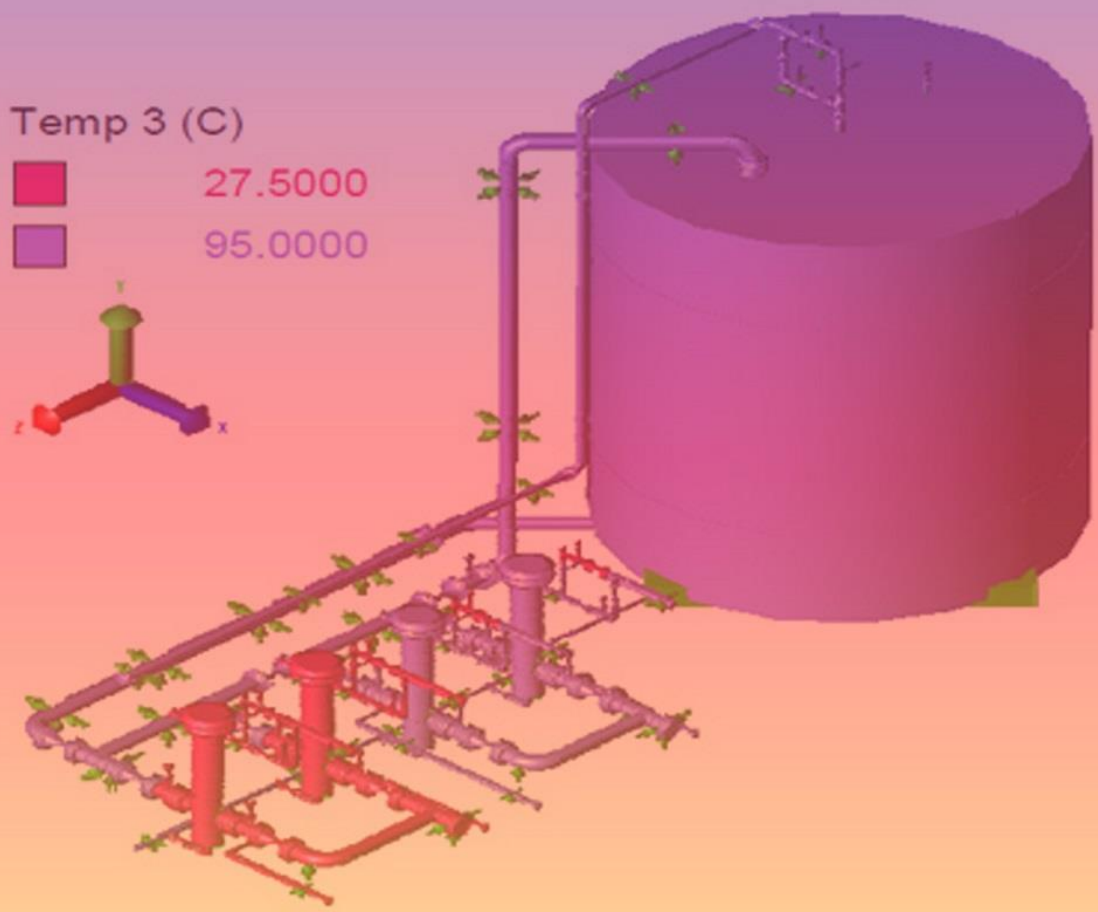
# Procedure of Piping Stress Analysis

3D-LABS



# 1 INTRODUCTION

Pipe Stress Analysis, is analysing the hot and large piping systems to determine, the stresses in the piping systems not exceeds the allowable limits set out in the project design code, Piping loads on equipment nozzles should be calculated and compared with vendor allowable nozzle loads, displacement to evaluate the need for guides to limit the pipe in lateral movement and/or stops to limit pipe movement on supports, Piping loads on support are used in the design of supports.

# 2 PIPING STRESS ANALYSIS

## 2.1 OBJECTIVES OF PIPE STRESS ANALYSIS

**Structural Integrity:**

- Design adequacy for the pressure of the carrying fluid.
- Failure against various loading in the life cycle. Limiting stresses below code allowable.

**Operational Integrity:**

- Limiting nozzle loads of the connected equipment within allowable values.
- Avoiding leakage at joints.
- Limiting sagging & displacement within allowable values.

**Optimal Design:**

- Avoiding excessive flexibility and also high loads on supporting structures. Aim towards an optimal design for both piping and structure.

## 2.2 GOVERNING CODES AND STANDARDS

Codes and Standards specify minimum requirements for safe design and construction (i. e. provides material, design, fabrication, installation and inspection requirements.)

Following are the codes and standards used for Refinery Piping:

1. ASME B31.3: Process piping Code
2. ASME B31.1: Power Piping Code
3. Centrifugal Pumps: API 610
4. Positive Displacement Pumps: API 676
5. Centrifugal Compressors: API 617
6. Reciprocating Compressors: API 618

7. Steam Turbines: NEMA SM23/ API 612
8. Air Cooled Heat Exchanger: API 661
9. Fired Heaters: API 560
10. Flat Bottom Welded Storage Tanks: API 650
11. Heat Exchangers: TEMA/ Vendor Specific.
12. Vessel/Column: Vendor Specific

## 2.3 STRESSES IN A PIPING SYSTEM

Sources for generation of stress in a Piping System:

- Weight
- Internal/External Pressure
- Temperature change
- Occasional Loads due to wind, seismic disturbances, PSV discharge etc.
- Forces due to Vibration.

Sustained Stresses are the stresses generated by sustained loads. (E.g. Pressure, Weight). These loads are present continuously throughout plant life.

Resistive force arising out of sustained stresses balance the external forces keeping the system in equilibrium. Exceeding sustain allowable stress value causes catastrophic failure of the system.

As per **ASME B 31.3, (clause 302.3.5)** " The sum of the longitudinal stresses, SL, in any component in a piping system, due to sustained loads such as pressure and weight, shall not exceed the product Sh x W ". Where, Sh=Basic allowable stress at maximum metal temperature expected during the displacement cycle and W=weld joint strength reduction factor.

Pressure Stresses are taken care of by calculating and selecting proper pipe thickness. The pressure thickness (t) of a straight pipe can be obtained as per **ASME B31.3 from the equation (Clause 304.1.2) mentioned in Fig.1a:**

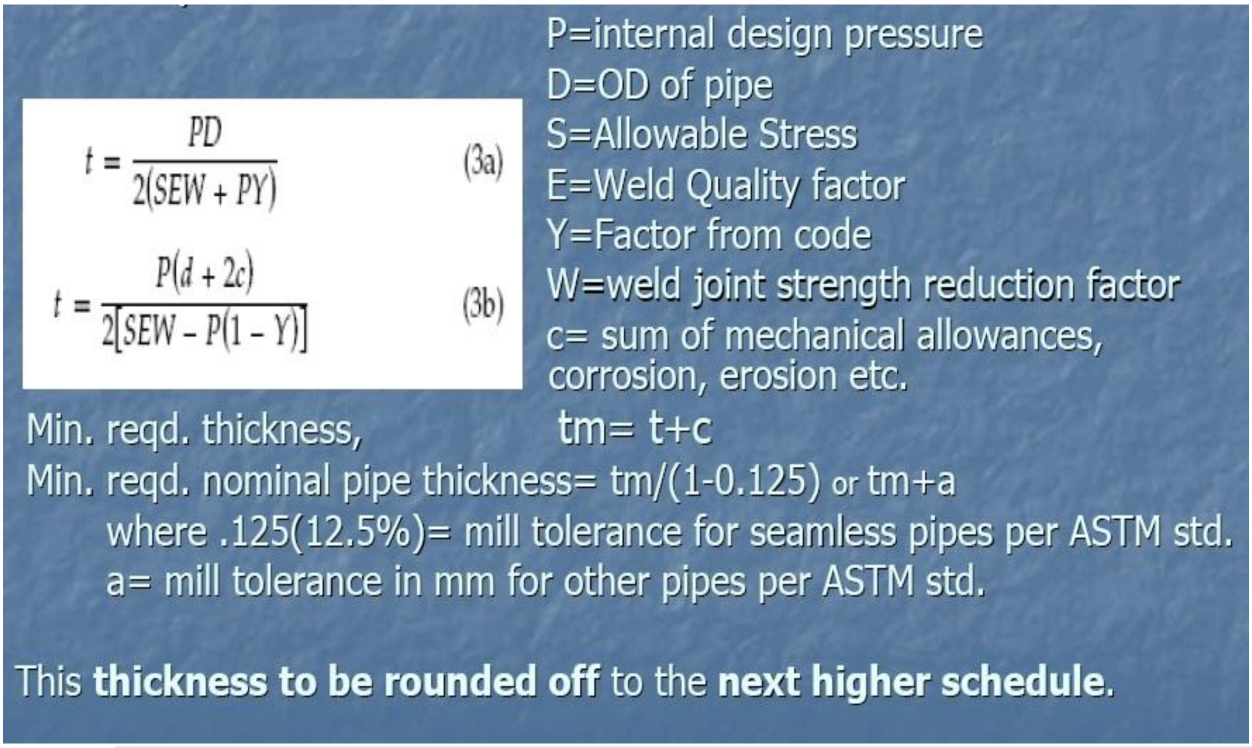



**Fig. 1a: Equation for Thickness Calculation for Straight Pipe based on ASME B 31.3**

Change in length of a pipe of length L due to temp change (ΔT) is given by ΔL=L α ΔT Here, α =Co efficient of thermal expansion = change in length of unit length element due to unit change in temp.

Two "α" values in Code (Table C1 and C3 in ASME B31.3 Appendix C):

Table C1 denotes total linear thermal expansion between 700 F to Indicated temp (unit=in/100ft).

Table C3 denotes mean coefficient of linear thermal expansion between 700 F to indicated temp (μin/in/0F).

Expansion stresses are generated when the free thermal growth due to temperature change is restricted. These are self-limiting or relenting.

SIF(Stress Intensification Factor): This is the ratio of the maximum stress intensity to the nominal Stress. SIF factors for different components can be obtained from Appendix D of ASME B31.3.

Displacement Stress Range due to thermal expansion is calculated based on equation SE = ( Sb^2+4 St^2)^0.5 per equation 17 from **ASME B31.3(clause 319.4.4).**

This SE value shall not exceed SA value where SA= Allowable Displacement Stress Range.

As per **ASME code B 31.3 (Clause 302.3.5)** the allowable displacement stress range (SA) can be given by the equation (Fig.2a):

$$S_A = f(1.25S_c + 0.25S_h) \qquad (1a)$$

**Fig.2a: Equation for Displacement Stress Range Allowable**

Here, f= Stress range reduction factor and Sc=basic allowable stress at minimum metal temp

When Sh > SL, the allowable stress range is calculated by the following equation (Fig. 3a): SL=Longitudinal Stress due to sustained loads.

$$S_A = f[1.25(S_c + S_h) - S_L] \qquad (1b)$$

**Fig.3a: Equation for Liberal Displacement Stress Range Allowable**

Occasional Stresses are generated by the occasional loads such as Wind, seismic, PSV discharge etc.

This loads act in a piping system for very small period of time, usually less than 10% of total working period.

As per ASME B31.3 clause 302.3.6 “The sum of the longitudinal stresses, SL, due to sustained loads, such as pressure and weight, and of the stresses produced by occasional loads, such as wind or earthquake should be ≤ 1.33 times the basic allowable stress, Sh”

Code does not explicitly explain the stresses generated due to vibration. The vibration problems are solved by engineering judgement and experience.

## 2.4 REDUCING PIPING STRESSES

- Supports for Weight
- Flexibility for thermal loading Eg. Expansion Loops.

**Flexibility check (as per clause 319.4.1, ASME B 31.3):** Refer Fig. 4a

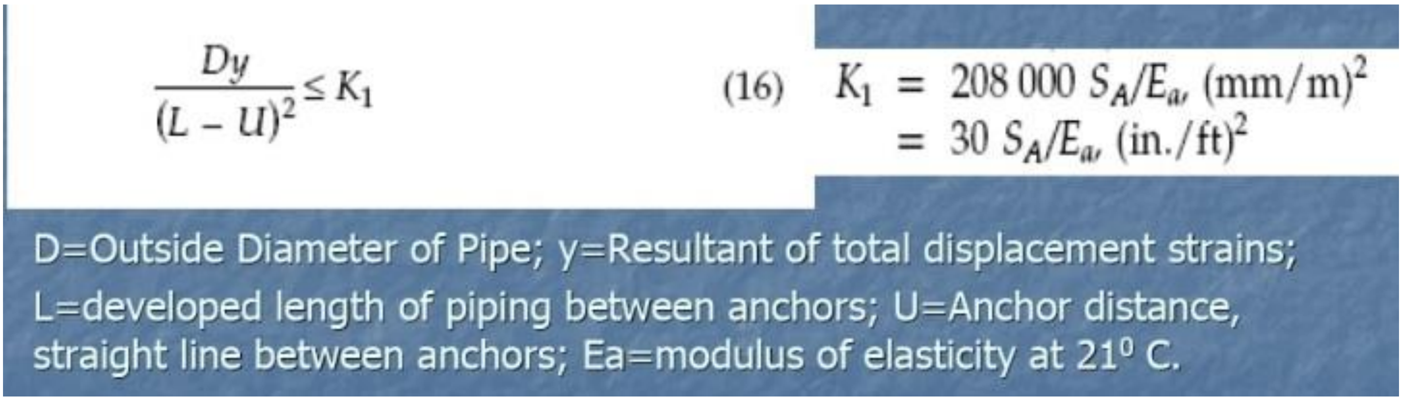



**Fig.4a: Flexibility Check Equation for Simple Systems**

## 2.5 BASIC ALLOWABLE STRESS

Minimum of (As per ASME B 31.3)

- 1/3rd of Ultimate Tensile Strength (UTS) of Material at operating temperature.
- 1/3rd of UTS of material at room temperature.
- 2/3rd of Yield Tensile Strength (YTS) of material at operating temperature.
- 2/3rd of YTS of material at room temp.
- 100% of average stress for a creep rate of 0.01% per 1000 hr.
- For structural grade materials basic allowable stress=0.92 times the lowest value obtained from 1 through 5 above.

## 2.6 LOADS ON A PIPING SYSTEM

There are two types of loads which acts on a piping system: Static loads and Dynamic Loads

Static loads are those loads which acts very slowly and the system gets enough time to react against it. Examples of static loads are shown in Fig.5a

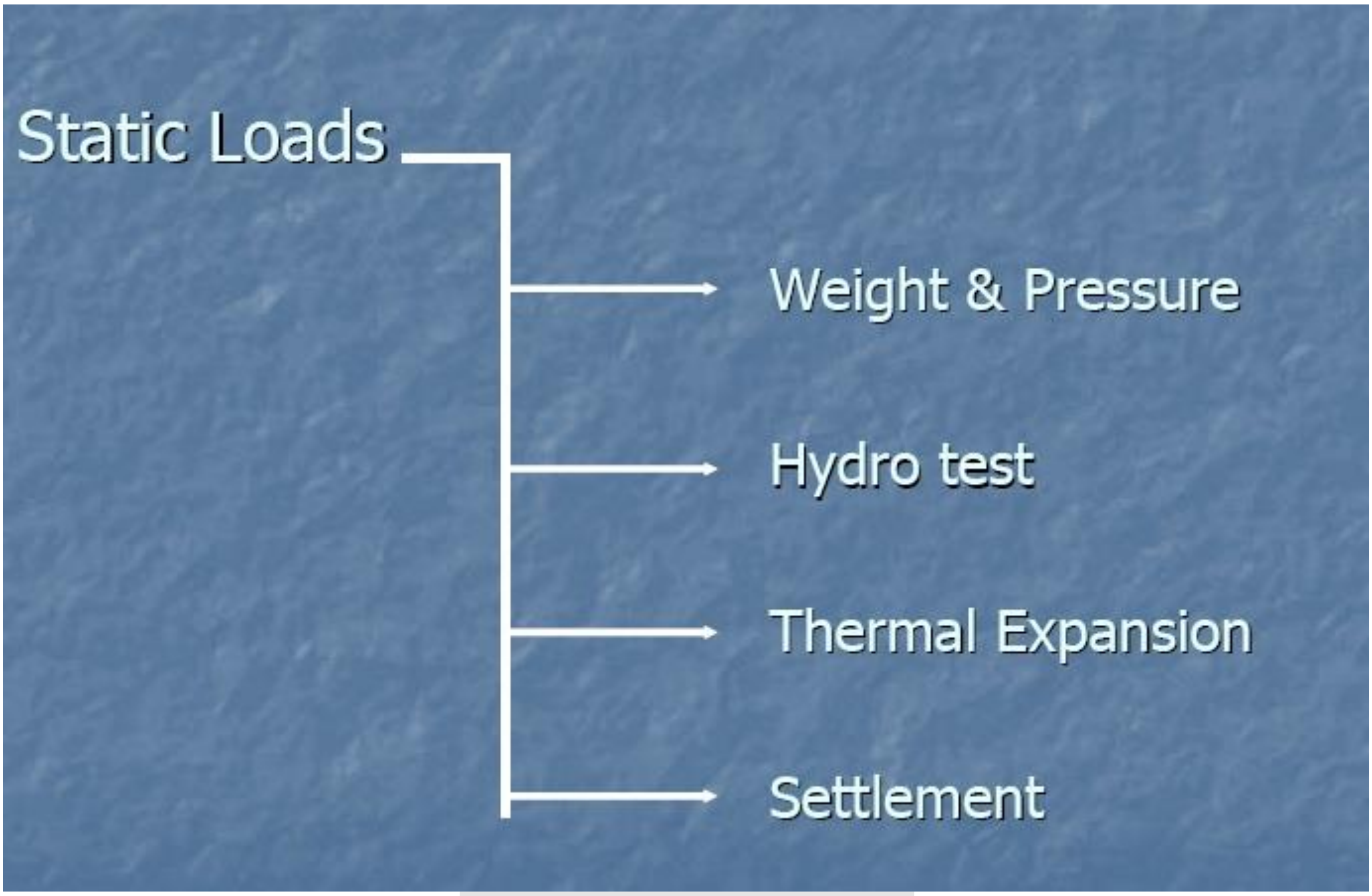


**Fig.5a: Examples of Static Loads**

On the other hand dynamic loads acts so quickly that the system does not get enough time to react against it. Examples of dynamic loads are shown in Fig.6a

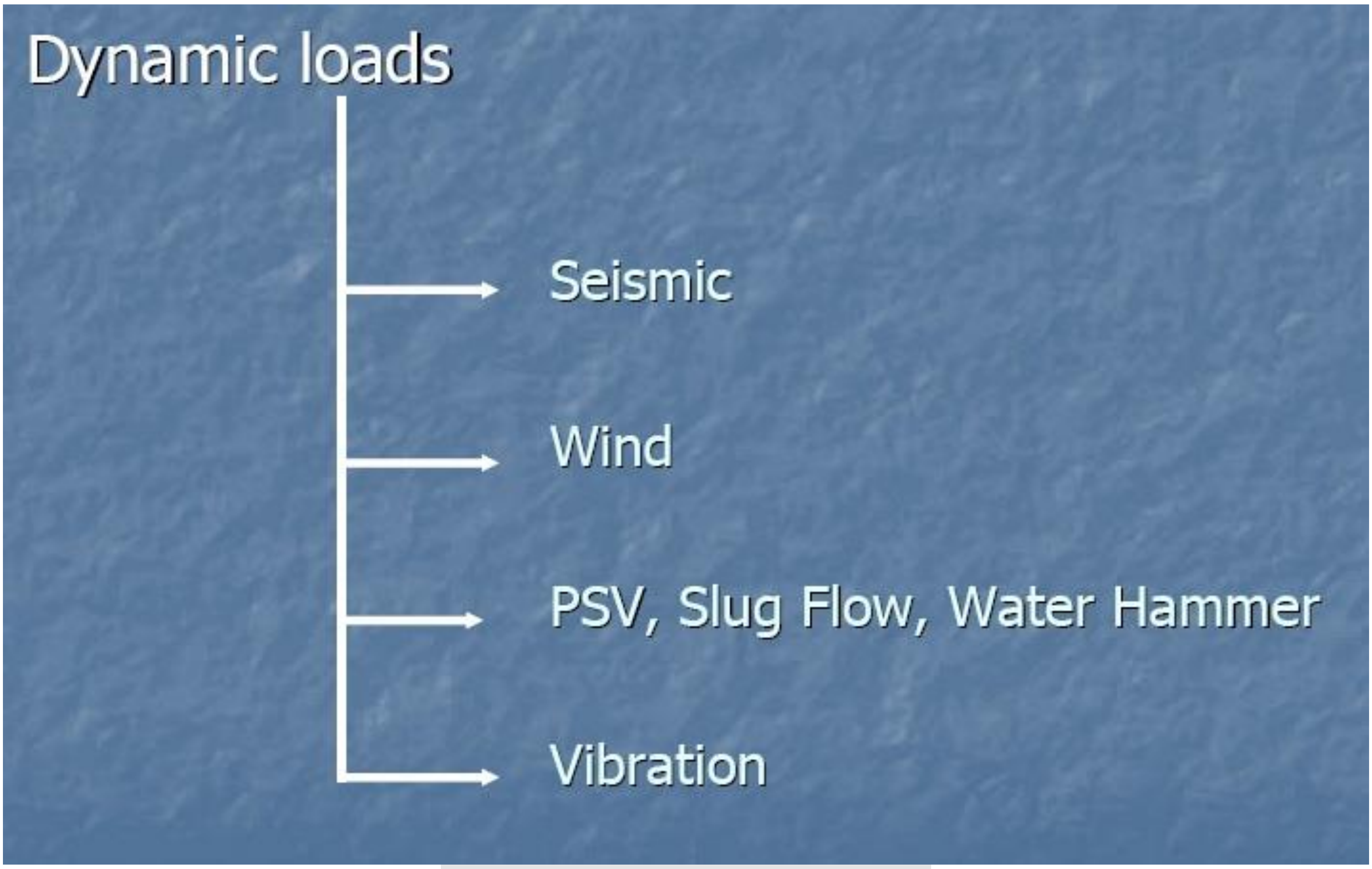



**Fig.6a: Examples of Dynamic Loads**

## 2.7 WORKFLOW DIAGRAM

The interaction of Piping Stress team with other disciplines in any organization are shown in Fig. 7a:

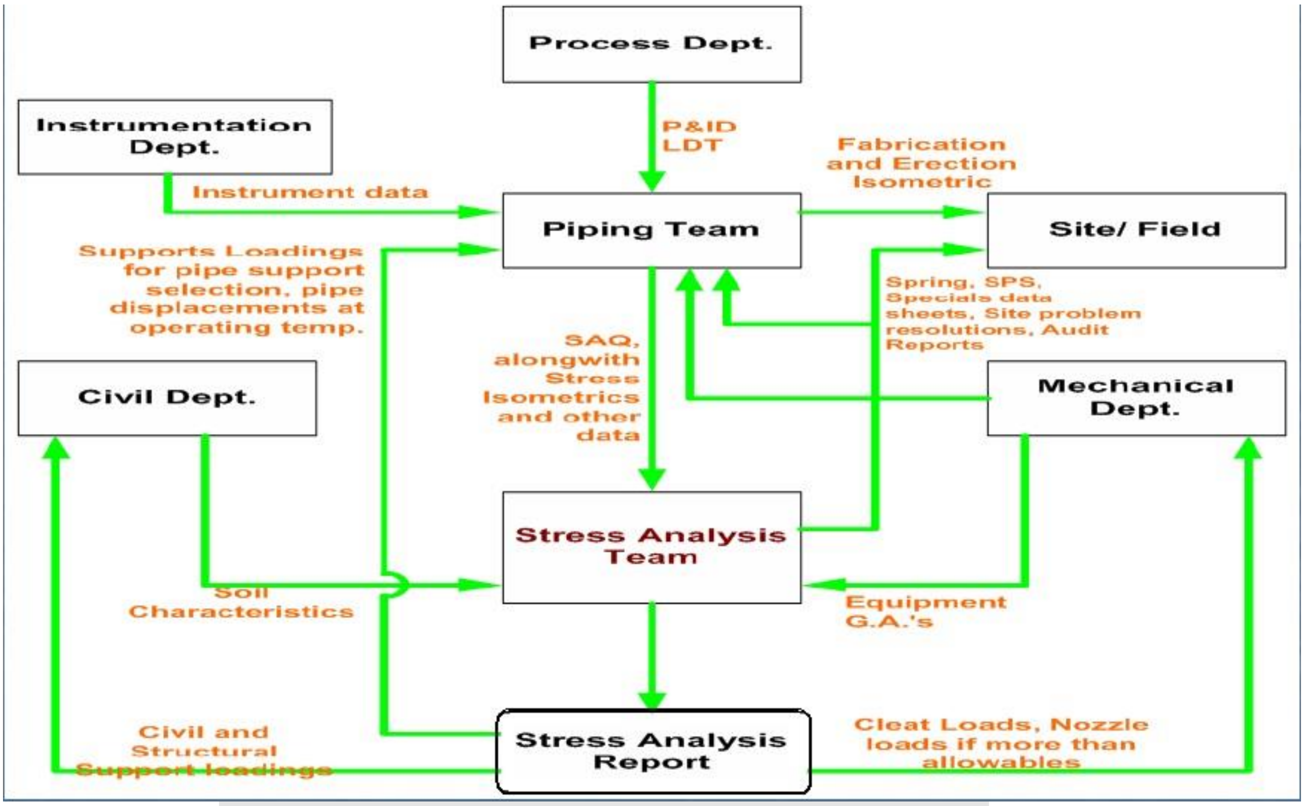



Fig.7a: Inter Departmental Interaction with Stress Team

## 2.8 CRITICAL LINES & ANALYSIS METHOD

- Highly Critical Lines (Steam turbine, Compressor connected pipelines) ⇨ By Computer Analysis
- Moderately Critical Lines (AFC connected lines) ⇨ By Computer Analysis
- Low critical Lines ⇨ Visual/Simple Manual Calculation/Computer analysis
- Non Critical Lines ⇨ Visual Inspection

# 3 STRESS ANALYSIS USING CAESAR-II

Two types of stress analysis in CAESAR-II are,

- Static analysis
- Dynamic analysis

In this article we are going to discuss about the static piping stress analysis.

## 3.1 INPUTS

- Stress Isometric from Layout team
- LDT And P&ID from Process
- Equipment GA and Other detailed drawings from Mechanical
- Process flow diagram/datasheet if required from process
- Piping Material Specification
- Valve GA and Datasheet from Instrumentation
- Soil Characteristics from civil for underground analysis
- Nozzle load limiting Standards
- Plot Plan for finding HPP elevation and equipment orientation.
- Governing Code

## 3.2 HOW TO MODEL IN CAESAR-II

**Step-1**

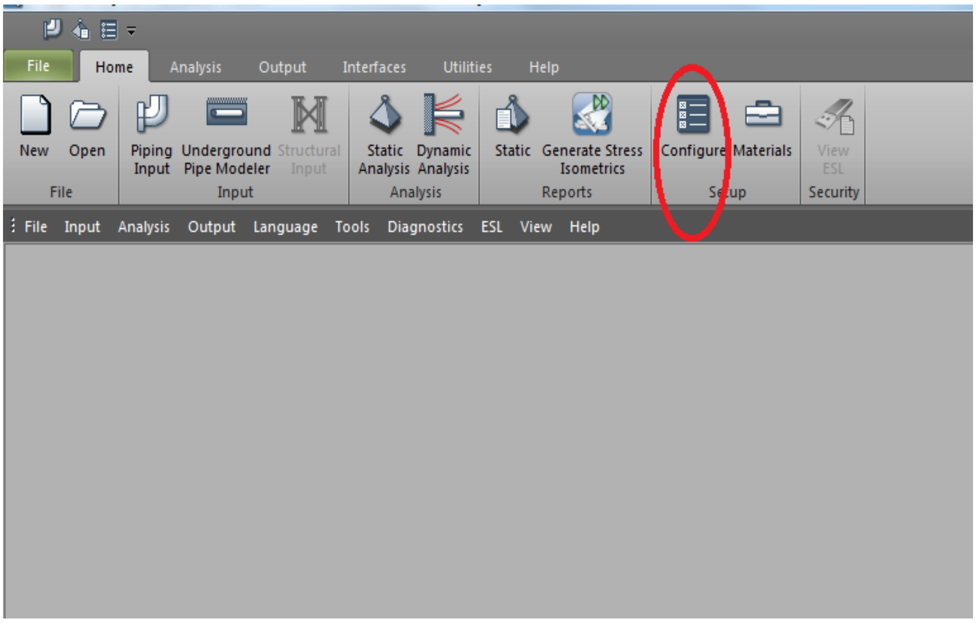



*Figure 1 HOME PAGE*

Open the CAESAR-II software

Configuration (refer fig2)

Change the unit and other parameters as per inputs

Save & exit

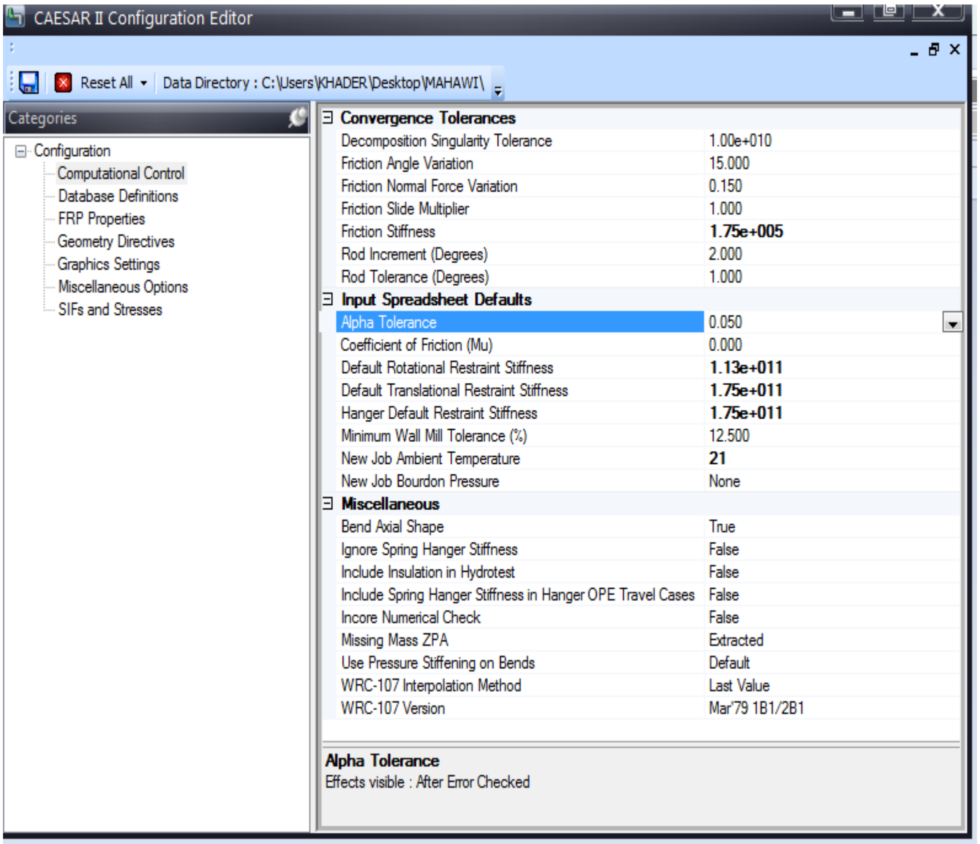
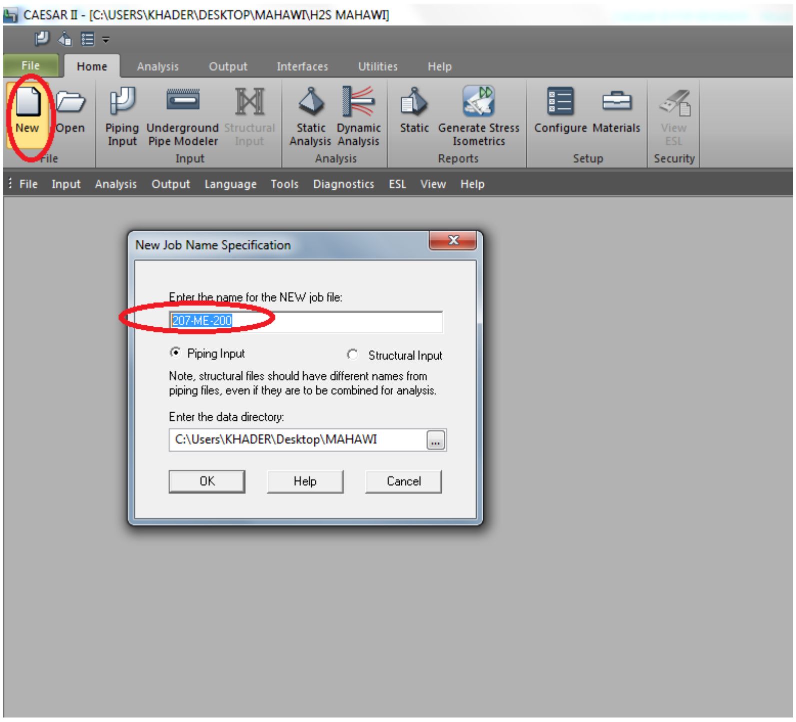



*Figure 2 CONFIGURATION EDITOR*

## <u>Step-2</u>




New

↓

Enter job name

↓

File location

↓

Ok

*Figure 3 HOME PAGE WITH NEW JOB FILE DIALOGUE BOX*

## <u>Step-3</u>

The highlighted box (refer figure 4) in the classic piping input tab shows the node number tool, here we can manage the node number of a pipe or fittings or etc. Node numbers must be numeric, ranging from 1 to 32000.We can also provide the name for the particular component.

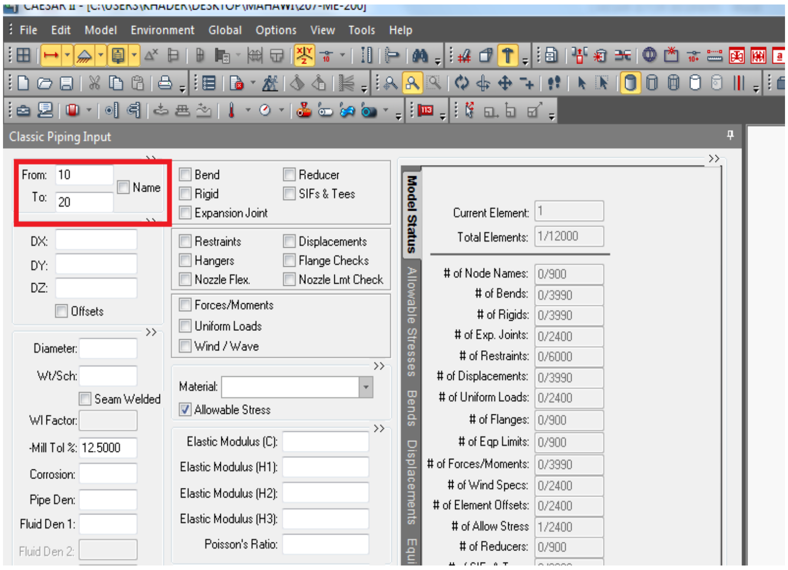


*Figure 4 ASSIGN NODE NUMBER (classic piping input)*

## Step-4

The highlighted box (refer figure 5) in the classic piping input tab shows the length (component position in DX, DY, DZ coordinate) of a component.

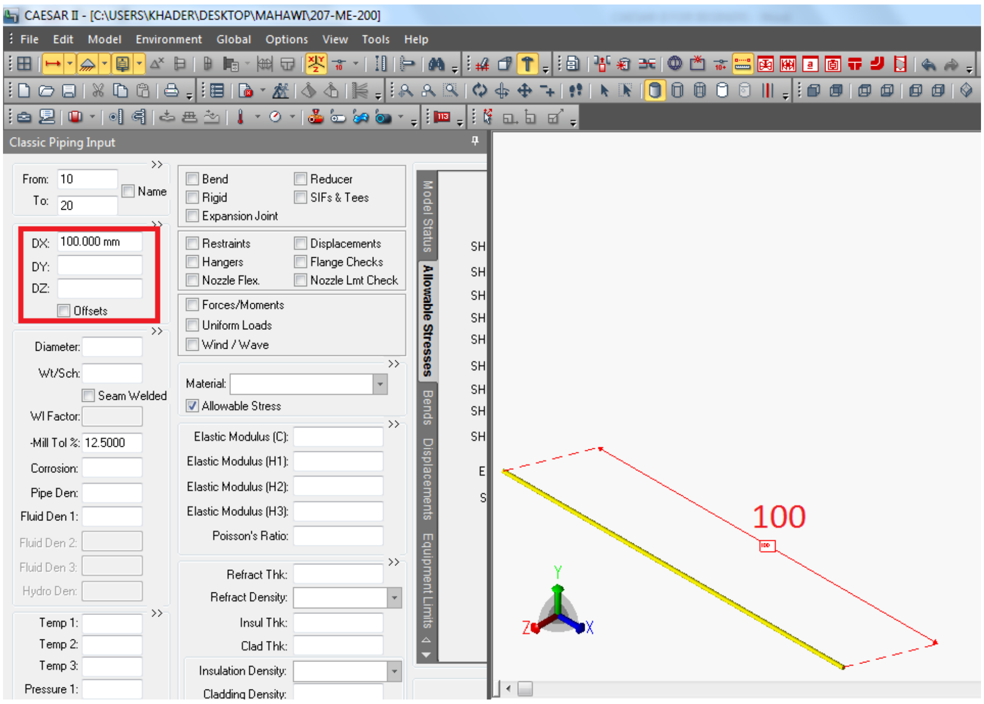


*Figure 5 length of the pipe (classic piping input)*

This is the place where we have to provide the length of a pipe or pipe fittings or valve F/F length.

The fig 5, shows a pipe goes 100mm towards the positive (+X) direction.

Likewise we can model a skid or pipeline. CAESAR II accepts [compound length]-[length]-[fraction] formats (such as feet - inch - fraction or meter - decimal - centimeters) as valid input values in most cells. You can use simple forms of addition, multiplication, and division as well as exponential format.

## Step-5

The highlighted box (refer figure 5) in the classic piping input tab shows the pipe parameters. Here we have to provide the pipe size (eg. 2” NPS or 60.3) and also the wall thickness (pipe schedule) and corrosion allowance and Fluid density.

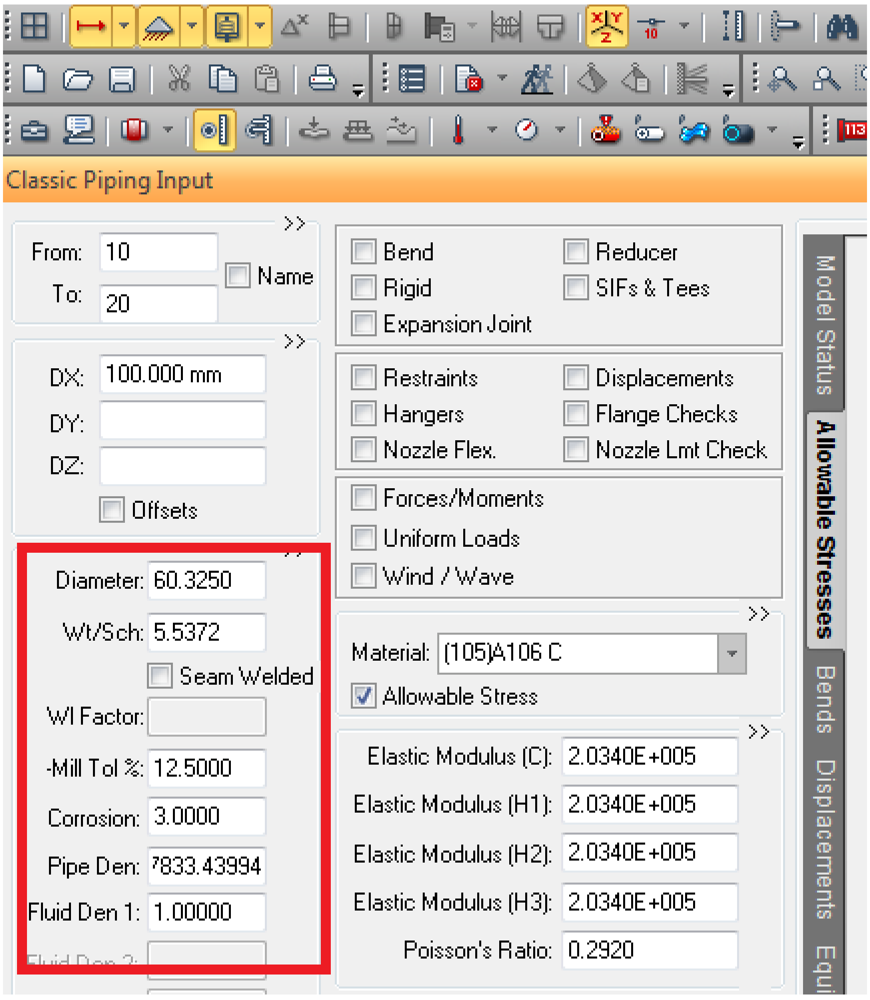



*Figure 6 pipe parameters (classic piping input)*

**Step-6**

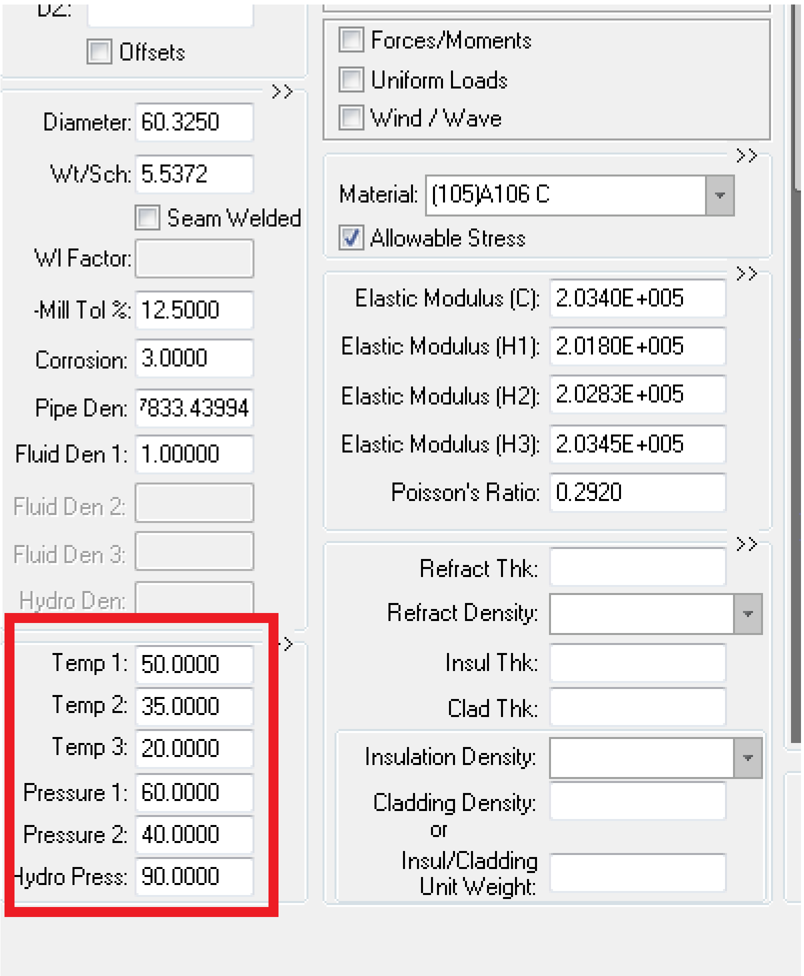



*Figure 7 Temperature & Pressure input table (classic piping input)*

As per the Analysis requirements the temperature values (Design temp., operating temp., minimum design temp. or atm temp.) and pressure values (design pressure, operating pressure) are filled to evaluate the piping stress analysis.

**Step-7**

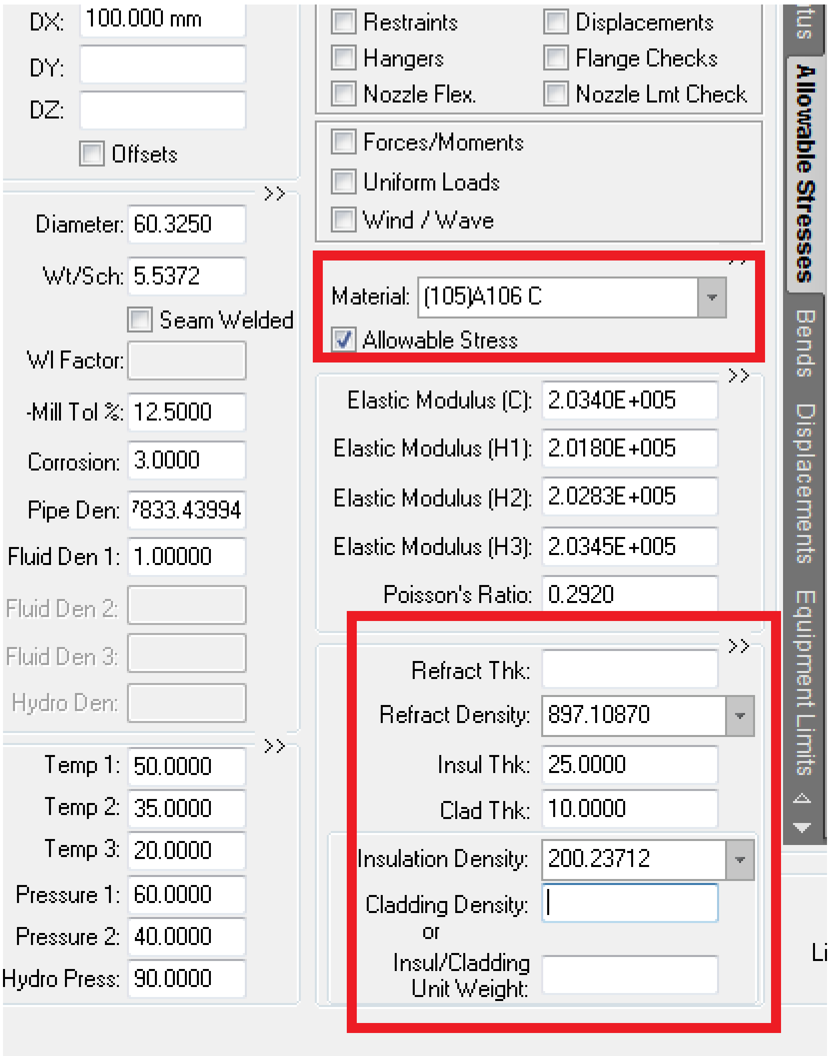
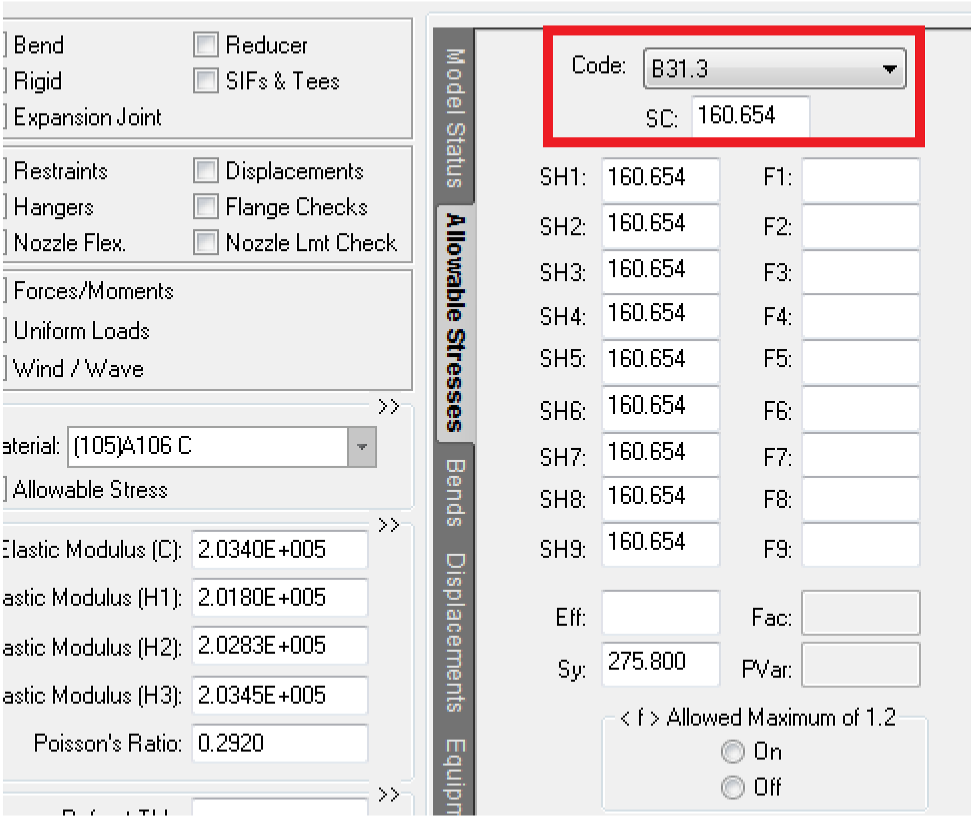



*Figure 8 Material and insulation dialogue box (classic piping input)*

We have to select the material of the pipe or component by using the tool (highlighted box) in the classic piping input tab.

And also, if the pipe or component having insulation / cladding, we have to provide the details using the tools (highlighted box) in the classic piping input tab.

**Step-8**




Using this dropdown button we can select the required piping code from the list.

*Figure 9 piping code (classic piping input)*

### 3.2.1 HOW TO MODEL PIPE FITTINGS

The below figure shows the piping components (pipe fittings) modeling tool in the classic piping inputs tab. Here we can add the piping components like tee, elbow (bend), and reducer, expansion joint, rigid.

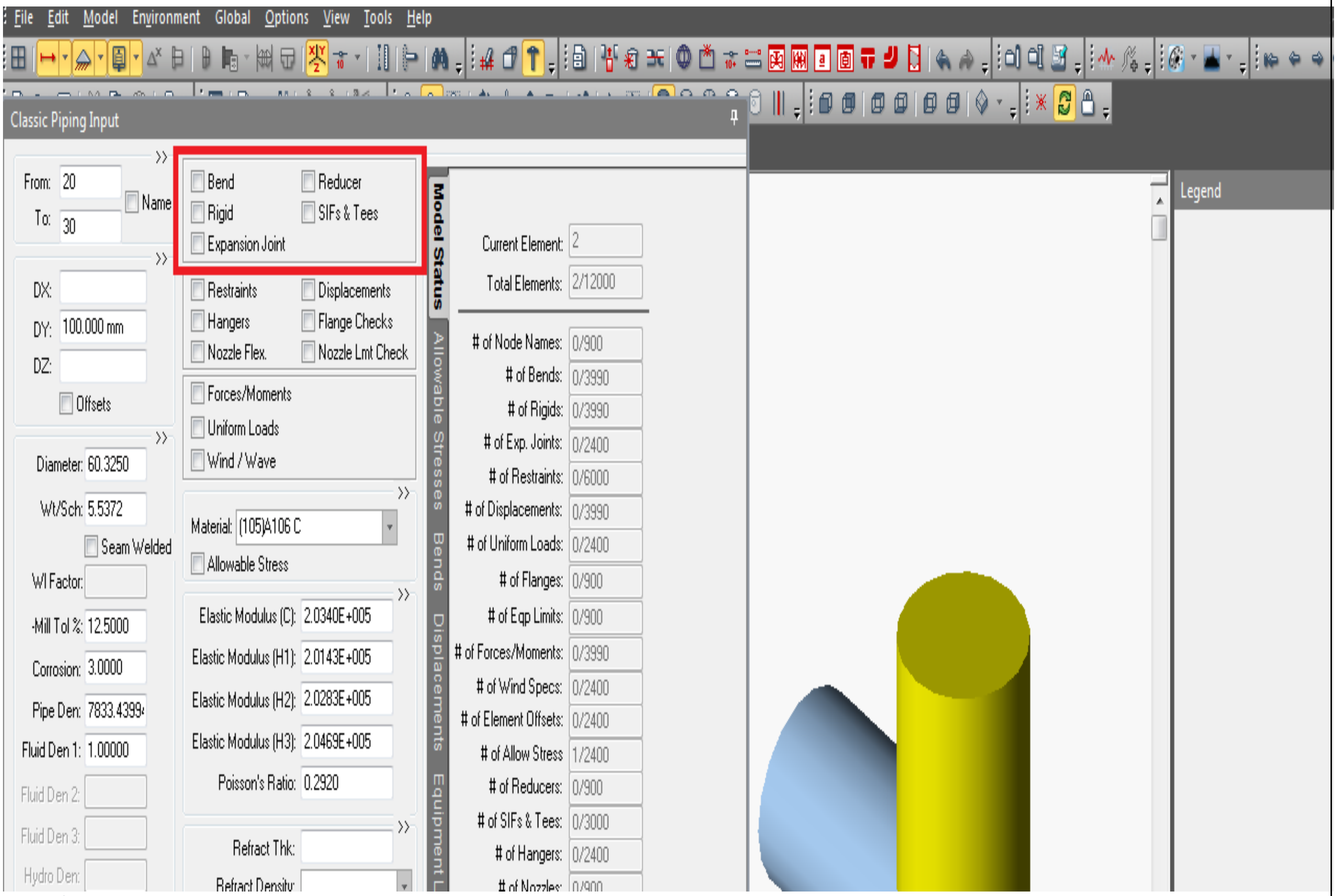



*Figure 10 pipe fittings modelling tools (classic piping input)*

We can add or delete the required options by double clicking the check box. For adding the pipe components doubble click on the check box at once an auxiliary dialouge box will appear using that dialouge box we can model the pipe fittings.

## BEND

By using the bend option we can model the elbow (bend or slope) of the pipeline.

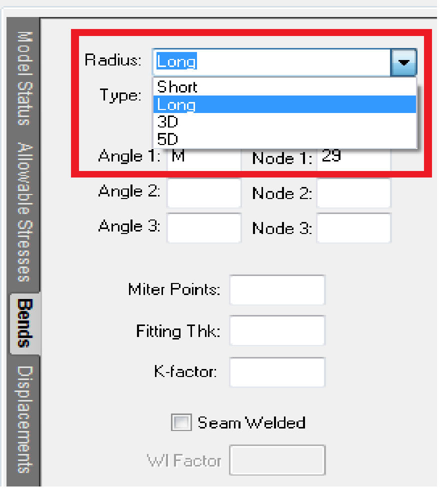


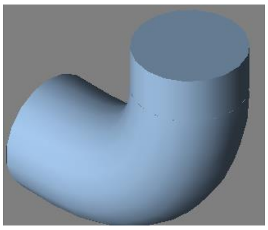

*Figure 11 Auxiliary dialogue box for Bend*

Model a pipe (eg.in X direction)

Double click on the bend option

Select bend type in auxiliary dialogue box

Model another pipe (eg.in Y direction)



## REDUCER

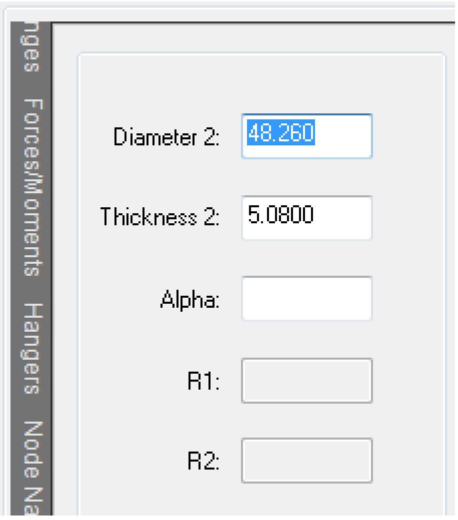
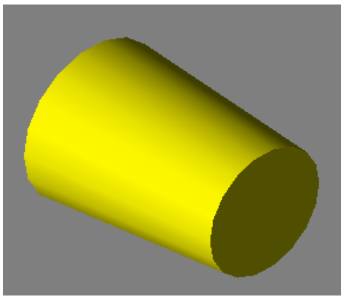



*Figure 12 Auxiliary dialogue box for Reducer*

Model the reducer length (eg.for 2” x 1.5”=76.2mm)

Double click on the ‘reducer’ option

Enter the reducing or increasing diameter & thickness



## TEE

Here we see one of the way to model a tee. We are going to model a butt welded equal tee for a 2” pipeline.

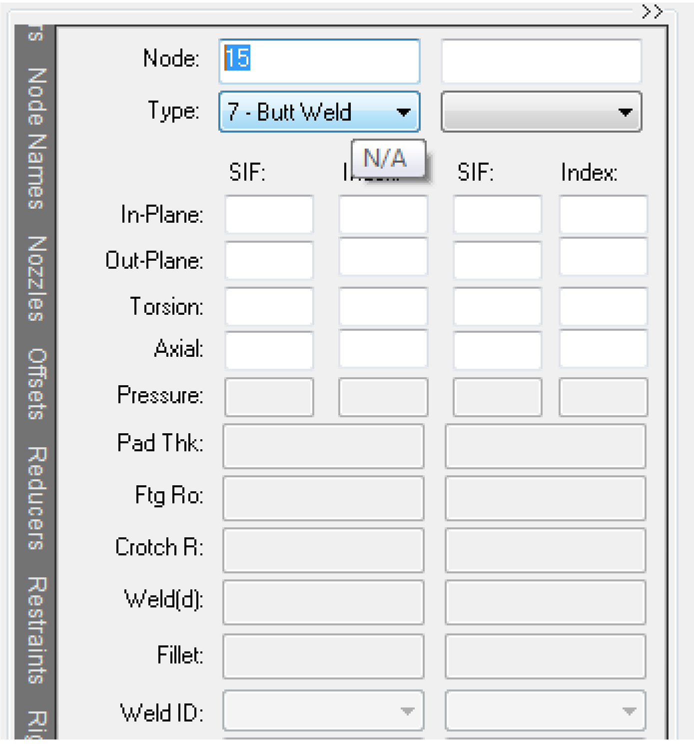



*Figure 13 Auxiliary dialog box for Tee*

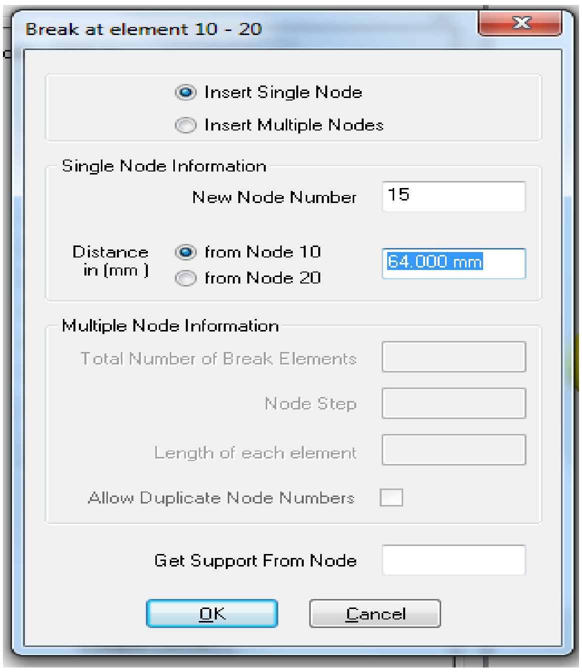



*Figure 15 Break element dialogue box*

Model a pipe in 'X' direction (128mm for 2")

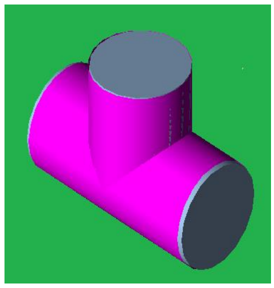
Break the pipe at mid (ctrl+B)

Assign new node number at breaking point

Select the first pipe piece & double click the SIFs&Tees option

Provide the node number & select the type of joint refer figure 13

Model a pipe in 'Y' direction (64mm for 2") Using insert element icon (ref.fig14)



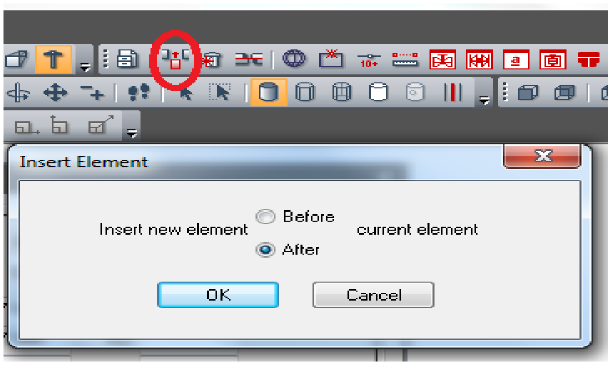



*Figure 14 Insert element Icon with dialogue box*

## EXPANSION JOINT

Expansion joints are used where thermal expansion need to be absorbed. And it also used to absorb the vibration of the rotary equipment.

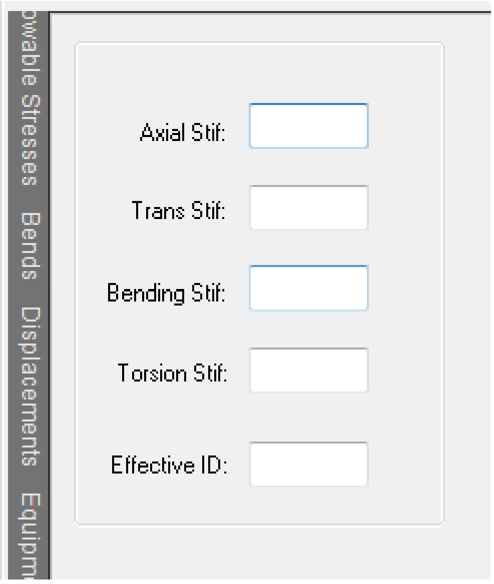


Figure 16 Auxiliary dialog box for Expansion joint

We have to get the Mechanical properties of the expansion joint from the Vendor and then fill that in the shown dialogue box (refer figure 16)

And also we can select the expansion joint detail from the CAESAR-II database by selecting Expansion joint icon.

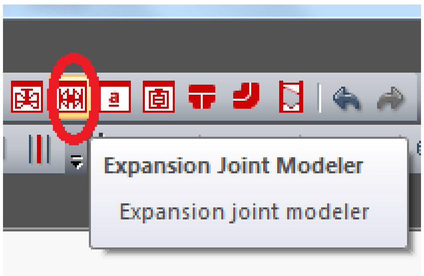


Figure 17 Expansion joint modeller Icon

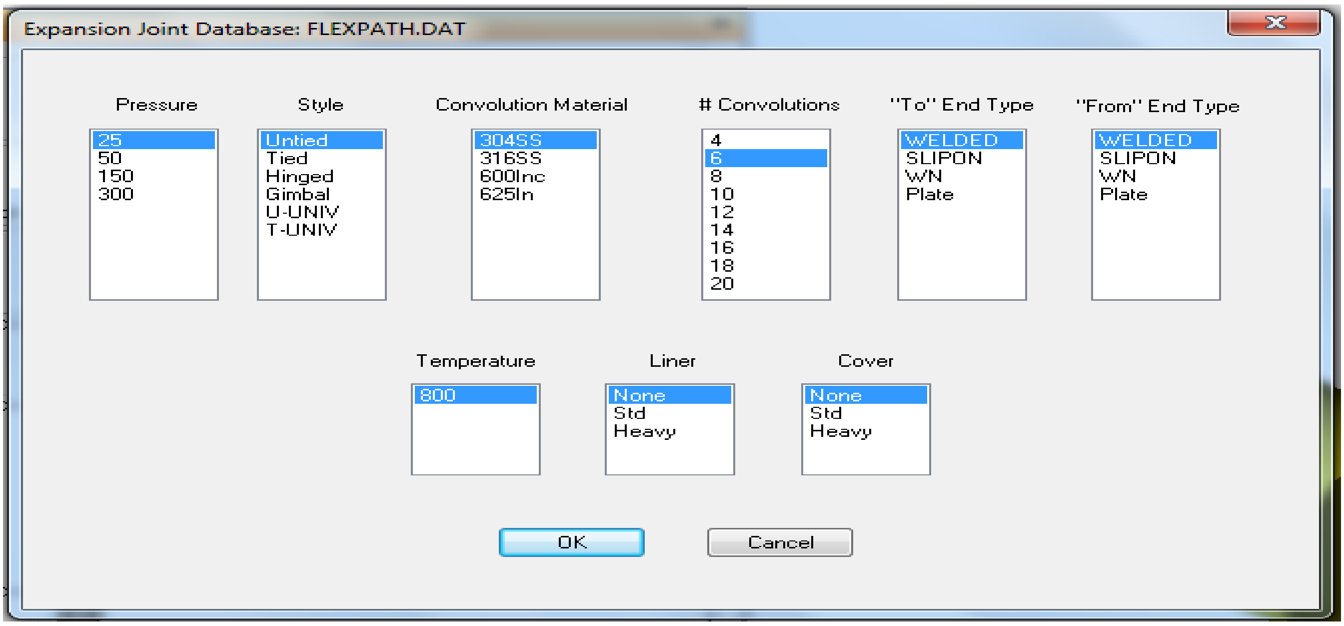


Figure 18 Expansion joint modeller database dialogue box

**RIGID**

Rigid are used to model valves and flanges and also used to connect the two nodes (i.e. while modelling a vessel the rigid are used to model nozzles at different location)

We can model flanges or valves as a rigid just by entering the flange or valve length and then select rigid. After selecting rigid fill the flange weight (in N) at the dialogue box (refer figure 19)

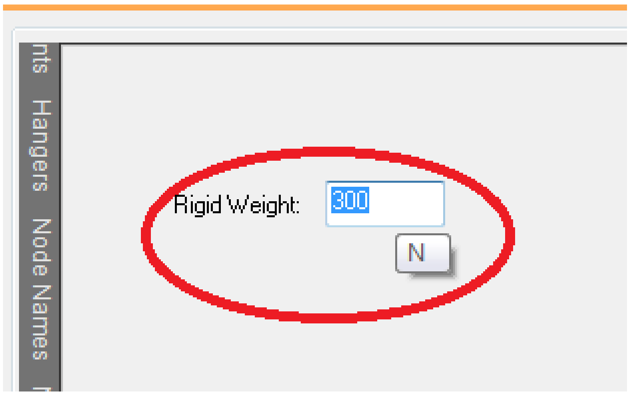
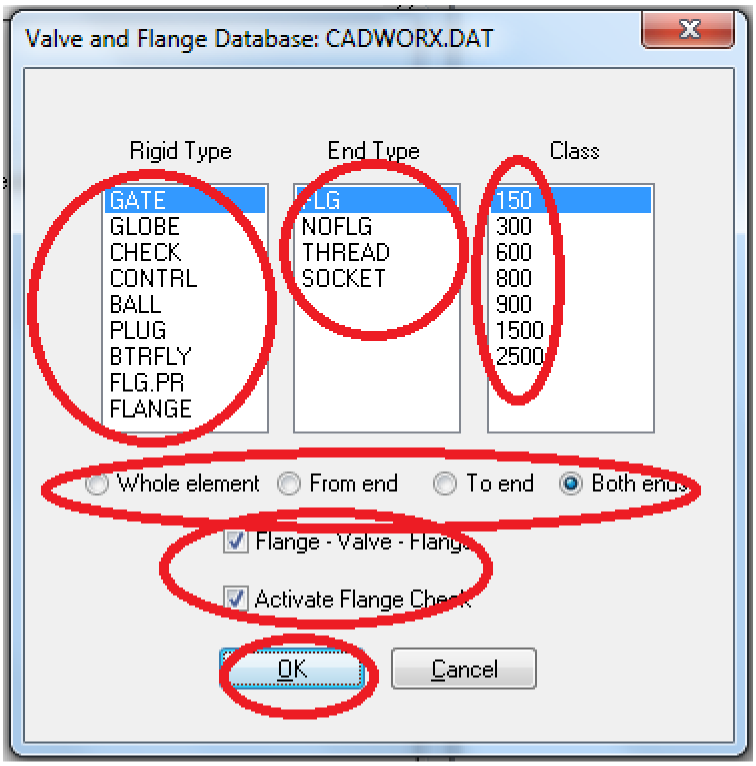



*Figure 19 Auxiliary dialog box rigid weight*

## VALVE

We can model a valve by rigid option or by selecting ‘valve flange database Icon’ 



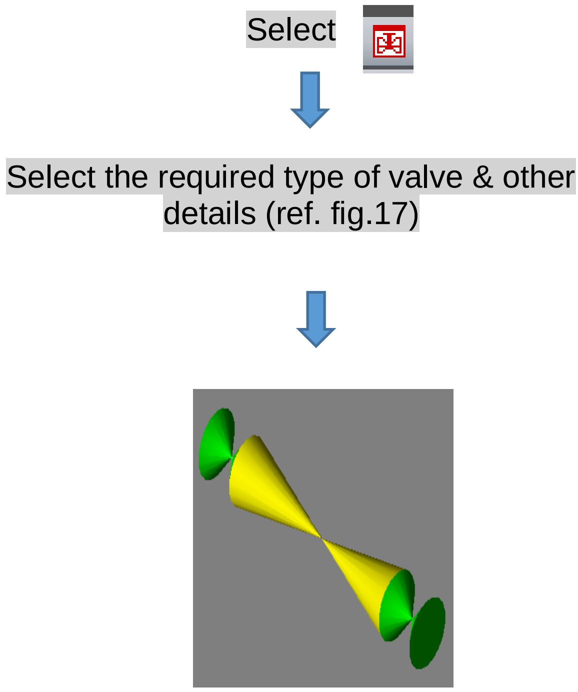




*Figure 20 Valve flange database dialogue box*

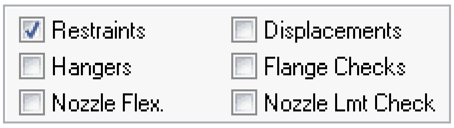
### 3.2.2 ADDITIONAL REQUIRED DATAS FOR ANALYSIS

The following details are required to analyse a piping. Depending up on the requirements of the piping system, these data are taken for analysis.

#### 3.2.2.1 RESTRAINTS

The restraints details are the important thing for a piping analysis process. Restraints are known as the supports of piping system.

There are many types of support available in the CAESAR-II software. We have to assign restraints as per the real model.




Tick indicates that you are supplying restraint data. Select or clear this option by double-clicking the Restraints check box on the Classic Piping Input dialog box.

This auxiliary dialog box tab controls data for up to four restraints for each element. Node number and restraint types are required. All other information is optional. If you omit The stiffness, entry defaults to rigid.

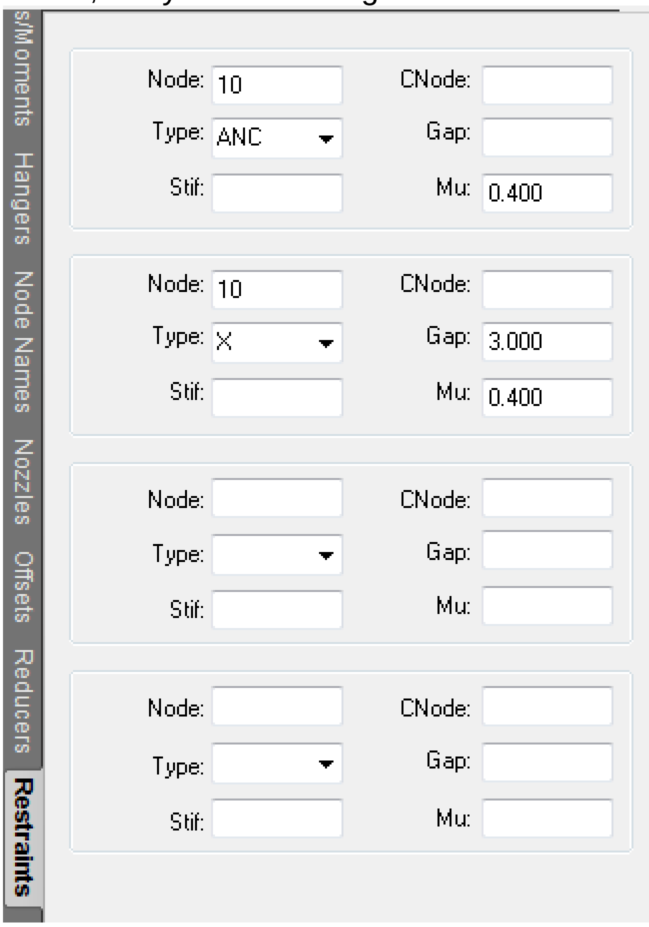



*Figure 21 Auxiliary dialogue box for adding restraints*

You can specify as many as four restraints for an element. If you need to specify more than four restraints on one element, you can place the additional restraints on any other element.

Do not use restraints in the following three situations:

- Imposed Displacements - Specify displacements for the point using the Displacement Auxiliary box.
- Flexible Nozzles - Use the Nozzles Flex check box to open the Nozzles Auxiliary Data box to input the vessel or tank characteristics required by WRC

297, PD 5500, or API 650 to calculate local nozzle flexibilities. After these flexibilities have been calculated, CAESAR II automatically inserts the necessary restraints and flexibilities into the piping model.

- Hangers program designed or pre-defined spring hangers - Use the Hangers check box to open the Hanger Auxiliary Data box.

The types of CAESAR II restraints are listed in the table below.

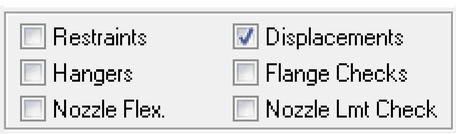
Restraint Types:

| Sign | Vector | Modifiers |
|---|---|---|
| | ANCHOR | |
| (+/-) | X<br>Y<br>Z | + stiff, gap, mu |
| (+/-) | RX<br>RY<br>RZ | + stiff, gap |
| | GUIDE | + stiff, gap, mu |
| (+/-) | LIMIT (axial) | + stiff, gap, mu |
| (+/-) | XROD<br>YROD<br>ZROD | + stiff, length, Fi |
| (+/-) | X2<br>Y2<br>Z2 | + K1, K2, Fy |
| (+/-) | RX2<br>RY2<br>RZ2 | + K1, K2, Fy |
| | XSPR<br>YSPR<br>ZSPR | + stiff, "x", F |
| (+/-) | XSNB<br>YSNB<br>ZSNB | + stiff |

#### 3.2.2.2 DISPLACEMENTS

Tick indicates that you are supplying displacement data. Select or clear this option by double-clicking **Displacements** on the **Classic Piping Input** dialog box.

The auxiliary dialog box tab controls imposed displacements for up to two nodes for each element. If a displacement value is entered for any vector, this direction is considered to be fixed for any other nonspecified vectors.



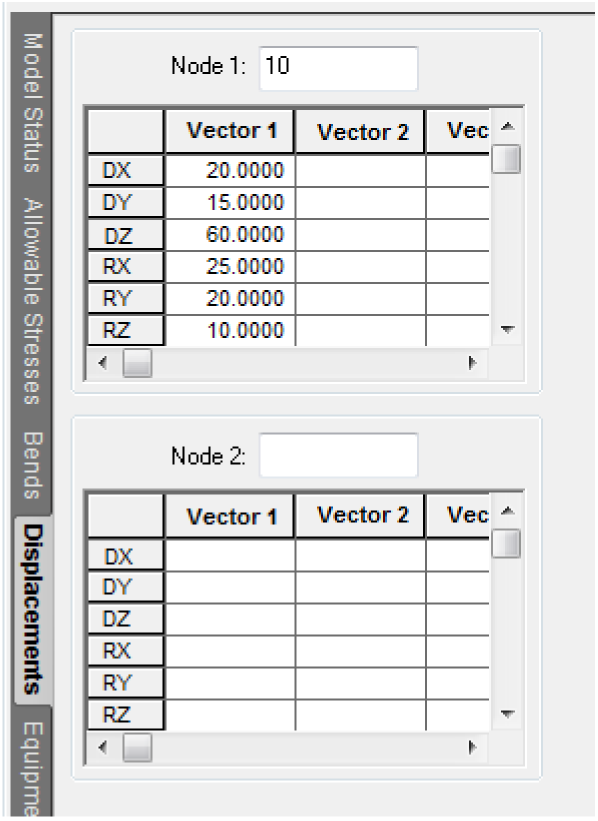
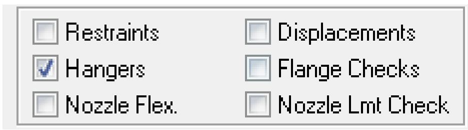


*Figure 22 Auxiliary dialogue box for Displacement Input*

### 3.2.2.3 HANGERS

Tick indicates that you are supplying Hanger data. Select or clear this option by double-clicking **Hangers** on the **Classic Piping Input** dialog box.

Spring Hangers are mostly used to absorb the vertical uplift, the auxiliary dialog box tab used to provide hanger data.



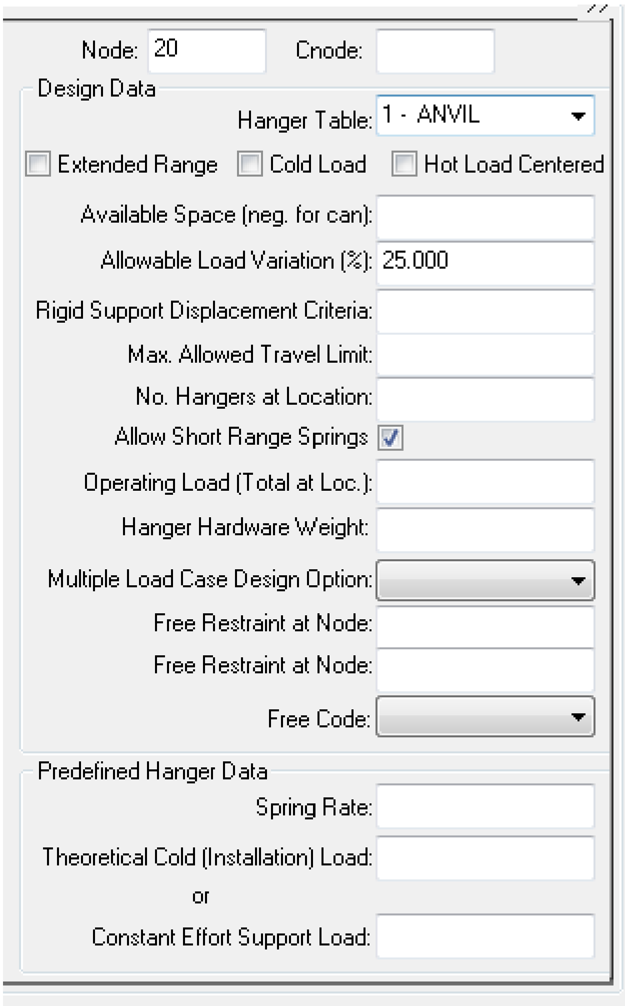



*Figure 23 Auxiliary dialogue box for Hanger design*

### 3.2.2.4 FLANGE CHECK

Flange check is used to determine the flange leakage in the piping system.
Tick indicates that you are supplying flange data to evaluate an in-line flange. Select or clear this option by double-clicking the Flange check box on the Classic Piping Input dialog box.

You can read the values for the Flange Class/Grade and Gasket Diameter, G boxes from a file if you select ASME - 2003 from the Flange Pressure Ratings dialog box. The G values are located in the ASME-2003.G text file in the system folder under the application data folder.

Flange evaluation is based on a specific load case temperature. To evaluate the flanges in a model, use the Load Case Options tab of the Static Load Case Editor to specify to which operating temperature the flanges should be evaluated.

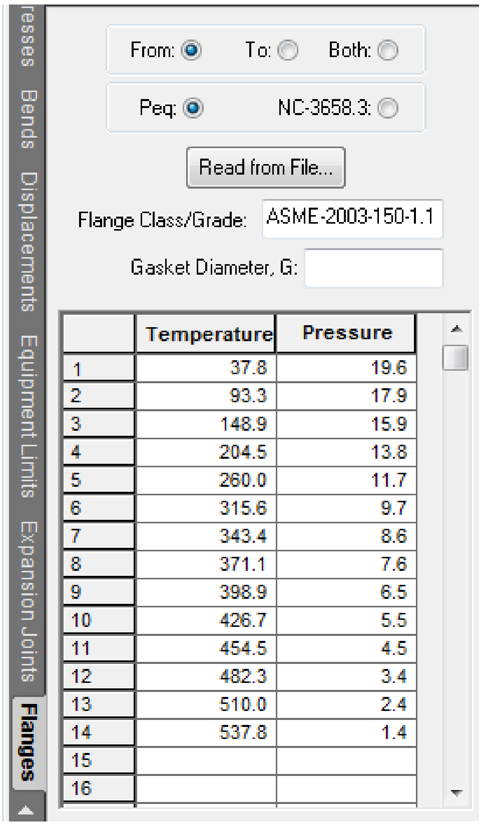
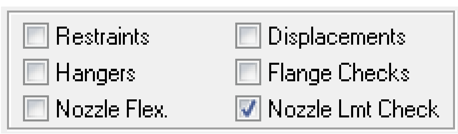




*Figure 24 Auxiliary dialogue box for Flange check*

### 3.2.2.5 NOZZLE LIMIT CHECK

Nozzle limit check is used to compare the allowable nozzle load of an equipment nozzle with the actual.

CAESAR II enables you to define overall *nozzle limits*. This permits CAESAR II to perform a first pass screening. Actual detailed nozzle evaluation can then be focused on those nozzles that fail this initial screening.



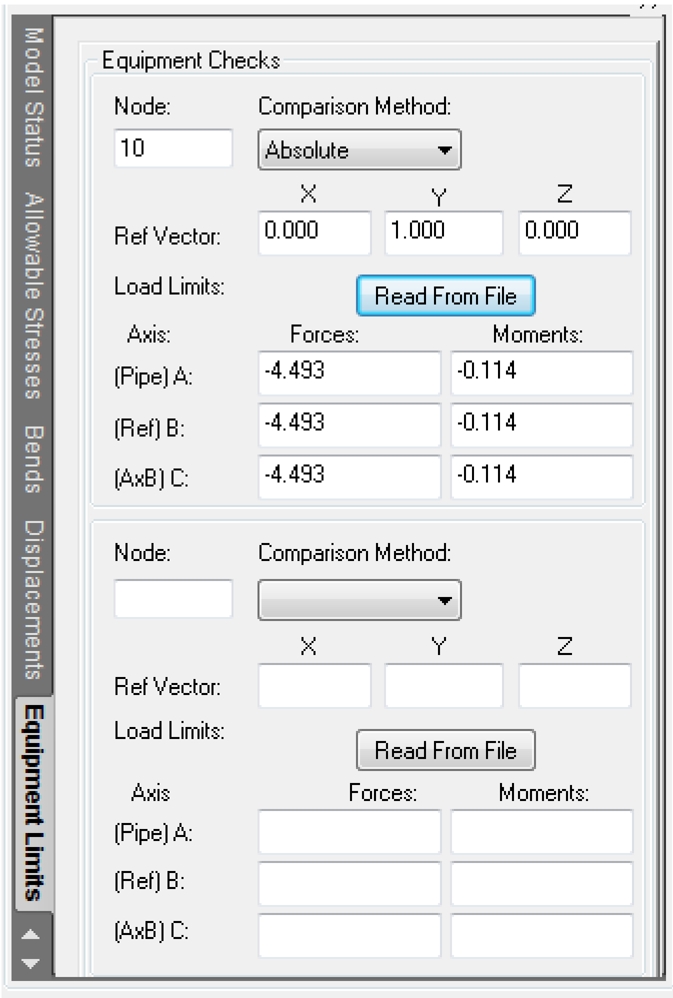
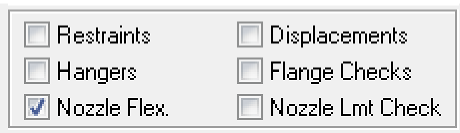



*Figure 25 Auxiliary dialogue box for Nozzle limit check*

### 3.2.2.6 NOZZLE FLEX

This option is used to define flexible nozzle connections. Here we can define the nozzle without modelling Vessel or Tank or equipment.

When you type values in this dialog box tab, CAESAR II automatically calculates the flexibilities and adds them to the active element. CAESAR II calculates nozzle loads according to WRC 297, API 650 or BS 5500 criteria.



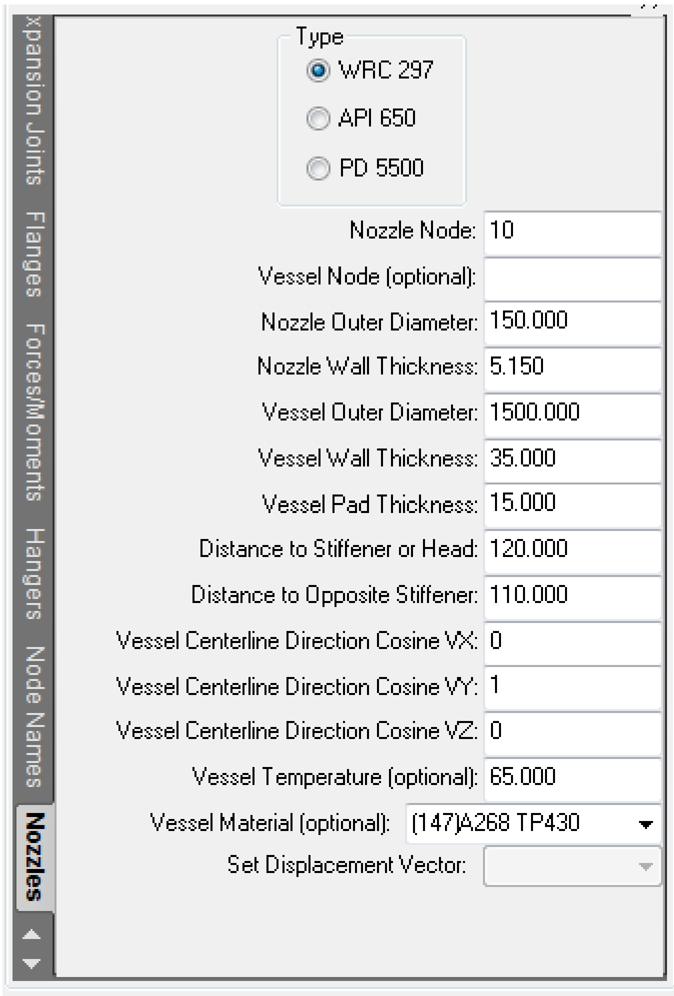




*Figure 26 Auxiliary dialogue box for Nozzle Flex*

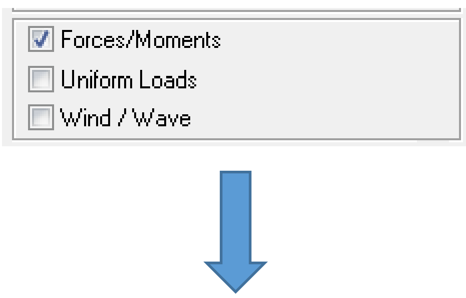
### 3.2.3 OCCASIONAL LOADS FOR ANALYSIS

The following three are the Occasional data used for piping analysis. Depending up on the requirement of the piping system these loads are provided.

#### *3.2.3.1 FORCES / MOMENTS*

This option is used to provide the Forces & Moments of external piping attachment to the working skid moreover this is used to provide the Forces & Moments of a PSV to PSV outlet.

This auxiliary database tab controls imposed forces or moments for up to two nodes per element. You can use up to nine force vectors.



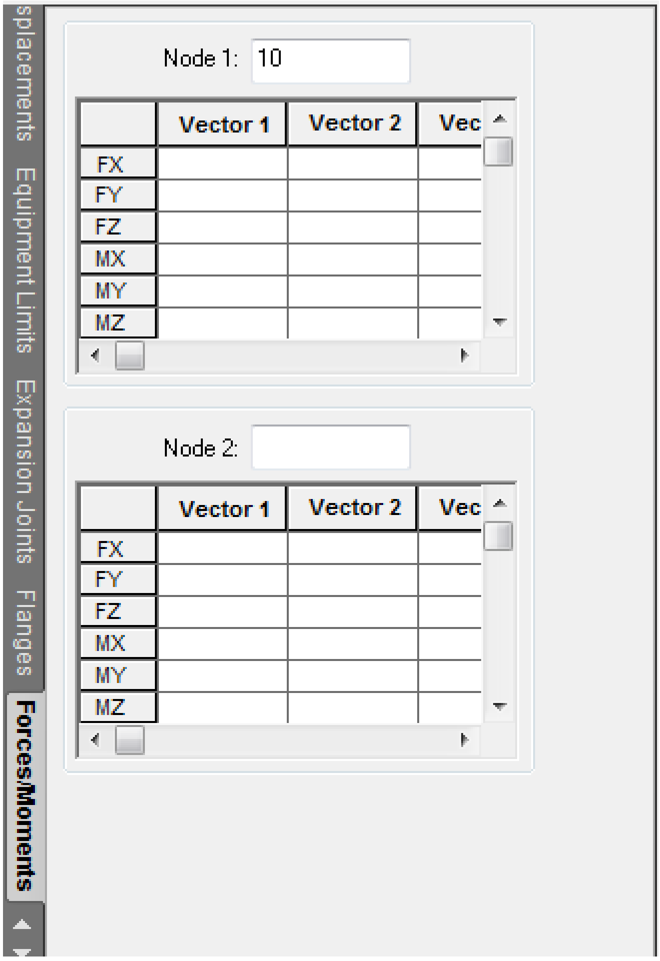


*Figure 27 Auxiliary database tab for Forces & Moments*

### 3.2.3.2 UNIFORM LOADS

The uniform loads option is to provide the Seismic loads & snow loads for a piping system.

This auxiliary database tab controls up to three uniform load vectors. These uniform loads are applied to the entire current element, as well as all subsequent elements in the model, until explicitly changed or zeroed out.

The uniform load data is distributive and applies to current and all following elements until you change it.

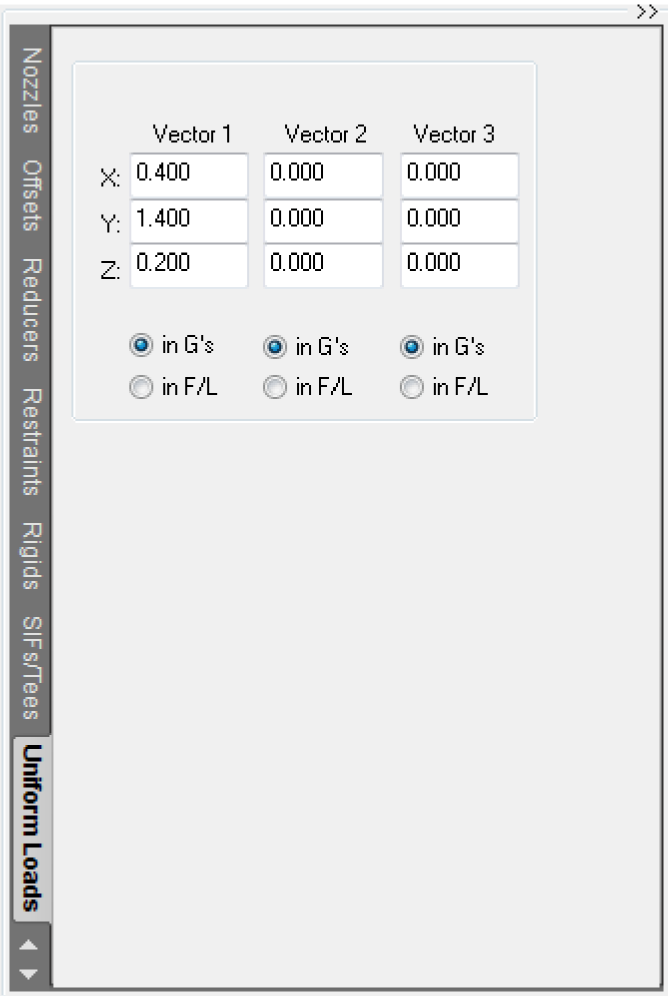



*Figure 28 Auxiliary database tab of Uniform loads*

### 3.2.3.3 WIND/WAVE

This option is used to provide wind load or wave load to a piping system.

This auxiliary dialog box tab indicates whether this portion of the pipe is exposed to wind or wave loading. The pipe cannot be exposed to both.

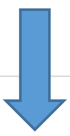
Selecting Wind exposes the pipe to wind loading; selecting Wave exposes the pipe to wave, current, and buoyancy loadings; selecting Off turns off both types of loading

.

This dialog box tab is also used to specify the Wind Shape Factor when Wind is specified. The dialog box tab is used to specify various wave coefficients when Wave is specified. The software automatically computes the wave coefficients if you leave these boxes blank.

Entries on this auxiliary dialog box tab apply to all subsequent piping, until changed on a later element.

Specific wind and wave load cases are built using the **Static Load Case** Editor.

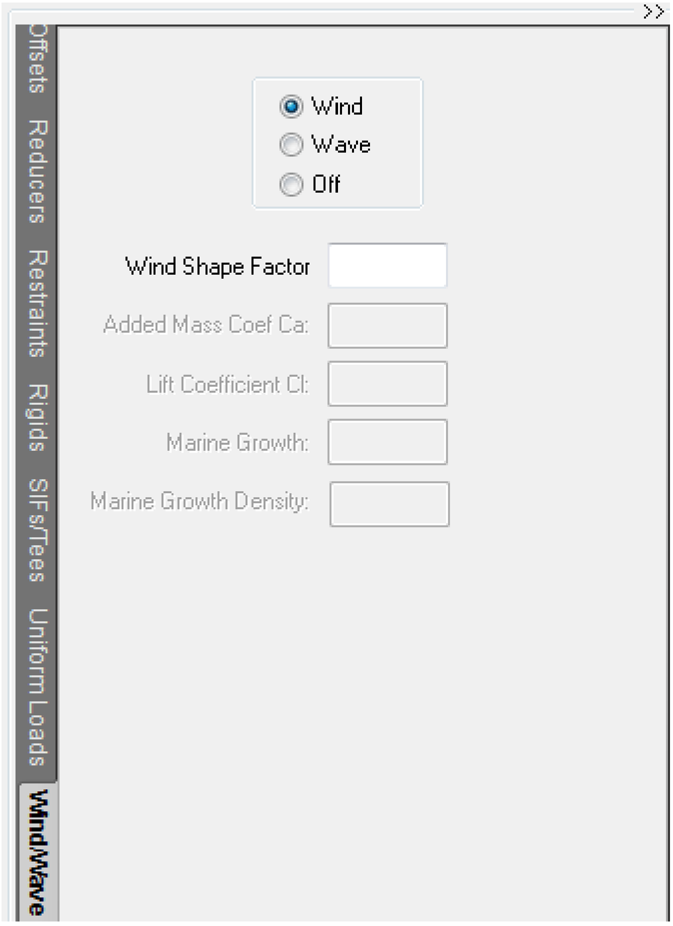



*Figure 29 Auxiliary database tab for Wind/Wave*

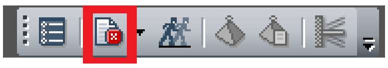
## 3.3 HOW TO ANALYSE IN CAESAR-II

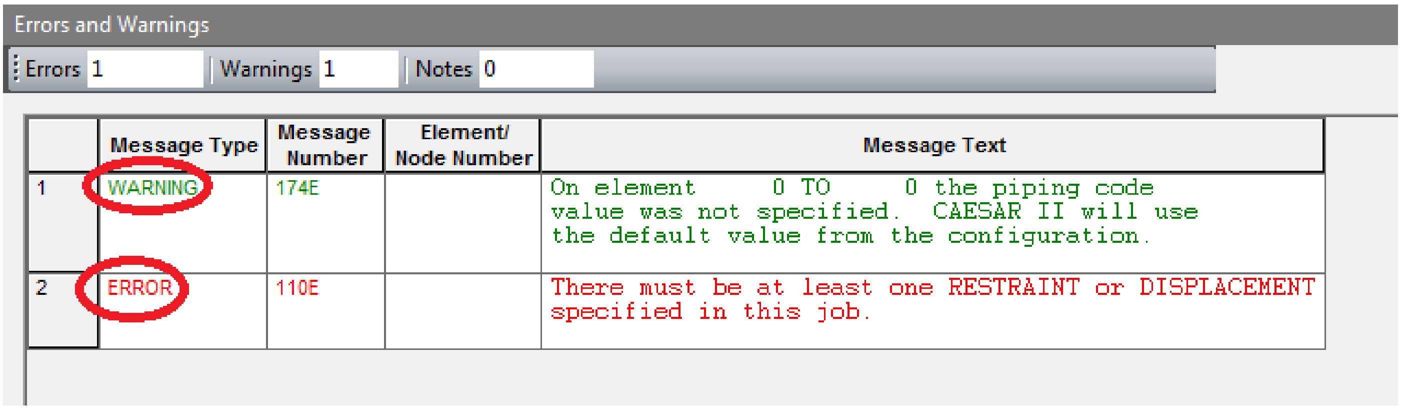
After completing the modelling, we have to analyse the piping using required load cases to evaluate, stresses in the piping system, displacement, loads on the support (restraint) of the piping, loads on the equipment nozzle.

Using **CAESAR-II tools** tool bar we can analyse the piping system.

### 3.3.1 ERROR CHECK

This ICON is for checking the Errors and Warnings in a modeling.After rectifying the error only we can analyse the piping system.



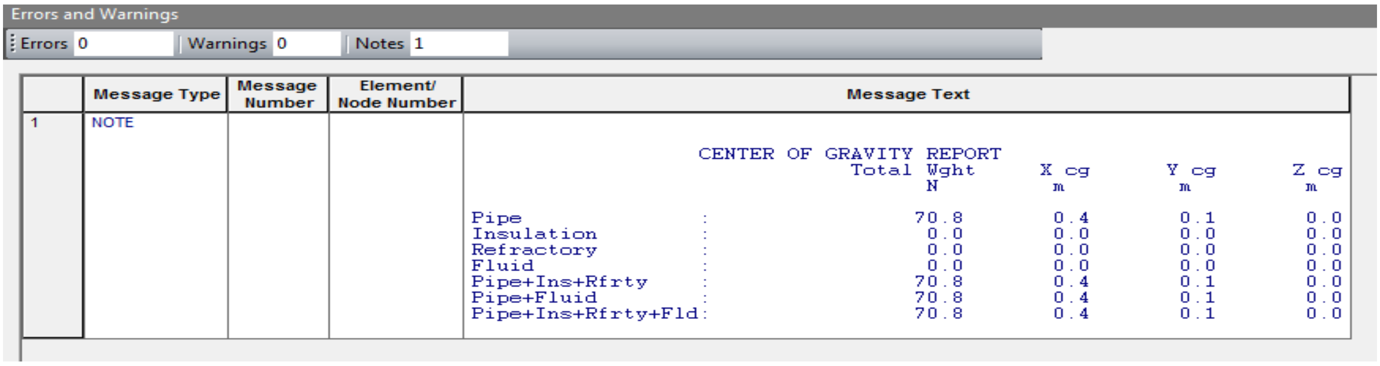


Errors and Warnings

Errors 1 | Warnings 1 | Notes 0

| | Message Type | Message Number | Element/ Node Number | Message Text |
|---|---|---|---|---|
| 1 | WARNING | 174E | | On element 0 TO 0 the piping code value was not specified. CAESAR II will use the default value from the configuration. |
| 2 | ERROR | 110E | | There must be at least one RESTRAINT or DISPLACEMENT specified in this job. |

After rectifying the errors



Errors and Warnings

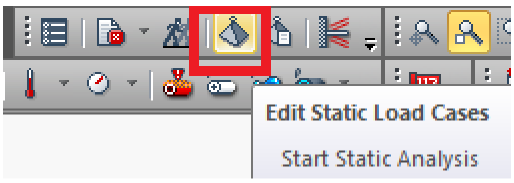
Errors 0 | Warnings 0 | Notes 1

| | Message Type | Message Number | Element/ Node Number | Message Text |
|---|---|---|---|---|
| 1 | NOTE | | | CENTER OF GRAVITY REPORT |

| | Total Wght N | X cg m | Y cg m | Z cg m |
|---|---|---|---|---|
| Pipe : | 70.8 | 0.4 | 0.1 | 0.0 |
| Insulation : | 0.0 | 0.0 | 0.0 | 0.0 |
| Refractory : | 0.0 | 0.0 | 0.0 | 0.0 |
| Fluid : | 0.0 | 0.0 | 0.0 | 0.0 |
| Pipe+Ins+Rfrty : | 70.8 | 0.4 | 0.1 | 0.0 |
| Pipe+Fluid : | 70.8 | 0.4 | 0.1 | 0.0 |
| Pipe+Ins+Rfrty+Fld: | 70.8 | 0.4 | 0.1 | 0.0 |

### 3.3.2 EDIT STATIC LOAD CASES

This ICON is for edit the required load cases combination to analyse the piping system. Using this option we can edit, create, and upload load cases to analyse.



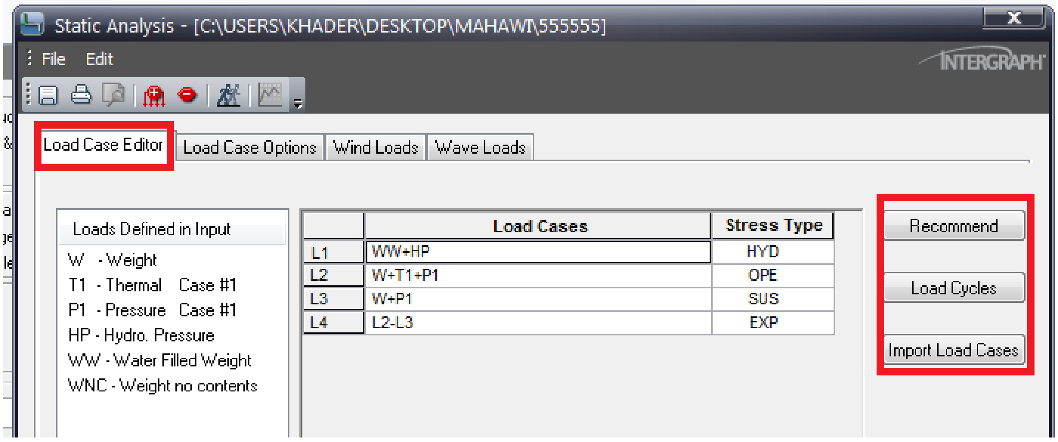



*Figure 30 Load Case Editor (using this option we can create or edit or import load cases for a piping system)*

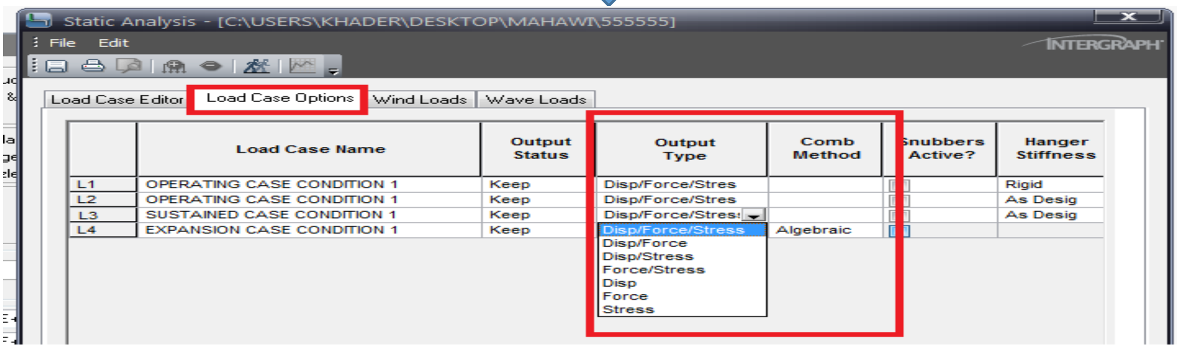
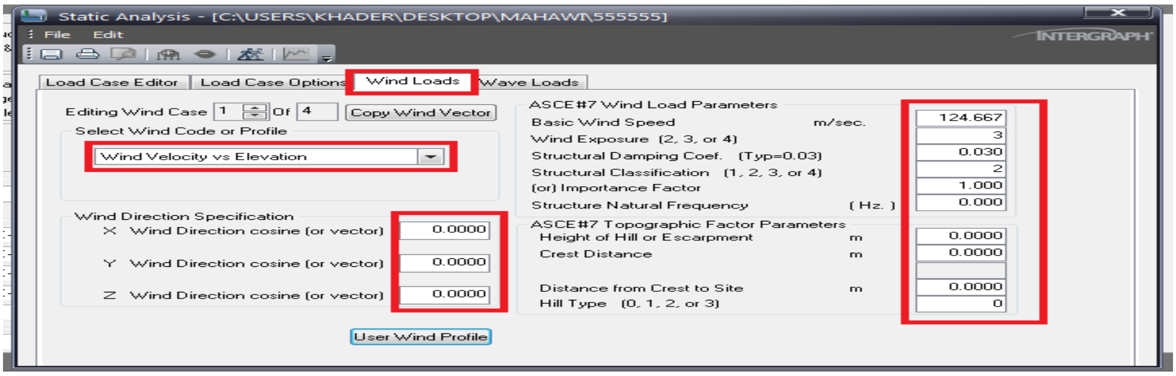



*Figure 31 Load Case Option (using this we can manage load case combination method and output type etc.)*




Likewise the Wave load (if applicable) details also entered in the editor.

### 3.3.2.1 INTRODUCTION TO LOADCASE

<u>TYPESS OF LOAD</u>

- Primary Loads – Force driven, cause catastrophic failure.
  (Weight, Pressure, Point Loads, Uniform Loads, Hanger Loads, Wind and Wave loads.)
- Secondary Loads – Strain based, cause fatigue failure.
  (Temperature, Displacements.)

*Figure 32 Wind load (Using this option we can manage the wind data..)*

## AVAILABLE LOAD TYPES IN CAESAR II

- W (Weight), WNC (Weight No Contents)
- WW (Water-filled Weight)
- P (Pressure), HP (Hydro test Pressure)
- T (Temperature), D (Displacement)
- H (Hanger Pre-loads), F (Concentrated Loads)
- U (Uniform Loads)
- Win (Wind), Wav (Wave)

## AVAILABLE STRESS TYPES IN CAESAR II

- OPE – Operating
- SUS – Sustained
- EXP – Expansion
- OCC – Occasional
- HYD – Hydro test
- HGR – Hanger Design
- FAT – Fatigue

## LOAD CASE DEFINITION

- Operating case contains all loads in the system.
  L1 = W+P1+T1+H (OPE) this is called a basic load case

- Sustained Case contains only primary loads.
  L2 = W+P1+H (SUS) another basic load case

- Expansion Case is the difference between the operating and sustained cases.
  L3 = L1-L2 (EXP) this is called a combination load case

## COMBINATION LOAD CASES

- Used to add or subtract results from previously defined primitive load cases.
- Necessary for proper EXP and OCC code stress definition.
- Not used for restraint or equipment load definition, nor for displacement reporting.

## OCCASIONAL LOAD CASES

- For most piping codes (not the offshore codes):
  -Set up an OPE case that includes the occasional load
  -Subtract the standard OPE case from the OPE that includes the occasional load. We
  -call this the segregated occasional load case.

-Add the above load case results to the SUS load case results for the code stress check.

**Example for Occasional Load Cases**:

- Assume we have a uniform load representing a seismic load, U1.

| | |
|---|---|
| L1 = W+P1+T1 | (OPE) standard operating |
| L2 = W+P1 | (SUS) |
| L3 = W+P1+T1+U1 | (OPE) operating with occasional load |
| L4 = L1-L2 | (EXP) |
| L5 = L3-L1 | (OCC) segregated occasional |
| L6 = L2+L5 | (OCC) * occasional code stress case |

- * use scalar combination method.

COMBINATION METHODS

- Algebraic:

  -Used for subtracting two load cases.
  -Takes the displacements from the referenced cases and subtracts them.
  -Then computes forces, moments, and resultant stress from these displacements.

- Scalar:

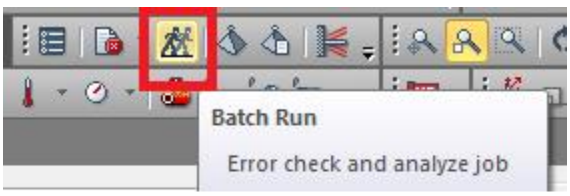
  -Used for adding two load cases.
  -Adds the stresses from the two referenced load cases.
  -Unlike algebraic the stresses are not recomputed from displacements.

Notes:

- Don't use algebraic for adding two load cases.
  -You can't take credit for occasional loads acting opposite to operating loads.
- Don't use scalar for subtracting two cases.
  -This results in a lower code stress than actual.

### 3.3.3 BATCH RUN

After defining the Load cases for the analysis we have to run the model to get the results.



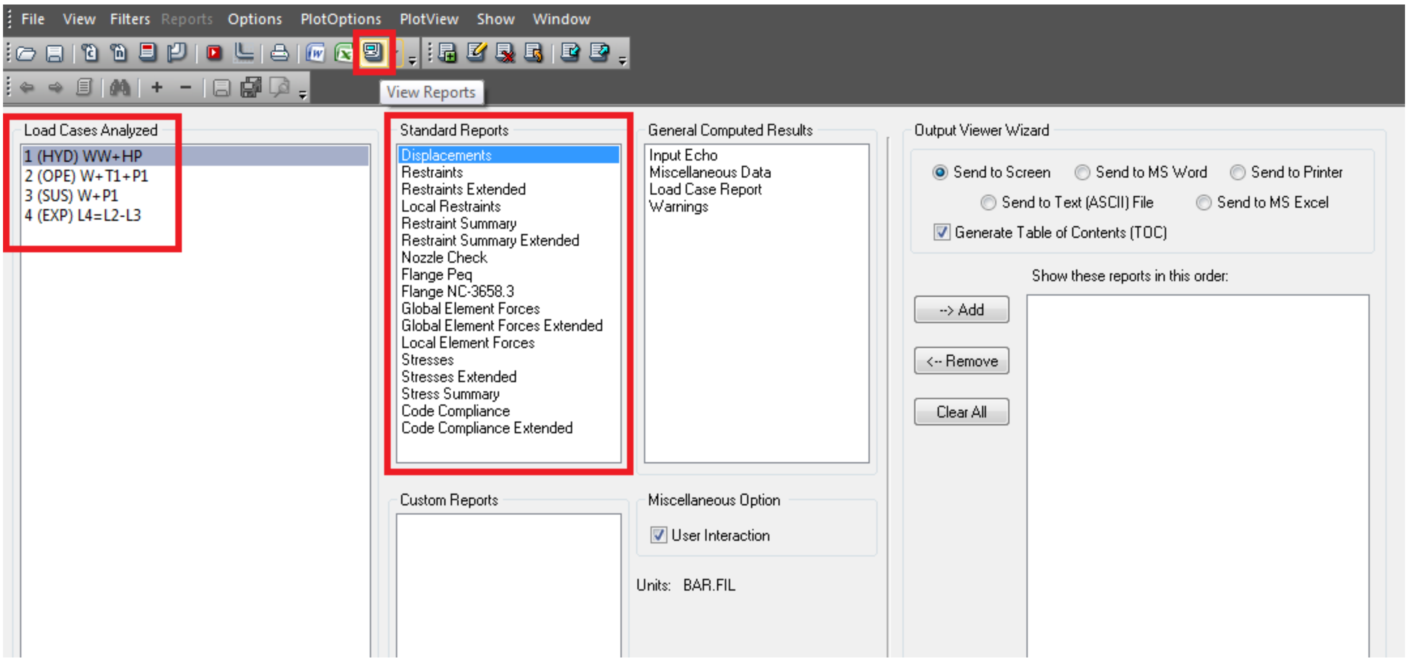


*Figure 33 Output processor (The highlighted options used to quickly view and verify the results*

(1)Displacements

CAESAR II 2014 Ver.7.00.00.2800, (Build 140416) Date: AUG 16, 2017 Time: 16:47

Job Name: 555555

Licensed To: SPLM: Edit company name in <system>\company.txt

DISPLACEMENTS REPORT: Nodal Movements

CASE 1 (HYD) WW+HP

| Node | DX mm | DY mm | DZ mm | RX deg. | RY deg. | RZ deg. |
|---|---|---|---|---|---|---|
| 10 | -0.000 | -0.000 | 0.000 | 0.0000 | -0.0000 | -0.0000 |
| 18 | -0.000 | -0.027 | 0.000 | 0.0000 | -0.0000 | -0.0056 |
| 19 | 0.002 | -0.033 | 0.000 | 0.0000 | -0.0000 | -0.0059 |
| 20 | 0.008 | -0.035 | 0.000 | 0.0000 | -0.0000 | -0.0059 |
| 30 | 0.051 | -0.035 | 0.000 | 0.0000 | -0.0000 | -0.0059 |

*Figure 34 Results (for selected load cases and standard report)*

## 3.4 HOW TO CREATE OUTPUT REPORT IN CAESAR-II

Once the analysis reports are satisfied (piping system is safe) and then we have to generate output reports.

Using output processor (refer figure 35) we can generate reports in MS word, MS excel or directly print the reports.

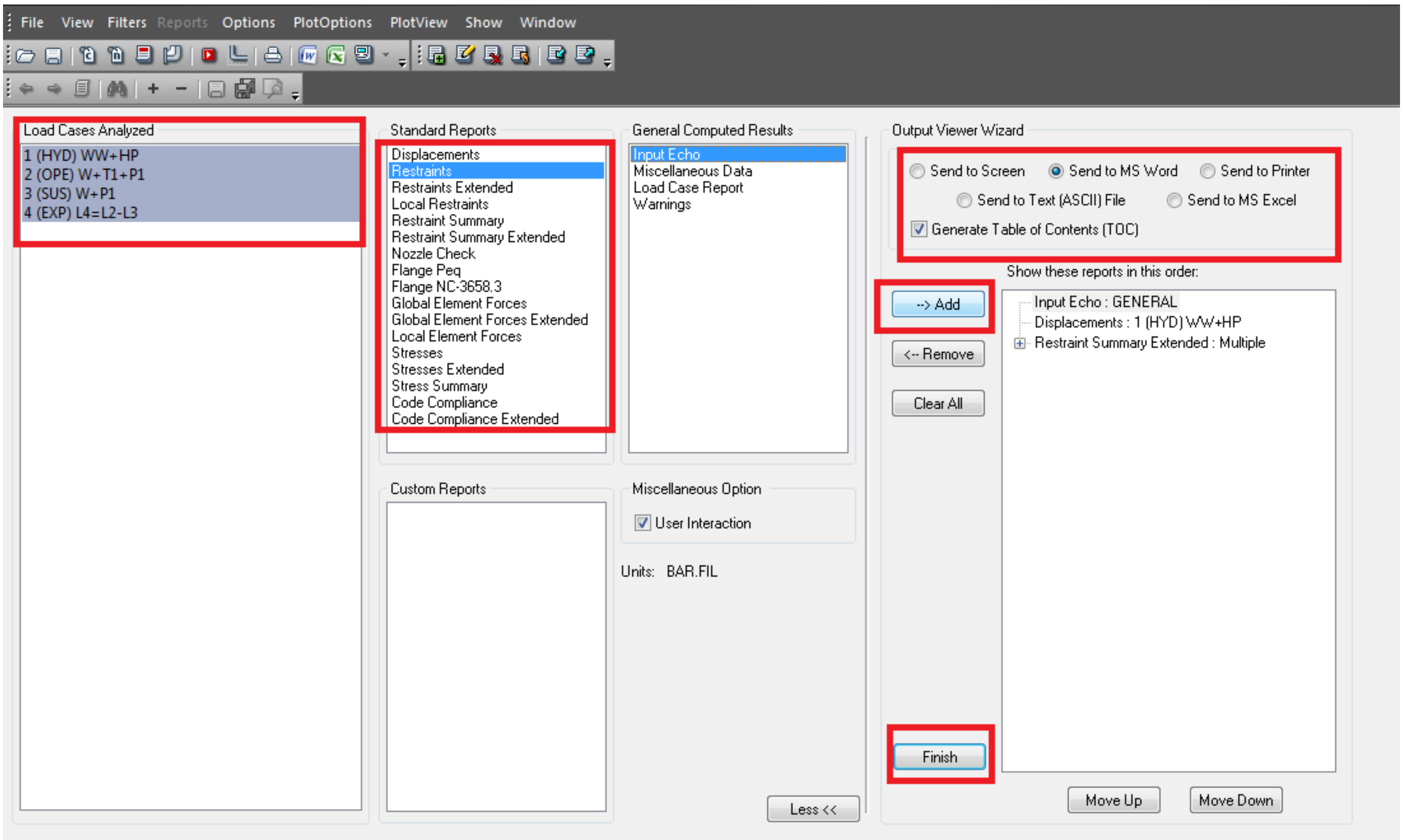


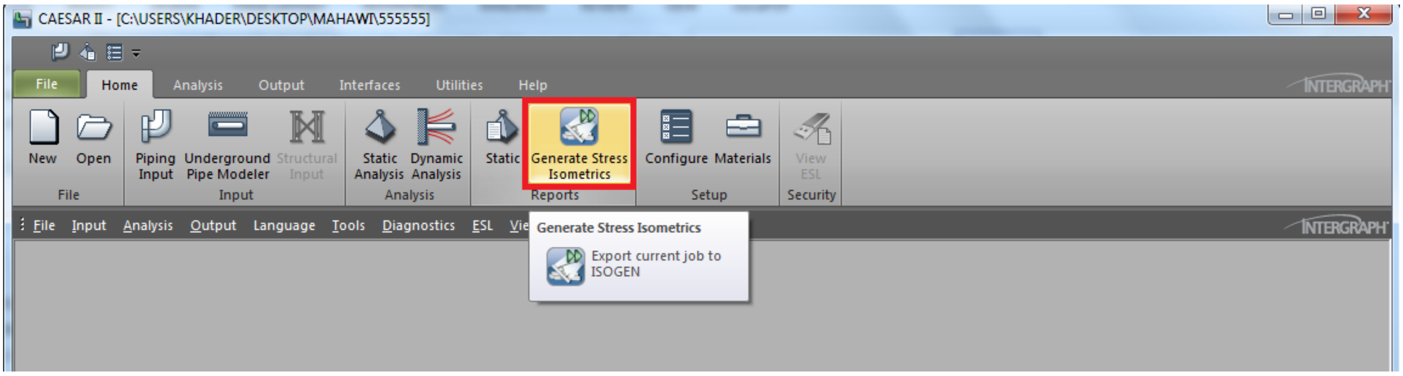

*Figure 35 Output Processor (the highlighted boxes shows the required option to generate the report)*

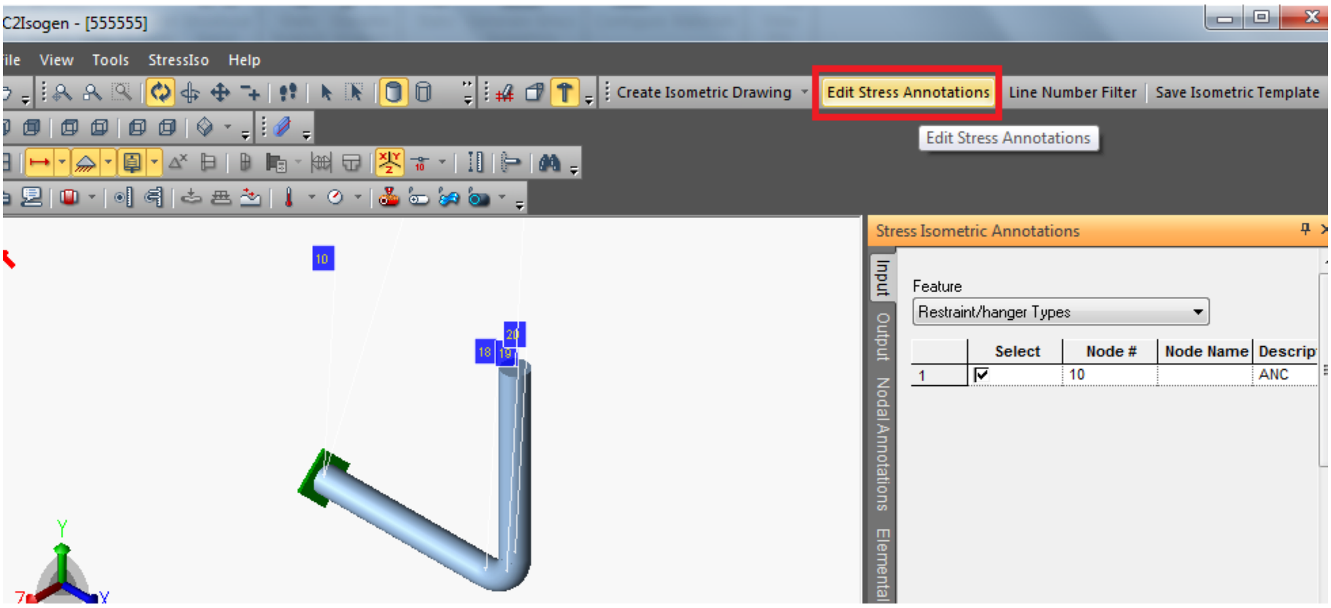
<u>OUTPUT TYPES</u>

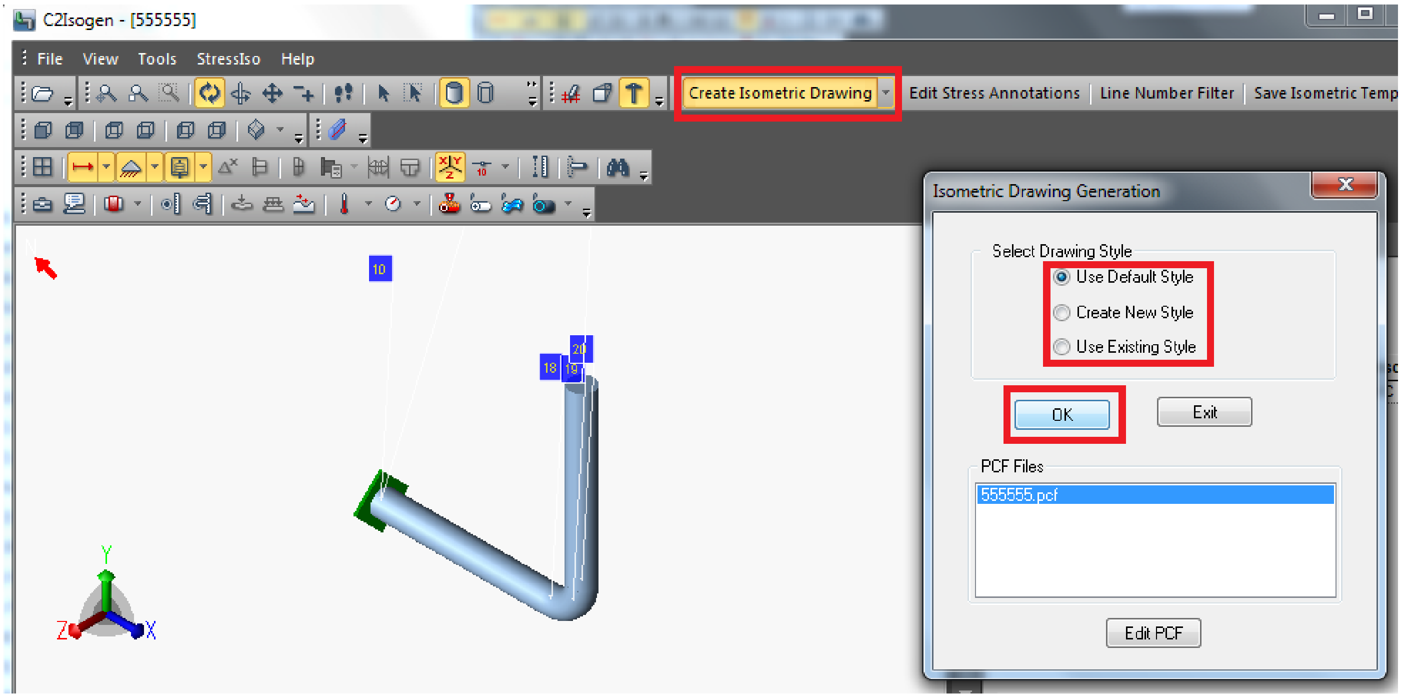
- Displacement:
  -Usually reported only for basic load cases
- Force:
  -Usually reported only for basic load cases
- Stress:
  -Reported based on code requirements.

### 3.4.1 HOW TO CREATE STRESS ISO IN CAESAR-II

Generating stress iso is to provide detailed view of a stress model.


CAESAR II - [C:\USERS\KHADER\DESKTOP\MAHAWI\555555]
File
Home
Analysis
Output
Interfaces
Utilities
Help
New
Open
Piping Input
Underground Pipe Modeler
Structural Input
Static Analysis
Dynamic Analysis
Static
Generate Stress Isometrics
Configure
Materials
View ESL
File
Input
Analysis
Reports
Setup
Security
Generate Stress Isometrics
Export current job to ISOGEN


C2Isogen - [555555]
View
Tools
StressIso
Help
Create Isometric Drawing
Edit Stress Annotations
Line Number Filter
Save Isometric Template
Edit Stress Annotations
Stress Isometric Annotations
Feature
Restraint/hanger Types
Select
Node #
Node Name
Descrip
1
10
ANC
Input
Output
Nodal Annotations
Elemental



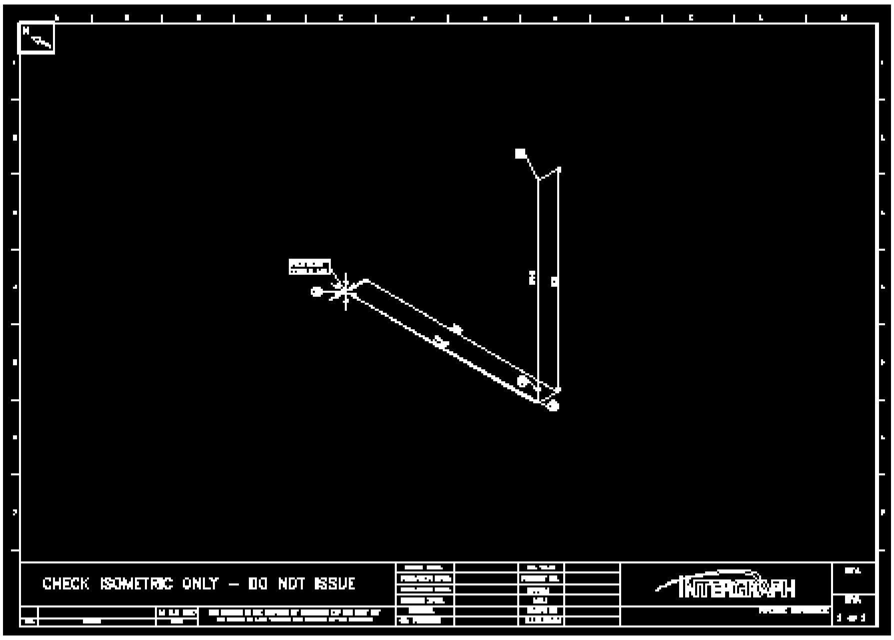




*Figure 36 Stress Iso*

## 4 CONCLUSION & RECOMMENDATIONS

This article provides an introduction about the piping stress analysis using CAESAR-II software.

A piping stress engineer have to conclude and recommend the amendment regarding piping system based on the stress analysis results.

## CONCLUSION:

Displacement:

Based on the Piping design, the computed displacements are within the acceptable range, under all load cases analysed.

Stresses in pipework:

Calculated stresses are within the allowable limits according to taken code for Code Compliance.

## RECOMMENDATIONS:

- Maximum Pipe support loads as a result of the analysis shall be used as a basis for the appropriate Civil / Structural support design.
- Loads on the connecting skids shall be provided to the Skid VENDOR and has to be consider in Skid Piping and Support design.
- Loads on the equipment nozzle shall be provided to equipment vendor and has to be consider in the equipment nozzle design.