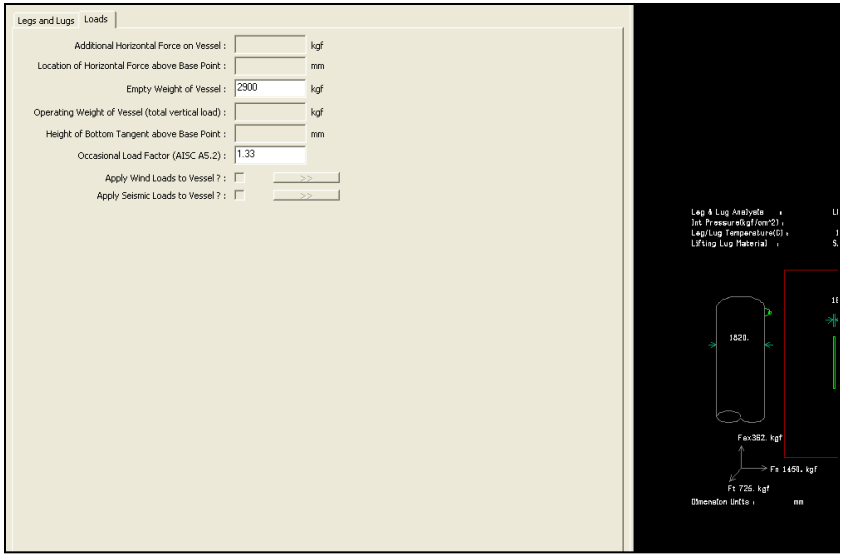
6. Code Cal

- Select type of lifting “Perpendicular” or “Flat” , press “F1” to see detail input.
- Input all data according, if your input data are reasonable, the outline drawing for your lifting lug will shown at the black area.



6. Code Cal

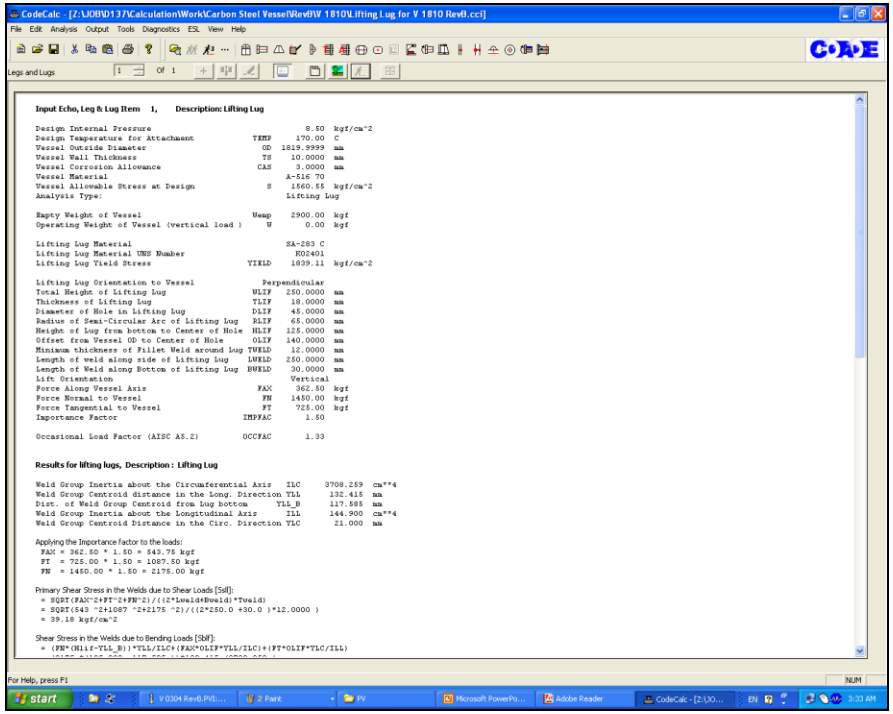
- Input empty weight at loads window.



- Occasional Load Factor = 1.33 (default value)

6. Code Cal

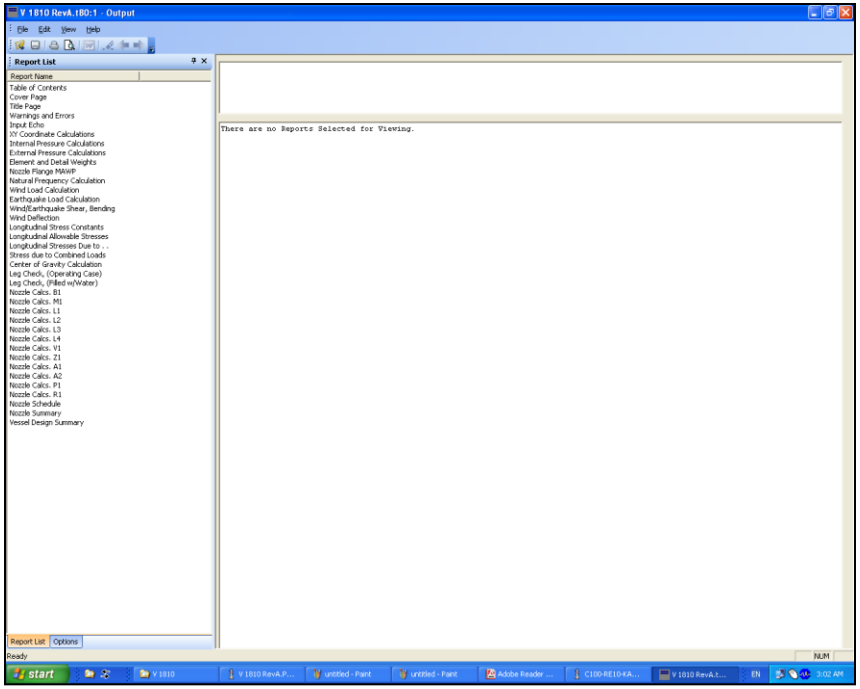
- Click  button to perform analysis, the result will shown.

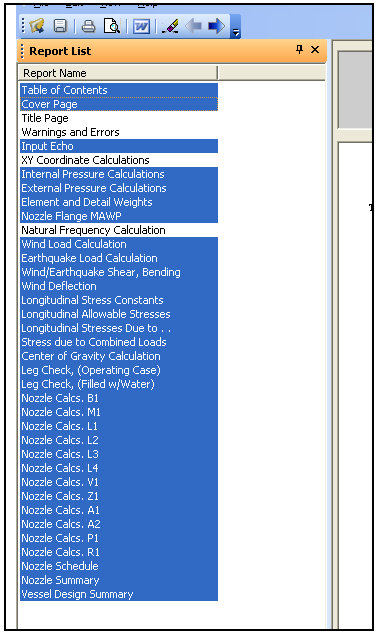


- Click  button to back to input window for adjust your data.

7. Convert to Word

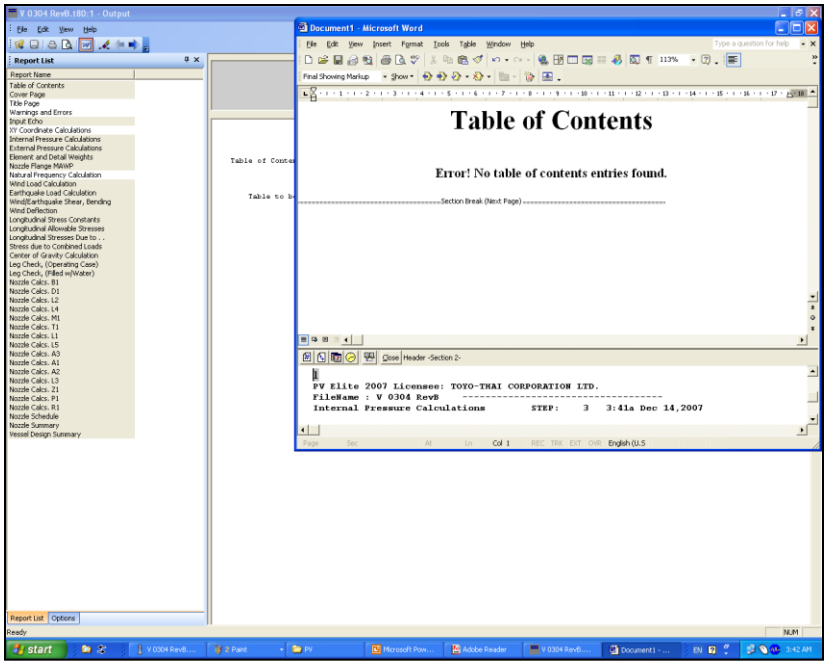
- After finish all of calculation and result window are shown.
- Collect the result that you want to report (by press “ctrl” and select)



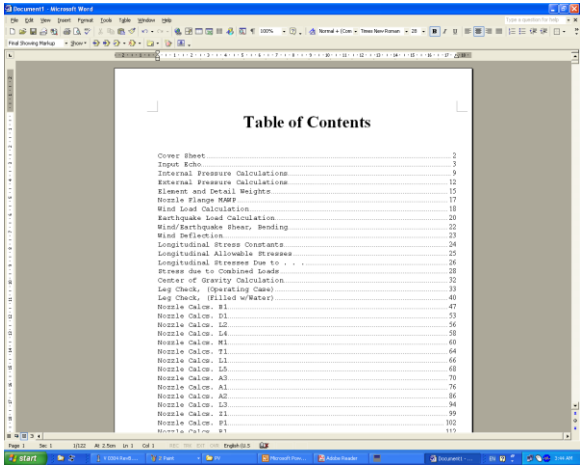


7. Convert to Word

- Click  button to convert your selected result to Word file, PV Elite will start to convert to Word file, this take a few second.



- Click button to convert your selected result to Word file, PV Elite will start to convert to Word file, this take a few second.



FINISH!!!!!

8. TRY EXAMPLE

- Further more, you can read the PV Elite User Guide and PV Elite Manual (full), at folder which your PV Elite program is located on.

- There are many example which very useful for your study case at PV Elite folder same as PV Elite Manual, let try it.

- There are many detail which don’t mention in this presentation such as Girth Flange and S&T heat exchanger, so please try the best in your exam.

THANK YOU VERY MUCH