به نام خدا

INTRODUCTION

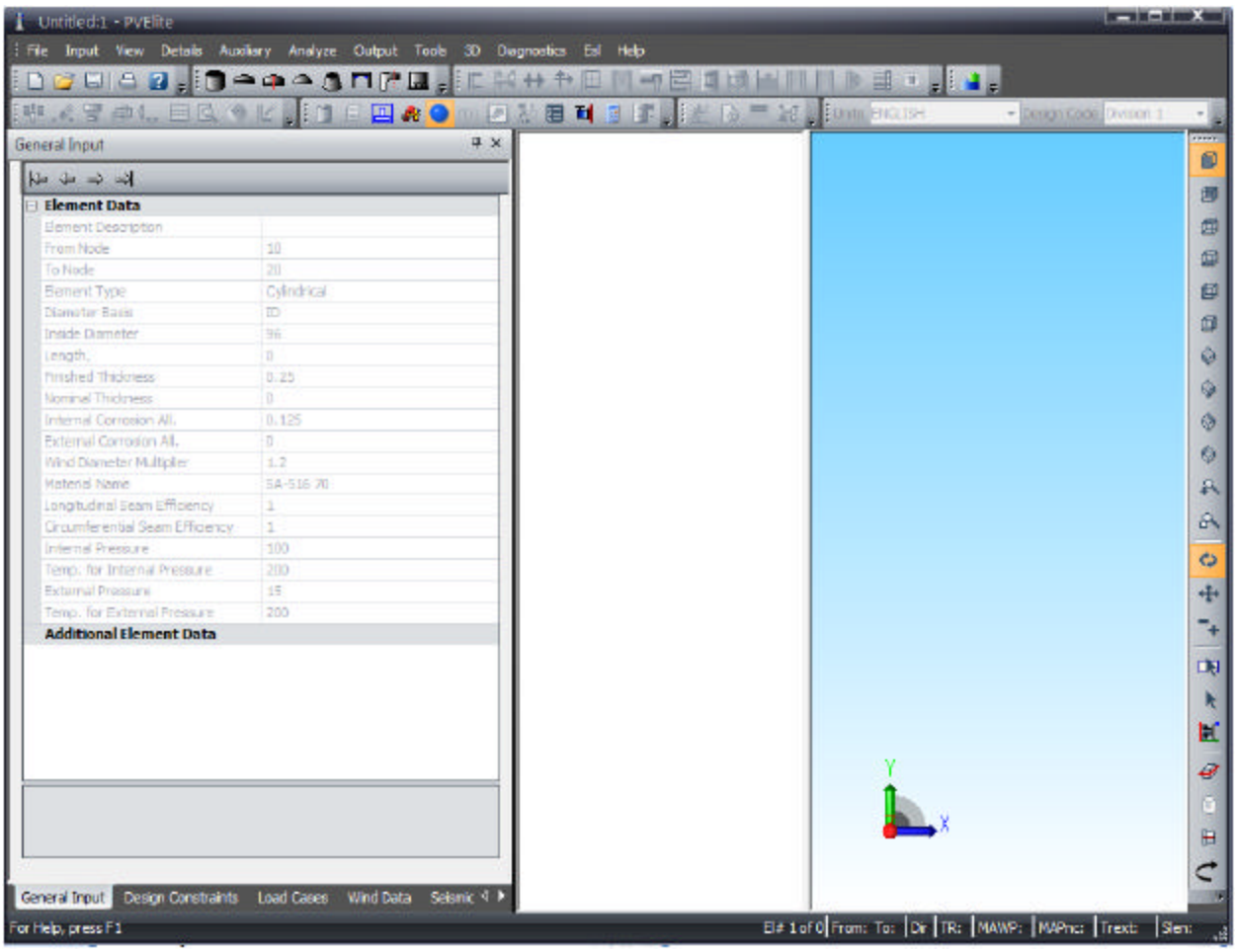
The 2009 version of **PV Elite** introduces an updated user interface. The interface is the area on the screen where you, the user enter all the information such as dimensions, materials and temperatures etc. For those of you who are familiar with the previous user interface, there are a number of changes to which you will need to become accustomed.

To make matters clear, we shall deal with the most important features of the new interface, and show the step by step procedures required to get a model built. We do not cover every aspect of the data entry, but will provide you enough information for you to understand the basic principles. Remember, this is only a *Quick Start* procedure. Further details in operating the program are to be found in the *On-line Documentation.*

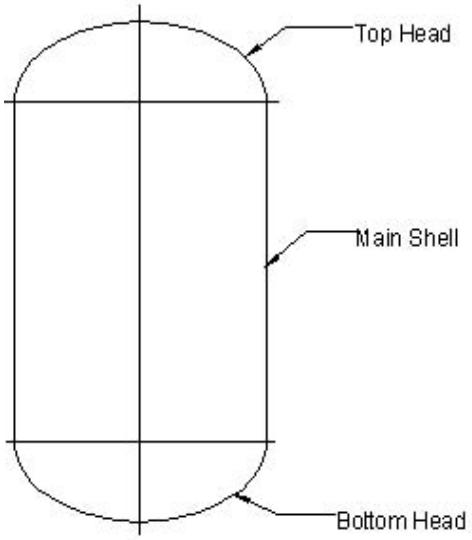
Also, a little experimentation on your part will soon have you up and running with confidence.

A Quick Look Around

Once you have **PV Elite** up and running, you will see this screen:

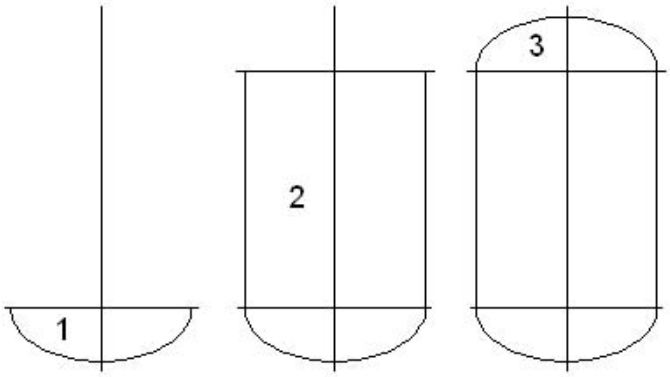


Before getting into any details let us simply build a simple vertical vessel. Our first attempt is a vessel with 3 parts as shown below:



The vessel is built in **PV Elite** from the bottom to the top; or, the case of a horizontal vessel, from left to right. We shall consider building a horizontal vessel later.

This is how the vessel is built:



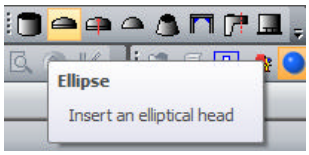
Component Buttons

Near the top of the main input window, there is a row of buttons called *Element* toolbar:

Working from left to right, the components referred are:

Cylindrical Shell	Ellipse Head (Ellipsoidal Head)	Torispherical Head (F & D Head)
Hemispherical Head	Conical Head or Transition	Welded Flat Head

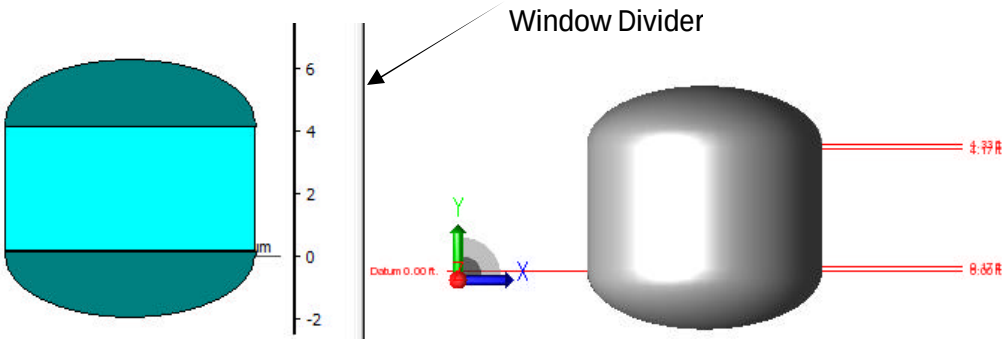
We shall not deal with the remaining icons (buttons) here, as I am sure you are anxious to start building your first vessel. Before we leave this subject however, if you let your mouse cursor hover over any of the buttons, you will get a 'tool tip', which briefly describes the function of the button:



Building the vessel from the bottom up

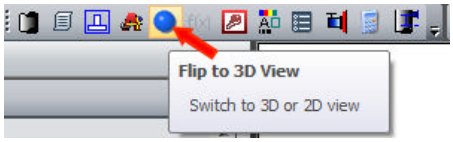
1. Bottom head: Click the ellipse head button: 
2. Main shell: Click the cylinder button: 
3. Top head: Click the ellipse head button: 

If you look at the right hand window, you should see the see the 3D model:



You will see that you get two models on the screen. On the left we have a 2D model, and on the right we have a 3D model. By moving the adjustable *Window Divider*, you can enlarge or reduce the left or right window; you can slide the window divider left or right. To do this, simply place your mouse cursor over the frame, hold down the left mouse button, and slide the vertical window divider left or right to suit your purpose.

You can also switch between 3D or 2D views by pressing on the *Flip to 3D view* button:



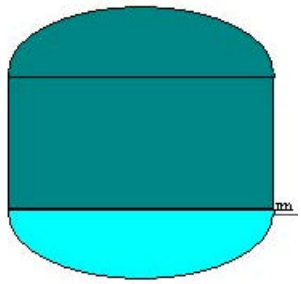
Note: If you want to go back to have both views in your screen; click on *View* and then *Split* on the menu bar, grab the window divider and stretch it till you have both 3D and 2D views.

Dimensions

The actual height and other dimensions are probably not what you want for your vessel. **PV Elite** made certain assumptions as you built the model, including diameter, thicknesses, lengths and materials.

Before we look at the dimensions, please click your mouse on the bottom head, either on the 2D or 3D model. This will **select** the bottom head as the **current** component.

If you look at the 2D model, you will see that it has turned a light green color:



Let us look at the dimension **PV Elite** chose for the current component (light green in the illustration above). The details will look like this[1]:

General Input

Element Data	
Element Description	
From Node	10
To Node	20
Element Type	Elliptical
Diameter Basis	ID
Inside Diameter, in.	96
Straight Flange Length, ft.	0.166667
Finished Thickness,in.	0.25
Nominal Thickness, in.	0
Internal Corrosion All., in.	0.125
External Corrosion All., in.	0
Wind Diameter Multiplier	1.2
Material Name	SA-516 70
Longitudinal Seam Efficiency	1
Circumferential Seam Efficiency	1
Internal Pressure, psig	100
Temp. for Internal Pressure, F	200
External Pressure, psig	15
Temp. for External Pressure, F	200
Additional Element Data	
Head Factor	2
Inside Head Depth, in.	24
Sump Head	
Parent Nozzle	

Here are the major assumptions[2]:

Inside Diameter:	96 inches
Straight Flange:	0.1667 feet – 2 inches
Finished Thickness:	0.25 inches
Internal Corrosion:	0.125 inches
External Corrosion:	0
Internal Pressure:	100 psi
External Pressure:	15 psi
Temperatures:	100 °F
Head Aspect Ratio:	2 (Head Factor)

There is other information, but let us now make some changes to suit our vessel. First we change the dimensions and the pressure:

New Values:

Inside Diameter:	60 inches
Finished Thickness:	0.5 inches
Internal Corrosion:	0.0625 inches
External Corrosion:	0.0625 inches
Internal Pressure:	120 psi
Temperatures:	200 °F

[1] Your screen may look different, but follow along, and it will become clear

[2] If you are accustomed to working in the metric system, we shall address that situation shortly

Click on the left side of the *Inside Diameter* field:

Inside Diameter, in.	96
Straight Flange Length, ft.	0.166667
Finished Thickness,in.	0.25

Now simply enter the new diameter by typing *60*:

Inside Diameter, in.	60
Straight Flange Length, ft.	0.166667
Finished Thickness,in.	0.25

Now press the <*Enter*> key on your keyboard twice, and this will move the cursor down to the next field:

Inside Diameter, in.	60
Straight Flange Length, ft.	0.166667
Finished Thickness,in.	0.25

If you do not want to change the straight flange, press the <*Enter*> key again, and it will move to the next field. You should be able to complete the entries for your new dimensions:

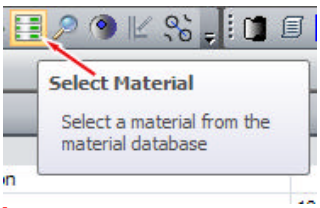
Element Data	
Element Description	BOTTOM HEAD
From Node	10
To Node	20
Element Type	Elliptical
Diameter Basis	ID
Inside Diameter, in.	60
Straight Flange Length, ft.	0.166667
Finished Thickness,in.	0.5
Nominal Thickness, in.	0.5
Internal Corrosion All., in.	0.0625
External Corrosion All., in.	0.0625
Wind Diameter Multiplier	1.2
Material Name	SA-516 70
Longitudinal Seam Efficiency	1
Circumferential Seam Efficiency	1
Internal Pressure, psig	120
Temp. for Internal Pressure, F	200
External Pressure, psig	15
Temp. for External Pressure, F	200
Additional Element Data	
Head Factor	2
Inside Head Depth, in.	15
Sump Head	
Parent Nozzle	

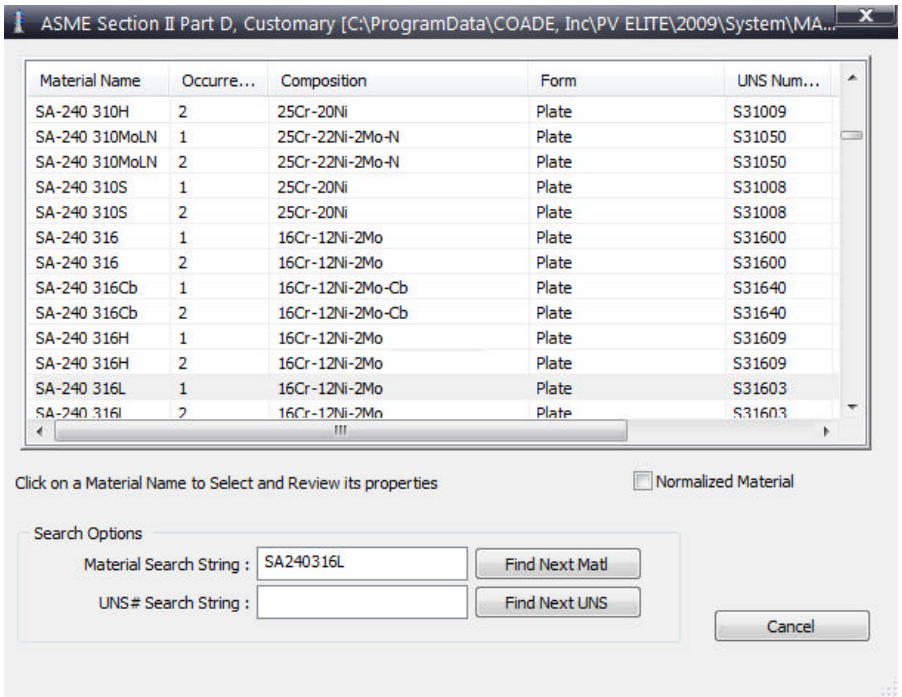
Make sure you have the same dimensions as shown above.

Material Specification

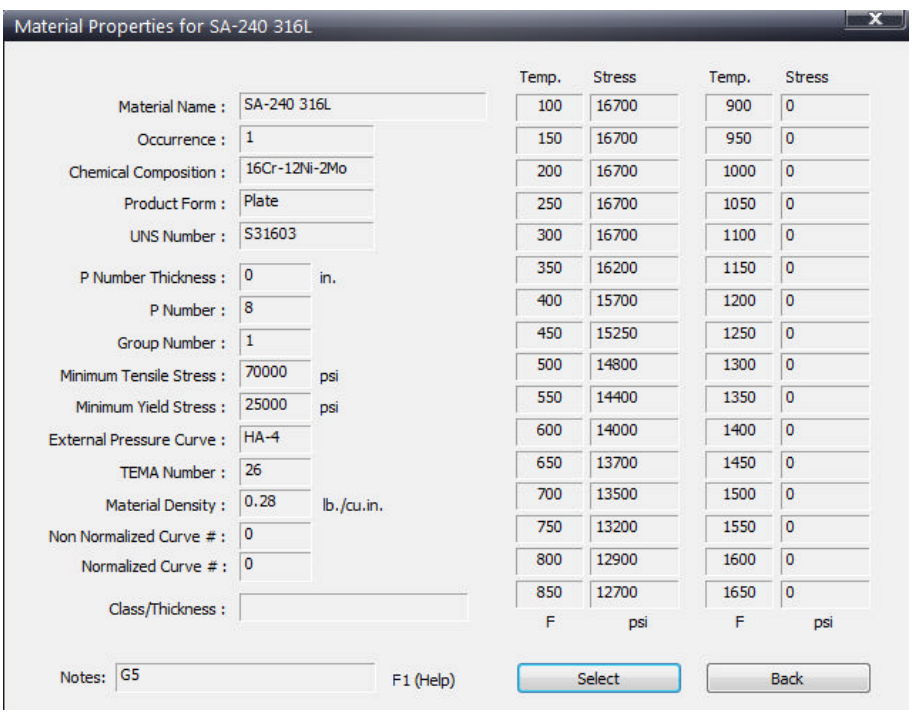
Change the material from *SA-516 70* to *SA 240 316L*.

Click on the material button near the top of the screen:
A new screen opens up:

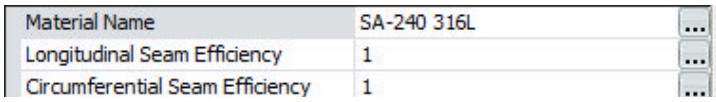




In the field called *Material Search String*, type in *SA240316L*. Don't worry about spaces and dashes. The material in the upper window will have a grey bar highlighting the first instance of the material. Click on the grey bar to get this screen shown below:

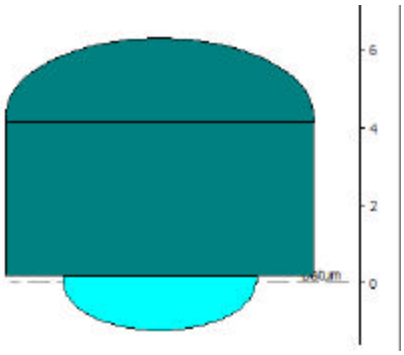


This screen gives you information about the material you have chosen. Simply click on the *Select* button, and the material for your bottom head will change as follows:



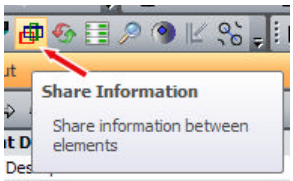
Updating the other components in the vessel

So far, we have entered the new data for the bottom head element. Also, if you look at the screen you will see the geometry difference of your bottom head to the rest of the elements:

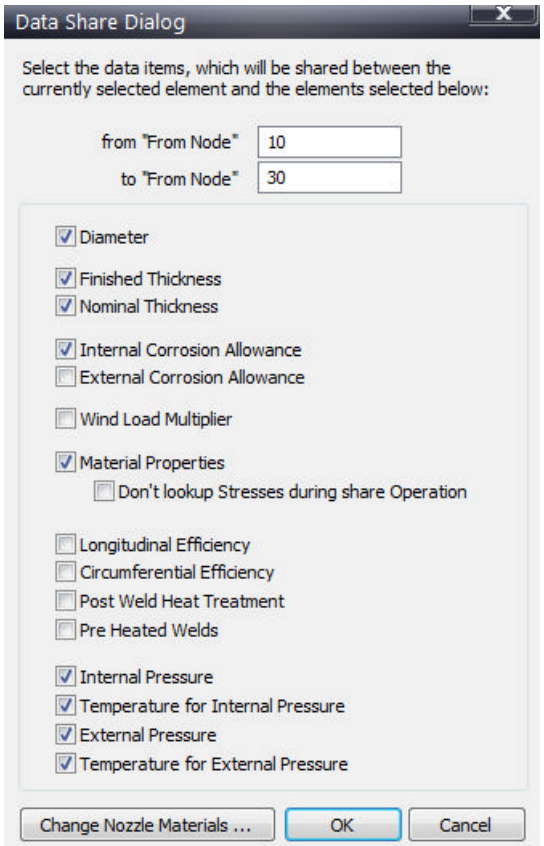


The vessel looks like this because the remaining components (main shell and top head) do not have their data fields updated. We are going to use a shortcut to copy the values for the bottom to the remaining components.

This is the *Share* button, located near the top of the screen:



Click on this button to get the following screen:



Notice the boxes that have been checked. The checks indicate that the items to which they refer will be shared with all the other upper components[3]. Make sure you have checked all and only the boxes shown as being checked on the left.

Now click the *Ok* button, and all the components should be updated to the new data

Your model now looks more realistic. You can see that at least the diameter of the bottom head has been shared with the main shell and top head.

[3] We discuss NODE numbers later.

The Status Bar at the bottom of the screen

So far, we have only entered the data relevant to our vessel, and produced the model on the screen

Select the *Main Shell* by clicking on it in either the 2D or 3D model. The main shell element will be highlighted.

Look at the bottom of the screen to the status bar:

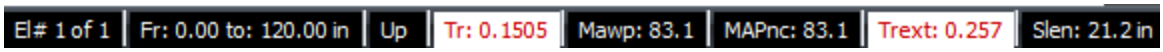
The status bar gives us a lot of information for the selected component. Let us consider the fields from left to right:

El# 2 of 3	:	Component number 2 of 3 components (from the bottom)
Fr: 0.17 To: 4.17 ft	:	Element spans from 0.17 ft to 4.17 ft from the datum line
Up:	:	Element Orientation
Tr: 0.2795	:	Computed thickness for the internal pressure (120 psi)
Mawp: 240.9	:	MAWP for this element (206.8 psi)
MAPnc: 275.6	:	Maximum allowable pressure new and cold (275.6 psi)
Trext: 0.264	:	Computed thickness for the external pressure
Slen: 35.0 ft	:	Maximum unsupported length for the external pressure

Let us now see what happens when there is a problem with an element.

Go to *Finished Thickness* field, and change the thickness from *0.5* inches to *0.2* inches.

Now look at the status bar at the bottom of the screen:

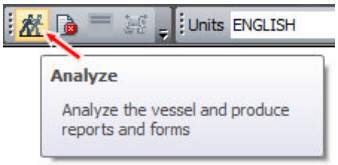


The required thickness for the internal and external pressures now appears in red, indicating that there is a problem. In our case, the problem is the thickness.

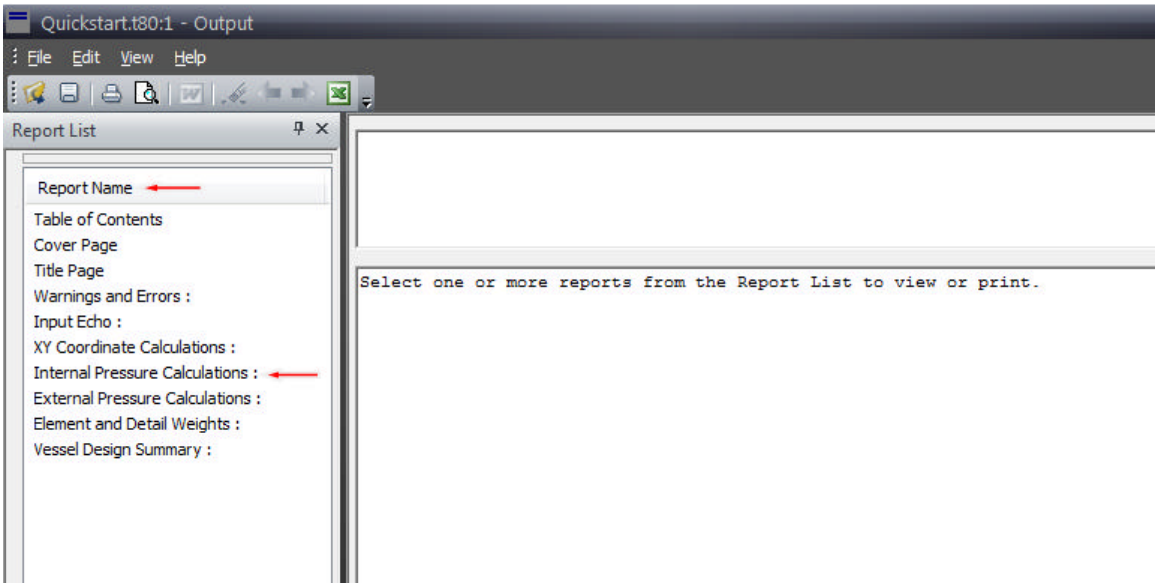
Change the thickness back to *0.5* inches, to render the status bar values in **black** lettering.

The Output Processor

Now we have built our simple model, we need to get the complete details of the calculations performed by **PV Elite**. At the top of the screen click on the *Analyze* button:



We now get the *Output Processor* as follows:



Your screen may look different, but it can easily be changed to look like the illustration above.

The left window has the heading *Report List*. In the right window, we are told *Select one or more reports from the Report List to view or print*. Click on *Internal Pressure Calculations* from the list. The right hand window now shows the results of the item you selected in the left hand window. The output window now looks as shown below:

```
PV Elite 2009 Licensee: COADE, INC.                    DEALR/EVAL COPY
FileName : Quickstart    ------------------------------------ Page <>
Internal Pressure Calculations :      Step:    3   3:05p  Nov 4,2008
```

Element Thickness, Pressure, Diameter and Allowable Stress :

From	To	Int. Press + Liq. Hd psig	Nominal Thickness in.	Total Corr Allowance in.	Element Diameter in.	Allowable Stress(SE) psi
BOTTOM HEA		120.000	0.50000	0.12500	60.0000	16700.0
MAIN SHELL		120.000	0.50000	0.062500	60.0000	16700.0
TOP HEAD		120.000	0.50000	0.062500	60.0000	16700.0

Element Required Thickness and MAWP :

From	To	Design Pressure psig	M.A.W.P. Corroded psig	M.A.P. New & Cold psig	Actual Thickness in.	Required Thickness in.
BOTTOM HEA		120.000	208.632	277.870	0.50000	0.34058
MAIN SHELL		120.000	240.932	275.578	0.50000	0.27945
TOP HEAD		120.000	243.354	277.870	0.50000	0.27808
Minimum			208.632	275.578		

MAWP: 208.632 psig, limited by: BOTTOM HEAD.

Internal Pressure Calculation Results :

ASME Code, Section VIII, Division 1, 2007 A-08

Elliptical Head From 10 To 20 SA-240 316L at 200 F

BOTTOM HEAD

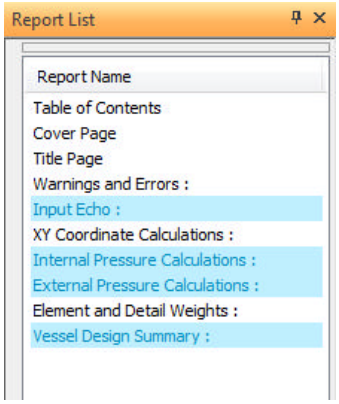
Thickness Due to Internal Pressure [tr]:
= (P*D*Kcor)/(2*S*E-0.2*P) Appendix 1-4(c)
= (120.000*60.1250*0.997)/(2*16700.00*1.00-0.2*120.000)
= 0.2156 + 0.1250 = 0.3406 in.

You can scroll up and down to look at the complete calculation for the internal pressure computations.

Generating and Printing the final pressure vessel report

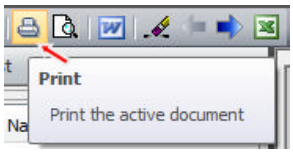
The illustration above shows the output only for the *Internal Pressure Calculations*. To select the items you want included in your final report do the following:

Hold down the *<Control>* key on your keyboard, click on the items you want included in the *Report List* (left) window. Your selected items will be highlighted with a blue background:



Once these items are selected, you are ready to generate the report.

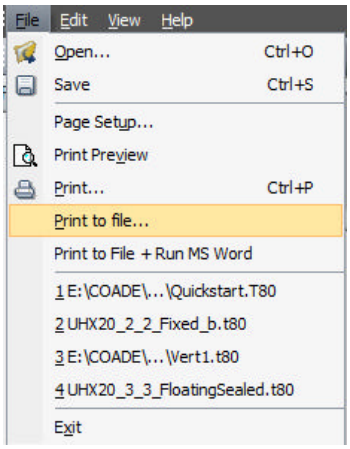
If you simply want to send the output to a printer, then you need to click on the *Print* button at the top of the menu bar:



This will print the report immediately with page numbers and headings on each page.

There are other options. For example, you may find it convenient to send the output to MS-Word™ if you have it installed on your computer. This feature allows you, the user to edit the results, and add your own notes or comments.

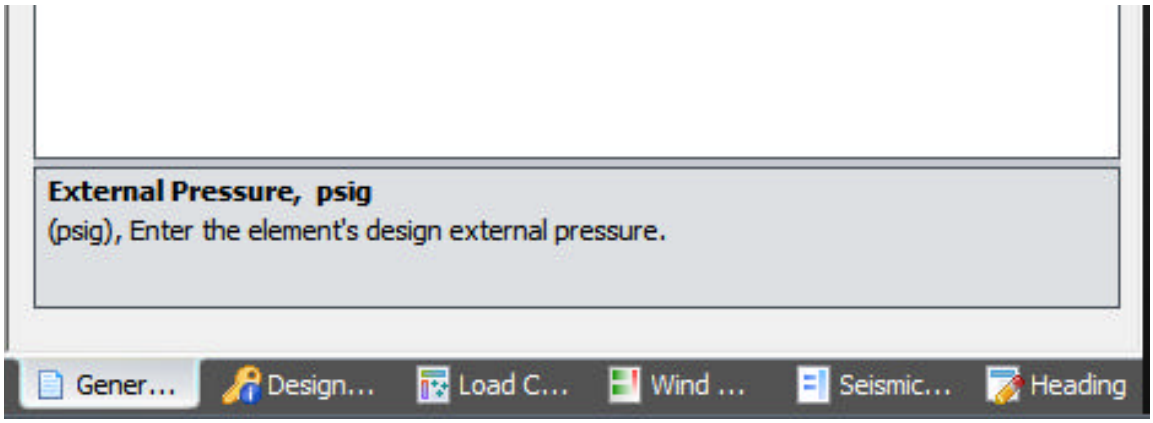
You can also send the output to a text file. From the file menu at the top of the screen, select *Print to file:*



You will be asked for a file name. This will be a text file with a filename extension of **.txt**. You will be able to access this file with any text editor such as Notepad™, or MS-Word™. When you send the output to a text file, all the colored text will be lost, and will simply be in black font color.

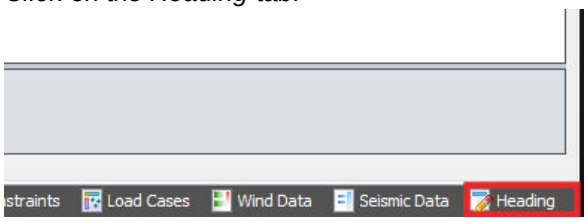
Data Input – Other information

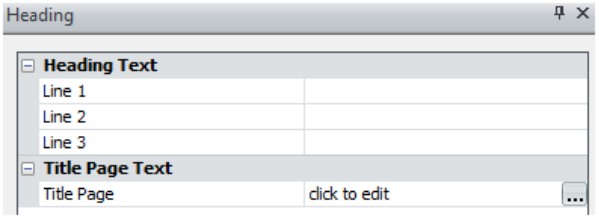
If you look at the bottom of the *Data Input* screen, you will see a series of tabs:



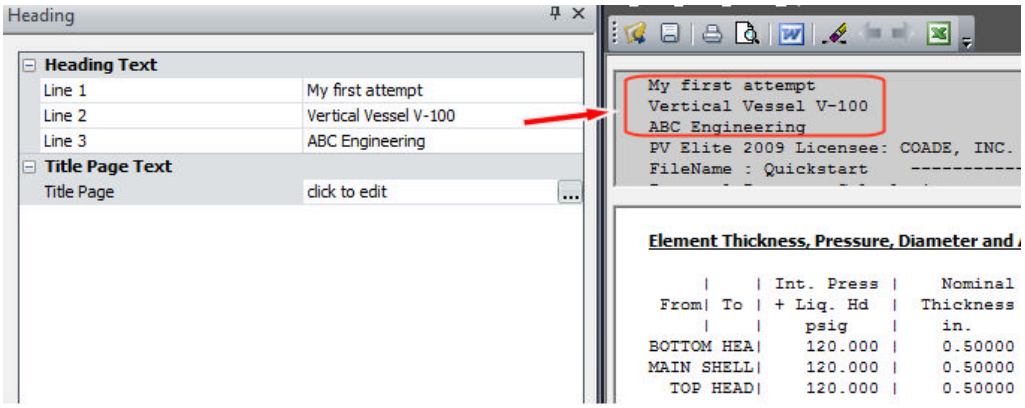
If you look carefully at these tabs, you will see that the left most tab is currently selected. The *General Input* tab refers to the basic dimensional, pressure, temperature and material selection for the currently selected element.

Click on the *Heading* tab:

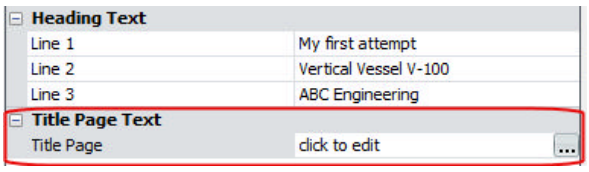




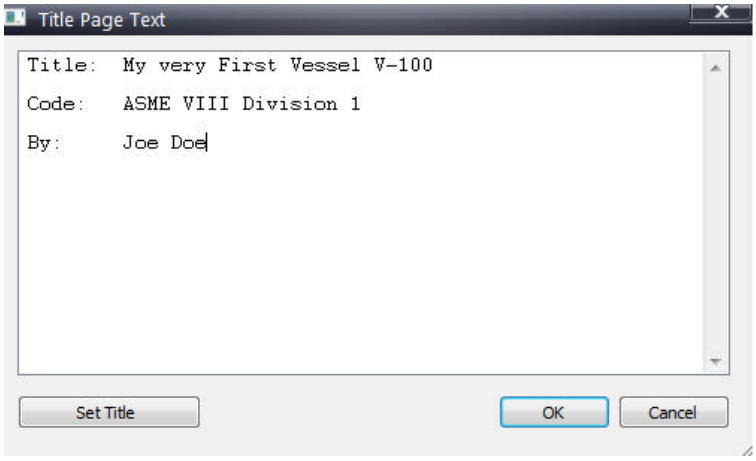
We have entered form information to demonstrate what can be typed in the first three fields (Lines 1, 2 and 3). In the final output, whether sent to the printer, or to MS-Word™, the three lines will appear at the top of each and every page of the report:



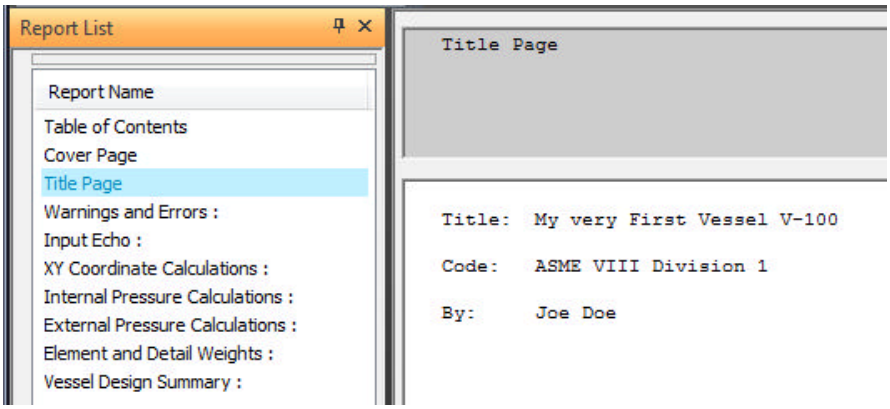
There is another field also while we have the *Heading* tab selected:



Click the little button to the right to have this screen open up:



Enter any information you care to, and it will be printed on the *Title Page* when sent to the printer or to a file. This will overwrite the information **PV Elite** would otherwise print as its default:



Design Constraints (Global Settings in PV Elite)

The *Design Constraints* tab allows us to set the default values for the entire vessel:

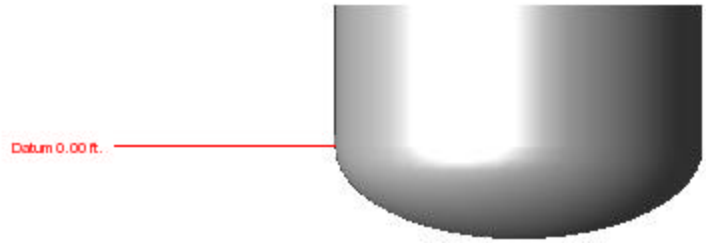
Design Data	
Design Internal Press, psig	100
Design External Press, psig	15
Design Internal Temp, F	200
Design External Temp, F	200

By entering the pressures and temperatures in these first four fields, **PV Elite** will use these values as the default values for the whole vessel. This saves time later.

<u>The datum field:</u>

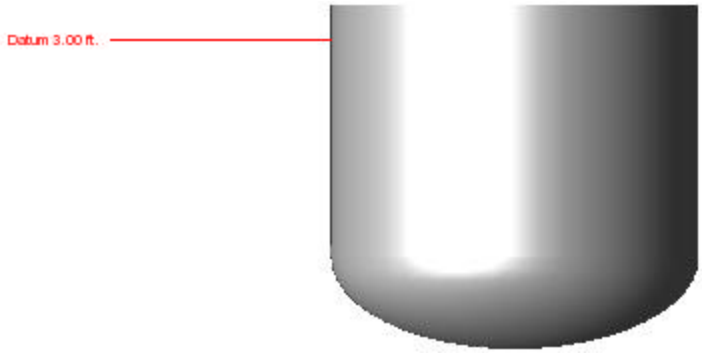
Design Data	
Design Internal Press, psig	100
Design External Press, psig	15
Design Internal Temp, F	200
Design External Temp, F	200
Datum Line Distance, ft.	0
Hydrotest Type	No Hydro

Look at your 3D model on the screen:



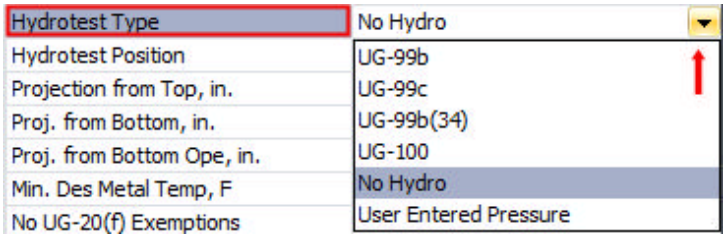
PV Elite has located the datum line at the tangent line of the bottom head. We can change the location of the datum line at any time to move it to a more convenient location to suit your purpose. Let us do this now to see what happens:

Set the datum line to 3 feet up from its current location. The model now looks like this:



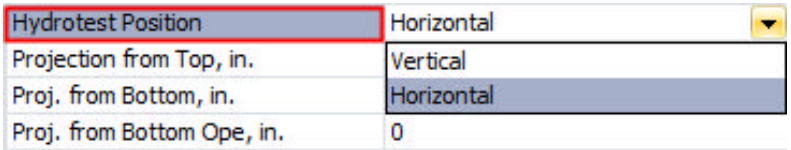
Once you have a skirt attached to the bottom of the vessel you may wish to move the datum line to the bottom of the skirt as it may be more convenient.

<u>Hydrotest Type and Position of the hydrotest:</u>



From the drop down box, you can tell **PV Elite** how it must calculate the hydrotest pressure.

The next box also allows you to let **PV Elite** know which position the vessel will be during the hydrotest:



Tall towers for example, are usually hydrotested in the horizontal position. **PV Elite** has to compute the hydrostatic pressure from the water in the vessel at hydrotest time. If the vessel is tested in the vertical positions, the pressure at the bottom of the vessel will be greater that if the vessel is tested in the horizontal position. Give careful consideration to the position that is appropriate to your situation.

Miscellaneous Weight %:

Many designers like to include extra weight to account for vessel attachments and internals not otherwise included in the models. The total weight of the vessel is multiplied by 1.0 plus this percent (i.e. 1.03, 1.05).

Design Code:

PV Elite allows the user to perform vessel calculations in several pressure vessel codes. The field to change the design code is located at the *Units/Code* toolbar:

The following design codes are supported by **PV Elite**:

- ASME Section VIII, Division 1
- ASME Section VIII, Division 2
- British Code PD 5500
- European Code EN 13445

Once a code is selected, the user will have to re-select the materials, as each code has its own design stress tables.

Is this a Heat Exchanger:

Note: This check box is optional.

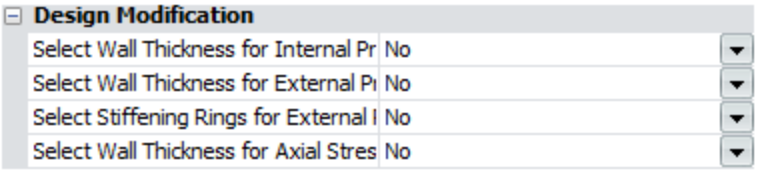
If the Dimensional Solutions 3D file interface button is checked, **PV Elite** will write out a text file that contains the geometry and loading information for this particular vessel design. If this box is checked, the program will simply write this data out to the Jobname.ini file created in the current working directory. More information about the Dimensional Solutions product line can be found on their website at the following web site: www.Dimsoln.com

To completely define an exchanger it is necessary to enter in the required information regarding the tubes, tubesheets and the floating head (if any). With the exchanger data, **PV Elite** can then compute the weights and required thicknesses of the exchanger components.

ASME Steel Stack:

PV Elite will analyze steel stacks in accordance with ASME STS-2000. A further discussion can be found in the PV Elite Help Facility.

Design Modification:

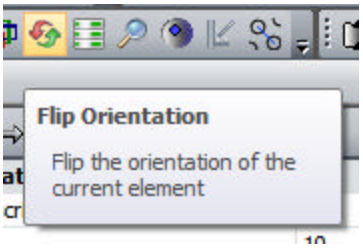


If any of the items is set to yes, **PV Elite** will correct the item should it fail in the analysis. For example, if 'Select Wall Thickness for Internal Pressure' be set to 'Yes', **PV Elite** will automatically increase the thickness of a component should it not be thick enough.

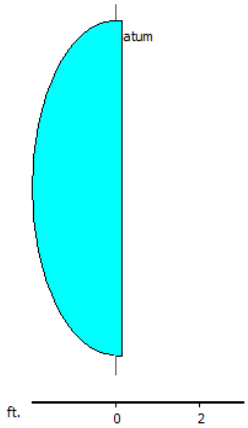
Building a Horizontal Vessel – Start a new vessel

As discussed above, the vessel is built either from bottom to top, or from left to right.

On the input data screen, start with the first head. Click on the *Ellipse Head* button. **PV Elite** assumes you are building a vertical vessel, and the element is in the vertical position. We need to flip this head to enable us to build a horizontal vessel. Click on the *Flip* button:



Once you click on this button, the head will then turn clockwise through 90 degrees to look like this:



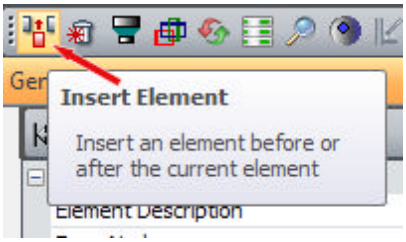
Now add the cylindrical shell and right hand head (as shown in the beginning of this document), until the vessel looks like this:

Inserting a Component

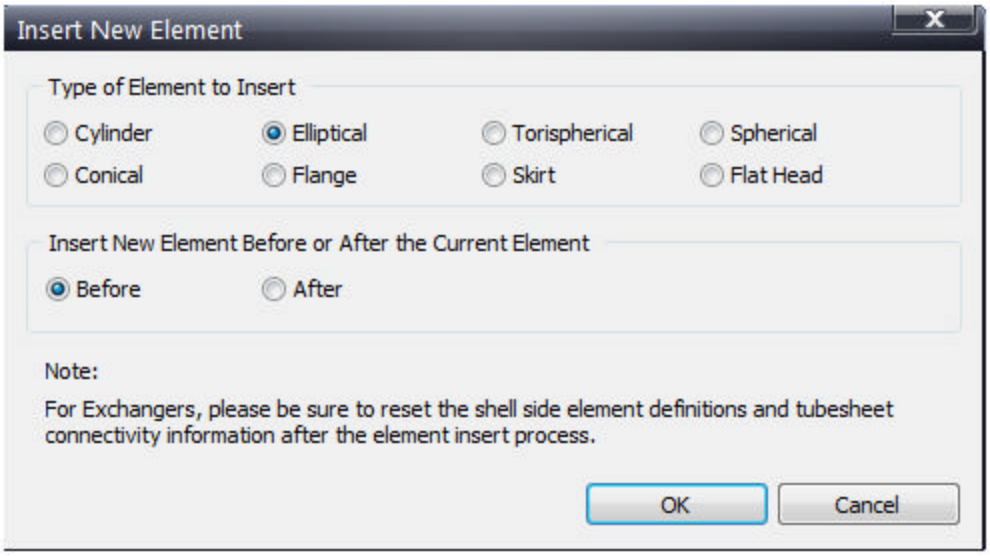
Often a component is missing, and has to be inserted into the model.

These are the steps you must take:

- Select the item on your 3D or 2D model (click on the component in the 2D model). The selected component will turn light green on the 2D model.
- Near the top of the screen click on the *Insert* button:



- The *Insert New Element* screen opens up:

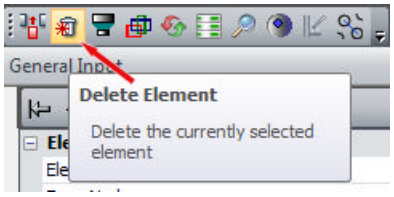


- Select the type of element you wish to insert
- Determine if you want to insert the element before or after the current element.

Deleting a Component

Follow these steps:

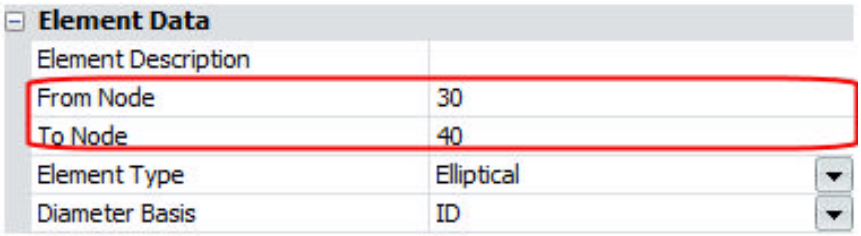
- Select the item you wish to delete (make sure there are no nozzles in this element).
- Click the delete button:



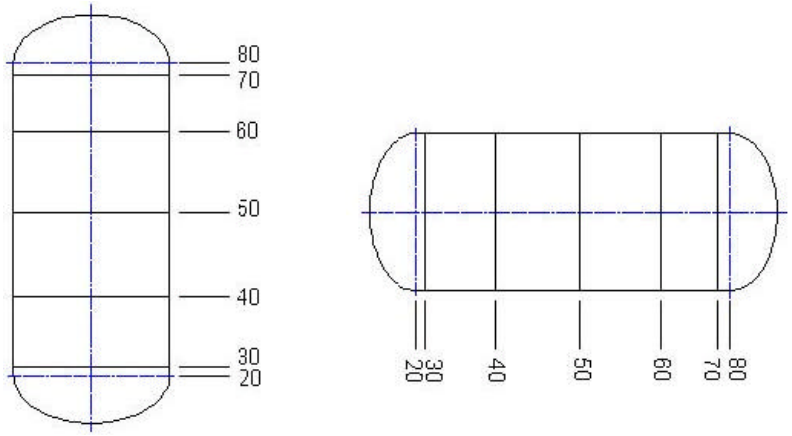
- Another screen will ask if you are sure to delete the current element.

Node Numbers

All elements that make up a vessel exist between nodes. On the top part of the *General Input* tab you will see two fields that mention *From Node* and *To Node*:



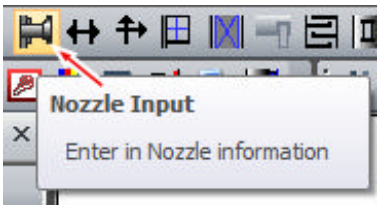
Every main element (not attachments such as nozzles) has two nodes. Consider these two vessels shown below:



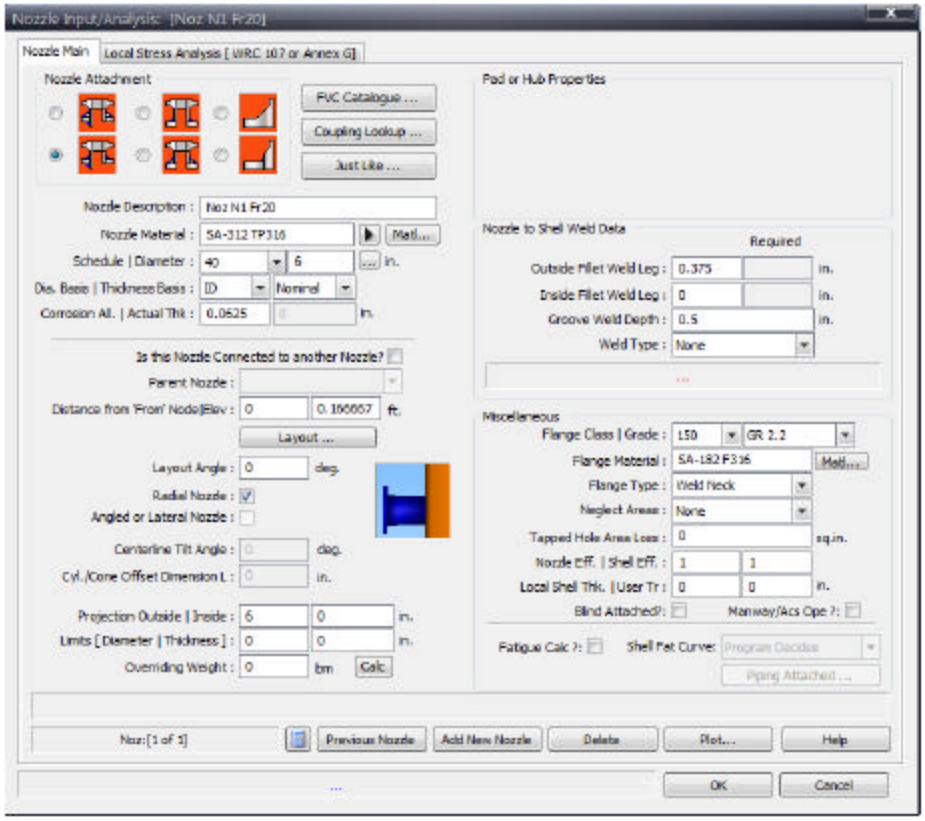
Notice that every element from the bottom to the top or from the left to the right exists between two node numbers. There is a *From Node*, and there is a *To Node*. In the case of the bottom (or left) head the nodes are situated at the tangent line and the weld line.

Adding a Nozzle to the Model

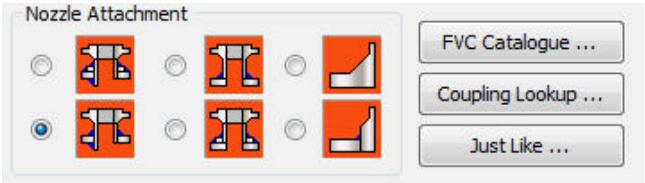
Before we can add a nozzle, **PV Elite** needs to know into *which component* we are going to add the nozzle. Tell **PV Elite** by selecting the component. Click on the shell in the 2D model, which will become highlighted. Click on the nozzle icon on the toolbar:



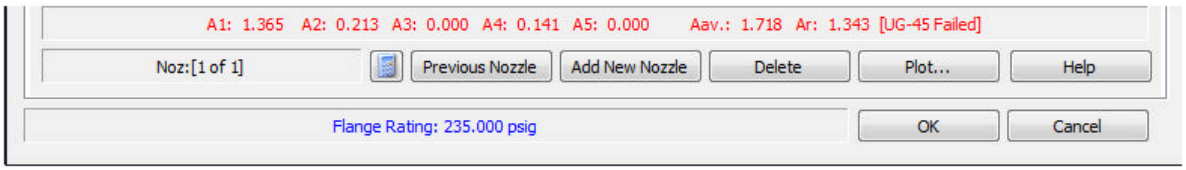
The nozzle dialog screen will appear and you will be able to select the type of nozzle you want for your element:



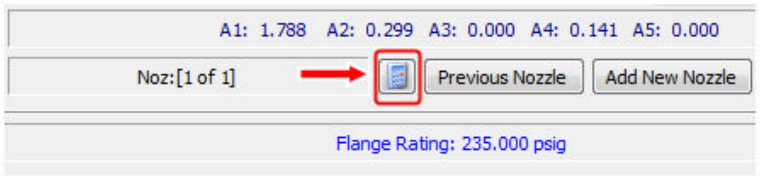
Different nozzle configurations can be considered. The choices are basically nozzles without pads, nozzles with pads, protruding nozzles without pads and protruding nozzles with pads. There are also special forged nozzles available for analysis:



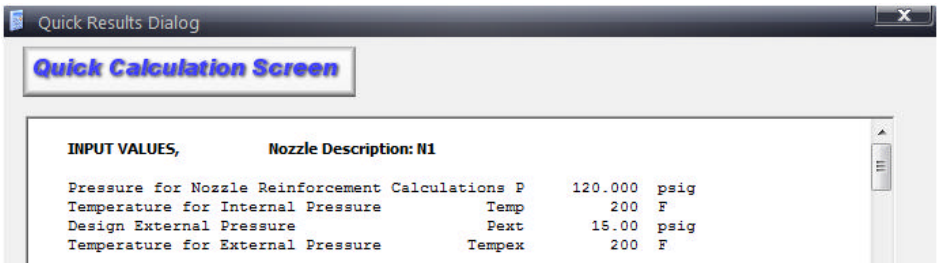
Select the default nozzle attachment *Insert Nozzle*. Click on the nozzle description field and change the description name of your nozzle to *N1*. Select the next fields or click the *<Enter>* key on your keyboard past the *Diameter* field to run your nozzle calculation. If your desired inputs fail the design code, you will notice at the bottom of the dialog screen the red warnings on the status bar, same as in the **PV Elite** main screen:



Also at the bottom of the *Nozzle Analysis* dialog box, you will see a calculator button below the status bar:

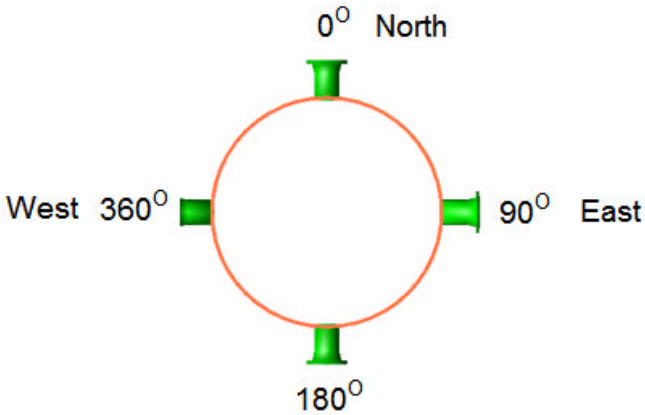


Press on the calculator button and the *Quick Results Dialog* screen will open up. This screen will give you a quick calculation report of your current nozzle:

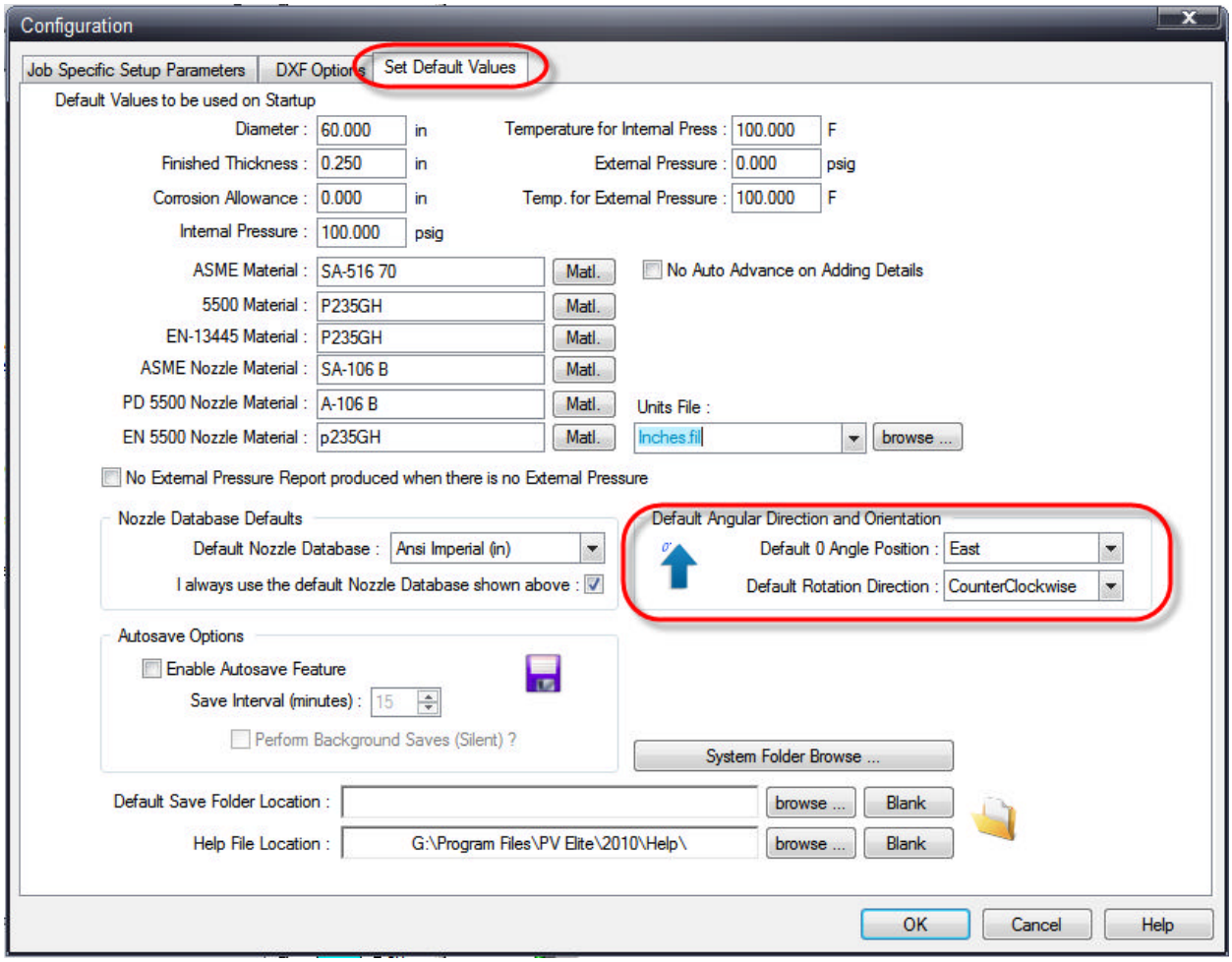


Nozzle Orientation around the Vessel

This section concerns the orientation of the nozzle around the vessel. Let us have a look at a nozzle orientation to see what is meant:



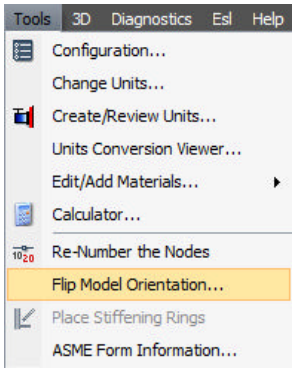
From the picture, the 0° position is at the top (North) and the nozzle at position 90° is to the right. It follows that the angles of the nozzle increase in the clockwise direction. However the zero position can occupy the North, West or East position as desired by the user. The angle can increase in the clockwise or the counter-clockwise direction. This innovation was introduced in the 2010 version of ***PV Elite***. The controls can be found on the Configuration dialogue screen. To get the Configuration dialogue screen go to: Tools -> Configuration on the menu bar, and click on the Set Default Values tab:



By pressing the little down arrows to the right of the list boxes, you can choose the orientation that best suits your purpose.

Flip Model Orientation

If a new model is built in the vertical position and it needs to change orientation to the horizontal position for different analysis purposes; **PV Elite** is able to flip the model with a single button found in the Menu bar under *Tools* then *Flip Model Orientation*:



PV Elite Help System

PV Elite has an extensive help system. Place your cursor on a particular field that you are interested in. Press the *<F1>* key on your keyboard, and an instant help window is available. The complete User Manual can also be accessed by clicking on *Help* in the menu bar and selecting *View Documentation*.