

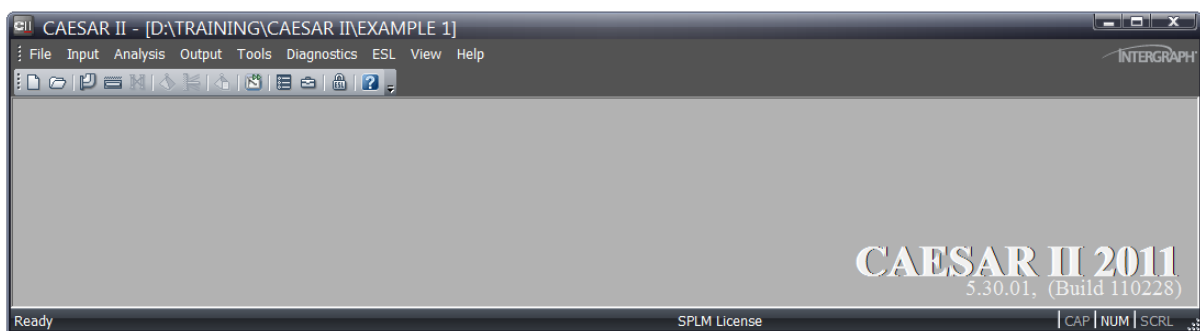
## Introduction

CAESAR II is pipe stress analysis software which uses beam theory to evaluate piping systems to numerous international standards. CAESAR II is **not** Finite Element Analysis (FEA) software, but instead uses a stick model built up of elements connected by nodes.

This course will introduce CAESAR II and demonstrate various modelling and analysis methods in order to evaluate and correct piping systems.

## Interface

When starting CAESAR II, the Main Window appears. This is the window where all tasks are started from. This includes opening/creating an input file, reviewing Load Cases, reviewing results or accessing any auxiliary modules such as WRC 107/297 processor or the ISOGEN stress isometrics module. All modules open up in their own separate window.



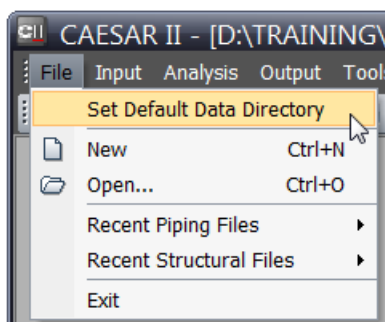
When opening a new file, the file will open, but you will return to the Main Window. You can then choose to go to the Input processor, Output processor or the results for this file.

These modules (and other auxiliary modules) and their interfaces will be introduced as they occur throughout the training.

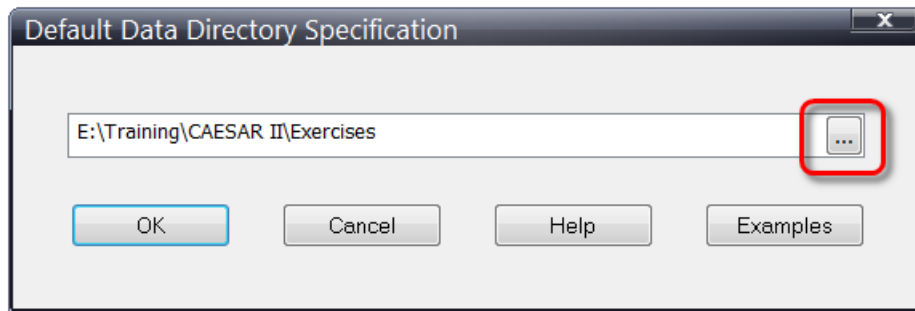
## Default Data Directory

CAESAR II has the option to specify the default working directory – that is all files working with will be saved/opened from this default location. Of course it is still possible to navigate using windows explorer functions, this setting is just the default location when selecting New/Open.

Select File > Set Default Data Directory from the CAESAR II Main window



Click on the ellipsis button at the end of the text field and browse to E:\Training\CAESAR II\Exercises



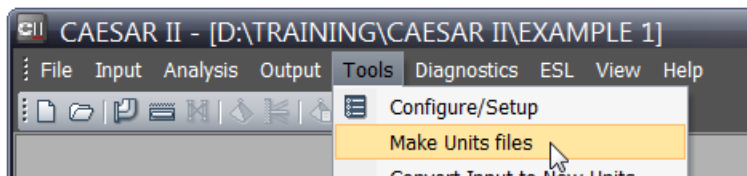
## Units

CAESAR II performs all internal calculations in English units. To enable the entry and review of data in alternative units (such as SI), units files are used by CAESAR II. These units files simply convert the internal CAESAR II English units to the user's preferred unit. Each CAESAR II file (referred to as a "Job File") uses a particular units file which is specified on creation of the job. Files can be converted from one units file to another if required. The units files have the extension \*.FIL and are located in the CAESAR II System directory, or in the same directory as the job file. The file to use is specified in the Configuration file.

### Create Custom Units File

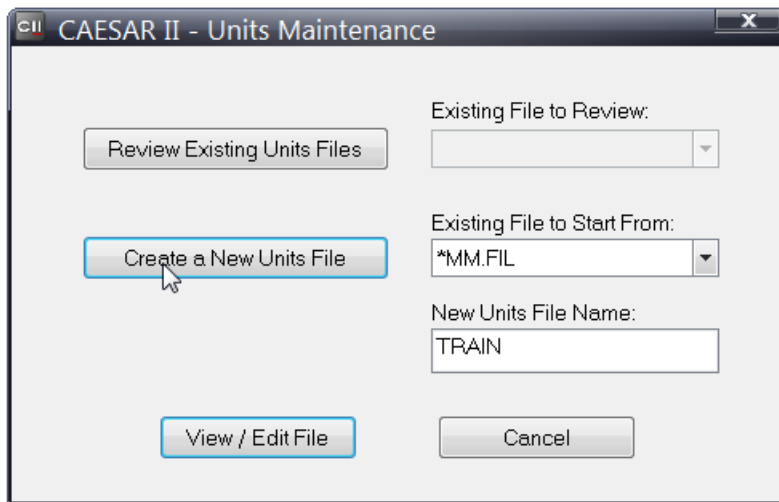
Throughout this course, we wish to use specific units for various parameters such as Pressure, Density etc. As such we require a units file which is different to the supplied default files. So we will create our own CAESAR II units file.

Select Tool > Make Units Files from the Main Window



This allows the creation of new units files, or the review of existing units files, useful if you receive a units file from a colleague and wish to check the units in use in the file.

Choose to Create New Units File and for the template file to use as a start point, select \*MM.FIL. Give the new file a name and click View/Edit file.

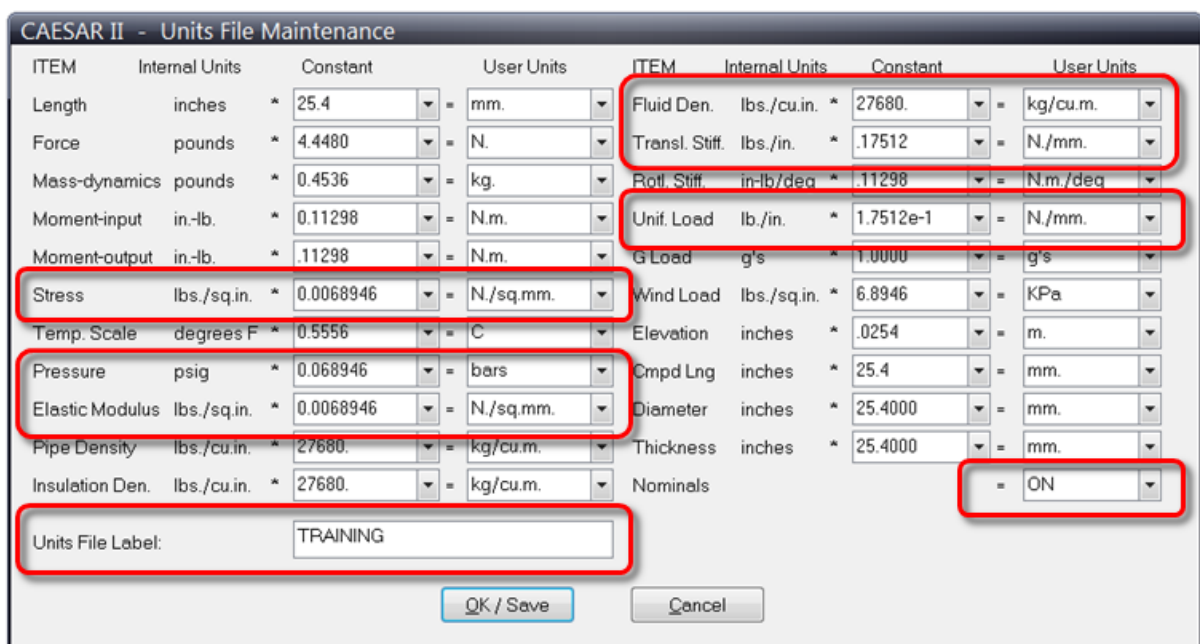


In the Units File dialogue box, change the following units from the MM defaults:

Stress → N/sq.mm.  
 Pressure → bars  
 Elastic Modulus → N/sq.mm.  
 Pipe Density → kg/cu.m.  
 Insul. Density → kg/cu.m.  
 Fluid density → kg/cu.m.  
 Transl. Stiffness → N/mm  
 Uniform Load → N/mm

Ensure Nominals is set to ON. This allows the entry of pipe nominal sizes and schedules into the input, which will be converted to actual diameters and wall thicknesses (e.g. enter 4 into the diameter field and CAESAR II will convert this to 114.3mm).

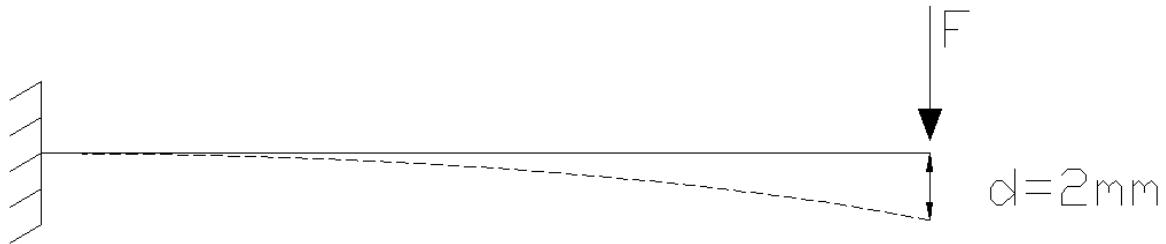
Give the file a label as well to easily identify the file.



## F = Kx Example

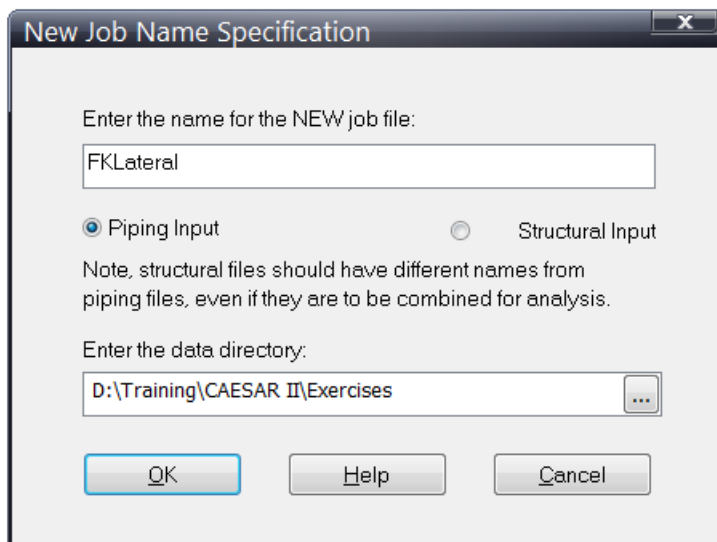
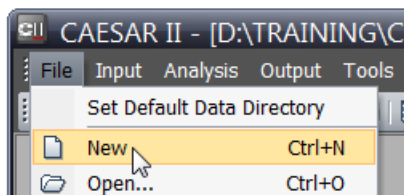
As CAESAR II uses a stick model, and beam theory, it is easy to prove this using a simple cantilever example. This example will introduce the basic modelling methods in CAESAR II and introduce the Input Spread Sheet, Load Case editor and the Output Processor. In addition, we can check the CAESAR II results against some simple hand calculations.

CAESAR II calculates forces using  $F = Kx$ . Using the example below, we will create a simple cantilever model, fixed at one end, and apply a displacement of 2mm at the other end. We can then calculate the force required to generate this 2mm displacement – and see this in the results.



First we will create the model in CAESAR II.

Create a new file in CAESAR II, called FKLateral

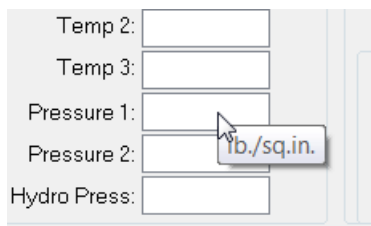


After creating the new job file, the first time it has opened, the units will be displayed to the user for confirmation. You will notice that the units file displayed here for our file is English (CAESAR II default units) **not** the units file we have just created. By default, CAESAR II uses the units file set in the Configuration/Setup as the default file for new jobs (and also as the units to use to display the output results).

ITEM	Internal Units	Constant	User Units	ITEM	Internal Units	Constant	User Units
Length	inches	* 1	= in.	Fluid Den.	lbs./cu.in.	* 1	= lb./cu.in.
Force	pounds	* 1	= lb.	Transl. Stiff.	lbs./in.	* 1	= lb./in.
Mass-dynamics	pounds	* 1	= lbm	Rotl. Stiff.	in-lb/deg	* 1	= in.lb./deg
Moment-input	in.-lb.	* 1	= in.lb.	Unif. Load	lb./in.	* 1	= lb./in.
Moment-output	in.-lb.	* 0.0833333	= ft.lb.	G Load	g's	* 1	= g's
Stress	lbs./sq.in.	* 1	= lb./sq.in.	Wind Load	lbs./sq.in.	* 144	= lb./sq.ft.
Temp. Scale	degrees F	* 1	= F	Elevation	inches	* 0.0833333	= ft.
Pressure	psig	* 1	= lb./sq.in.	Cmpd Lng	inches	* 0.0833333	= ft.
Elastic Modulus	lbs./sq.in.	* 1	= lb./sq.in.	Diameter	inches	* 1	= in.
Pipe Density	lbs./cu.in.	* 1	= lb./cu.in.	Thickness	inches	* 1	= in.
Insulation Den.	lbs./cu.in.	* 1	= lb./cu.in.	Nominals			= ON

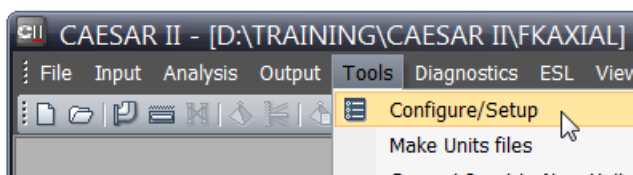
Units File Label:

Click OK on the units review screen and the input spreadsheet will open. To confirm/check the units, hover over any field in the input – the units used in this field will be displayed in the tooltips. For example, we changed the pressure units to bars, but the pressure field displays the units as lb./sq.in.

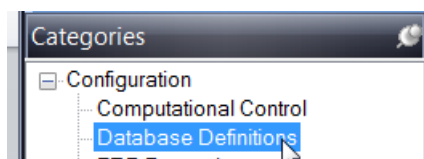


Close the input screen – we will change the units and return to the input with the correct units displayed.

In the CAESAR II main window select Tools > Configure/Setup



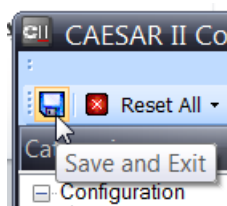
In the window which appears, select Database Definitions from the categories tree on the left.



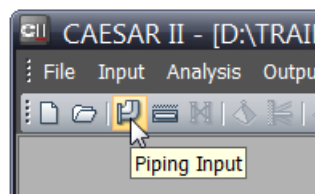
Now change the Units File Name setting to the units file just created.

<b>Databases</b>	
Alternate CAESAR II Distributed Data Path	
Default Spring Hanger Table	Anvil
Expansion Joints	FLEXPATH.JHD
Load Case Template	LOAD.TPL
Piping Size Specification	ANSI
Structural Database	AISC89
Units File Name	<b>TRAIN.FIL</b>
User Material Database Filename	*ENGLISH.FIL
Valve/Flange Files Location	*BAR.FIL
Valves and Flanges	*DEUTSCH.FIL
<b>ODBC Settings</b>	
Append re-runs to existing data	*FRANCE.FIL
Enable data export to ODBC compliant databases	*MM.FIL
ODBC Database File Name	*SI.FIL
	*TUV.FIL

Save and exit.



Return to the Piping Input. Again, the units file to be used will be displayed – this should now be your custom units file.



Verify that the correct units are in use via the tooltips

Temp 1:	<input type="text"/>
Temp 2:	<input type="text"/>
Temp 3:	<input type="text"/>
Pressure 1:	<input type="text"/>
Pressure 2:	<input type="text"/>
Hydro Press:	<input type="text"/>

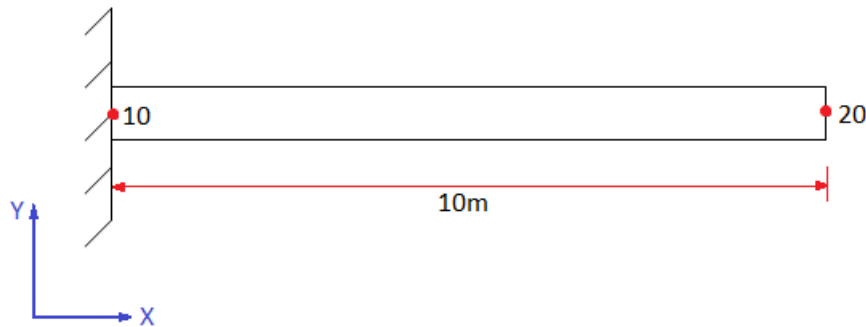
A tooltip labeled 'bars' is displayed over the Pressure 2 input field.

## Model Input

We will now create the simple cantilever model and apply a 2mm displacement at the free end.

The model will be as follows:

One element 10m in length going from node 10 to node 20 in the X direction, anchored at one end. 8" pipe with Standard wall thickness.



The input spread sheet will have defaulted to nodes 10 to 20, so simply enter 10000 in the DX field. We are in mm units already.

Enter the pipe diameter and wall thickness – this is 8" NS and STD wall thickness. As we have "nominals" set to ON, simply type in 8 in the diameter and hit enter. The actual OD for 8" pipe will be inserted. Repeat for the wall thickness, simply type in "S" and press enter.

>>	
From: 10	<input type="checkbox"/> Name
To: 20	
>>	
DX: 10000.000 mm	
DY:	
DZ:	
<input type="checkbox"/> Offsets	
>>	
Diameter: 219.0750	
Wt/Sch: 8.1788	

Now we must fill in the pipe properties. We need to know the material properties to carry out the analysis. Select A106 – B from the list of materials. Notice that all materials have a number to identify it, you can simply type in the material number here - in the case of A106-B this is 106.

Selecting the material will fill in the Elastic modulus and Poisson ratio and various material allowables under the Allowable Stress area, depending on the design code selected (B31.3 default).

Material: (106)A106 B

☒ Allowable Stress

Elastic Modulus (C): 2.0339E+008

Elastic Modulus (H1): 2.0340E+008

Elastic Modulus (H2): 2.0339E+008

Elastic Modulus (H3): 2.0339E+008

Poisson's Ratio: 0.2920

Code: B31.3

SC: 137.892

SH1:	137.892	F1:	
SH2:	137.892	F2:	
SH3:	137.892	F3:	
SH4:	137.892	F4:	

That is our pipe itself. We now need to anchor it at one end (node 10) and apply a displacement at the other end (node 20).

Place the anchor by double clicking the Restraints check box. All the check boxes shown in the middle column on the spread sheet must be double clicked to check/uncheck.

☒ Restraints

☐ Displacements

☐ Hangers

☐ Flange Checks

☐ Nozzle Flex.

☐ Nozzle Lmt Check

To define a restraint you must specify a minimum of the node that the restraint will be attached to, plus the type of restraint. Press F1 for more information on the different restraint types. We need an anchor, so select ANC and locate it at node 10.

Node: 10 CNode:

Type: ANC Gap:

Stif: Mu:

Now we will apply the 2mm displacement at the opposite end. Double click the Displacements check box to apply a displacement.

☒ Restraints

☒ Displacements

☐ Hangers

☐ Flange Checks

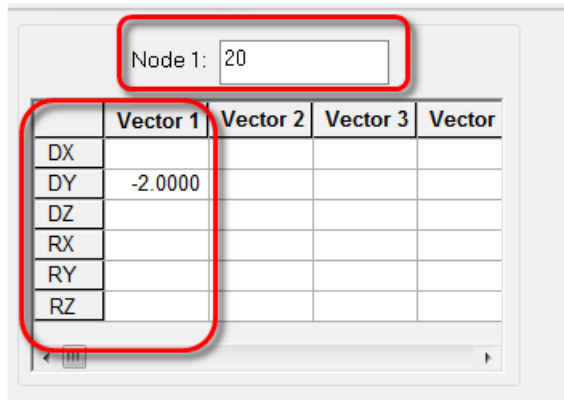
☐ Nozzle Flex.

☐ Nozzle Lmt Check



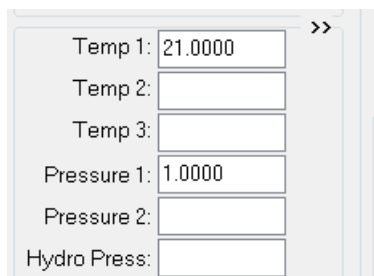
Specify the displacement at node 20, and specify a 2mm displacement downwards in the Y direction – i.e. enter -2 in the DY row.

Leave the remaining rows empty – do not specify 0. Specifying 0 fixes the node in the specified direction. Entering 0 in each row would be the same as an anchor. Leaving the values blank leaves the remaining directions free.



	Vector 1	Vector 2	Vector 3	Vector
DX				
DY	-2.0000			
DZ				
RX				
RY				
RZ				

Finally to complete the analysis we must specify a design temperature and pressure. In our case these are not really relevant as we are only concerned with displacement, so just enter 21°C in T1 and 1bar in P1 fields.

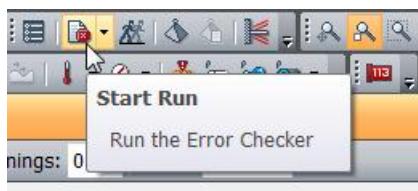


Temp 1:	21.0000
Temp 2:	
Temp 3:	
Pressure 1:	1.0000
Pressure 2:	
Hydro Press:	

The file can now be analysed.

Before analysis the input must be error checked in order to identify any issues which may prevent the analysis running (such as specifying both an anchor and an applied displacement at the same point), or anything which may provide incorrect results (such as Stress Intensification factors not present at a geometric intersection).

Run the error checker to check the model.



You should see only one note in the error checker report – the C of G. This can be useful for identifying problems such as incorrect densities applied – giving an incorrect weight for example.

Errors and Warnings																																																	
Errors: 0 Warnings: 0 Notes: 1																																																	
	Message Type	Message Number	Element/Node Number	Message Text																																													
1	NOTE			<p style="text-align: center;">CENTER OF GRAVITY REPORT</p> <table> <tr> <th></th><th>Total Wght</th><th>X cg</th><th>Y cg</th><th>Z cg</th></tr> <tr> <th>N.</th><th></th><th>mm.</th><th>mm.</th><th>mm.</th></tr> <tr> <td>Pipe</td><td>4162.5</td><td>5000.0</td><td>0.0</td><td>0.0</td></tr> <tr> <td>Insulation</td><td>0.0</td><td>0.0</td><td>0.0</td><td>0.0</td></tr> <tr> <td>Refractory</td><td>0.0</td><td>0.0</td><td>0.0</td><td>0.0</td></tr> <tr> <td>Fluid</td><td>0.0</td><td>0.0</td><td>0.0</td><td>0.0</td></tr> <tr> <td>Pipe+Ins+Rfrty</td><td>4162.5</td><td>5000.0</td><td>0.0</td><td>0.0</td></tr> <tr> <td>Pipe+Fluid</td><td>4162.5</td><td>5000.0</td><td>0.0</td><td>0.0</td></tr> <tr> <td>Pipe+Ins+Rfrty+Fld:</td><td>4162.5</td><td>5000.0</td><td>0.0</td><td>0.0</td></tr> </table>		Total Wght	X cg	Y cg	Z cg	N.		mm.	mm.	mm.	Pipe	4162.5	5000.0	0.0	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	4162.5	5000.0	0.0	0.0	Pipe+Fluid	4162.5	5000.0	0.0	0.0	Pipe+Ins+Rfrty+Fld:	4162.5	5000.0	0.0	0.0
	Total Wght	X cg	Y cg	Z cg																																													
N.		mm.	mm.	mm.																																													
Pipe	4162.5	5000.0	0.0	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	4162.5	5000.0	0.0	0.0																																													
Pipe+Fluid	4162.5	5000.0	0.0	0.0																																													
Pipe+Ins+Rfrty+Fld:	4162.5	5000.0	0.0	0.0																																													

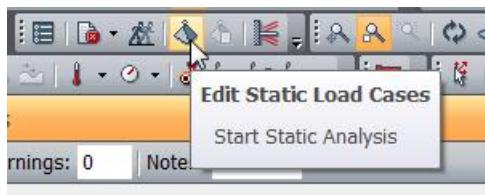
If you receive anything other than this C of G, review the model for any issues. A common error on this exercise is the following:

Errors and Warnings				
Errors: 1 Warnings: 0 Notes: 0				
	Message Type	Message Number	Element/Node Number	Message Text
1	ERROR	57E	10-20	<p>At node 10 there is a DISPLACEMENT and a RESTRAINT specified. This results in a numerically overspecified boundary condition. The user should either give the displacements of the point, or restrain the point, but not both. If modelling initial thermal movements at a restraint, define some unique connecting node for the restraint, and specify the displacement for the connecting node.</p>

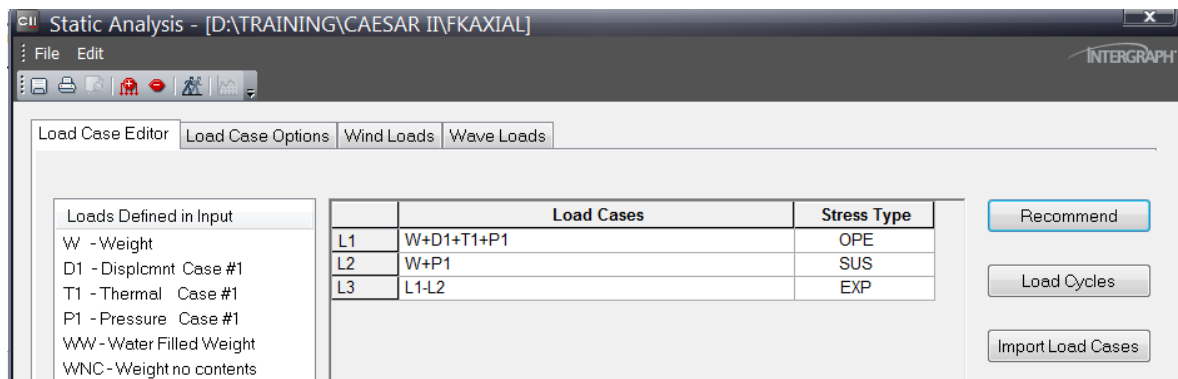
This indicates that the displacement and the anchor have been specified at the same location. Check that the Anchor is specified at Node 10 and the displacement is specified at Node 20.

## Load Case Editor

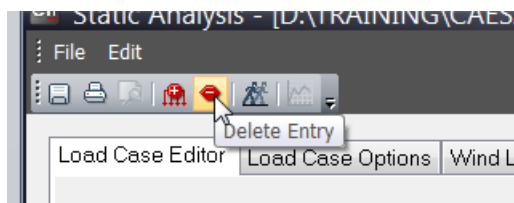
Once the error check is successful, we can create load cases to analyse the system. Access the load case editor. This button is only available after a successful error check.



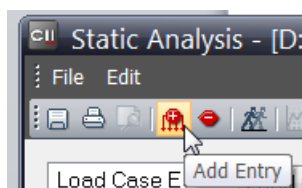
The load case editor will be shown.



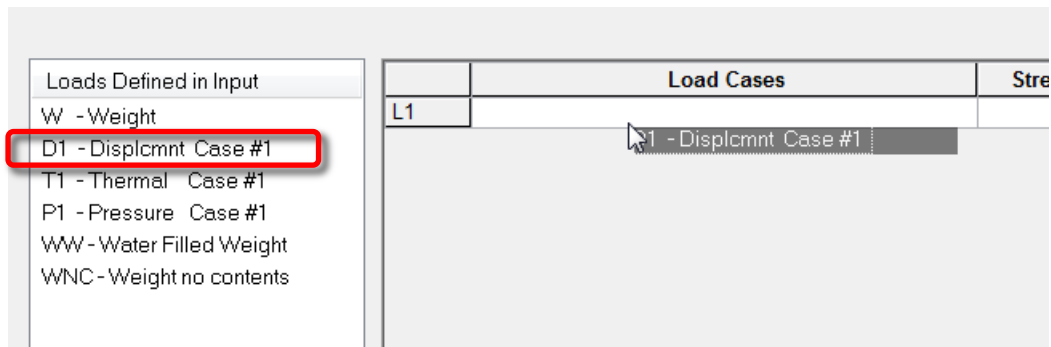
The default load cases are the Operating, Sustained and Expansion cases, as required by the design codes such as B31.3. Remove all these load cases, as we are only concerned with the displacement.



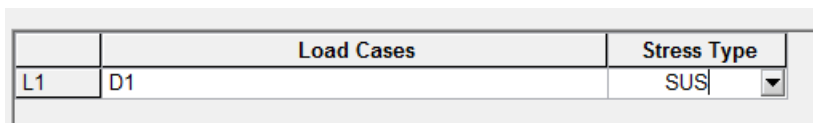
Add one new row.



Into the load case we can add any of the loads defined in the input into the load case. As we are only concerned with the displacement, drag in D1 – Displacement Case #1 into the L1 row.



Also select the stress type as SUS(tained).



The analysis will now take into account only the displacement reaction.

Before we analyse the piping system, let us first perform the hand calculation in order to check.

### Hand Calculation

As we know,  $F = Kx$

$F = \text{Force}$

$K = \text{Stiffness}$

$x = \text{Displacement}$

The stiffness K is

$$K = \frac{3EI}{L^3}$$

$$I = \frac{\pi}{64}(D^4 - d^4) \quad ; \text{Where } D = \text{Pipe OD and } d = \text{Pipe ID}$$

E being Modulus of Elasticity and L being the length.

So if we wish to know what force is required to displace the cantilever 2mm, we can calculate this quite easily.

$$E = 203 \times 10^3 \text{ N/mm}^2$$

$$L = 10,000 \text{ mm}$$

$$D = 219.08 \text{ mm}$$

$$d = 202.72 \text{ mm}$$

$$K = \frac{3 \times 203E3 \times \frac{\pi}{64}(219.08^4 - 202.72^4)}{(10E3)^3}$$

$$K = 18.379 \text{ N/mm}$$

So for a 2mm displacement, the force required is

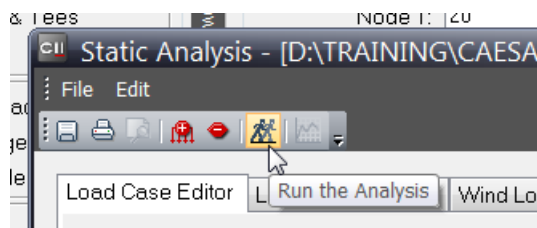
$$F = Kx$$

$$F = 18.379 \times 2$$

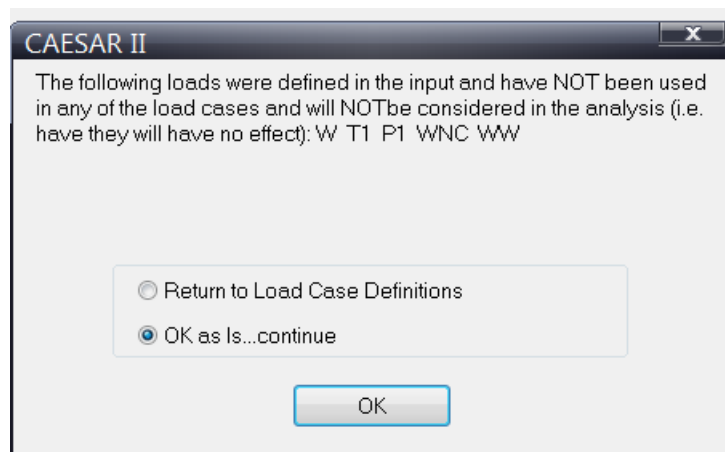
$$F = 36.758 \text{ N}$$

## Output Processor

Back in CAESAR II, run the analysis by clicking on the “Running Man” icon from within the load case editor.



You will see the following message explaining that certain loads have been defined in the model but are not included in any of the load cases to be analysed – this is OK in our case, but can serve as a useful warning if you have many loads/load cases defined. Select OK as is...Continue and click OK to analyse.

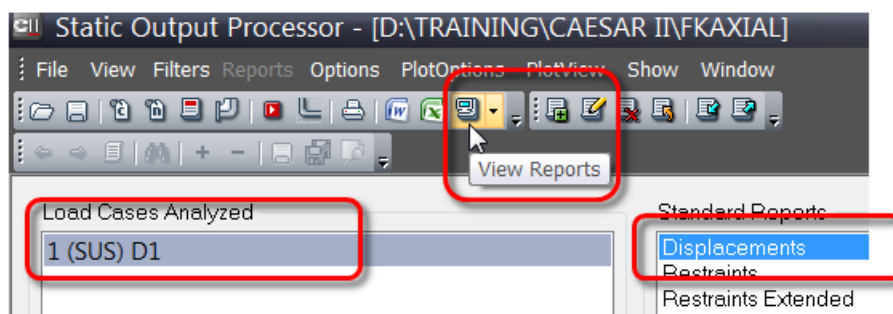


Once the analysis is complete, the Output Processor will be shown. We can view various results for any load case from here, plus general model reports such as the Input Echo. These reports can be viewed on screen, or output to Word/Excel/Text or straight to a printer.

In addition Custom report templates can be created, and any available report can be selected and added to the Output viewer Wizard, and exported/viewed to create/view a comprehensive report very quickly.

For now we will just check the displacement at node 20 to verify that it is 2mm, and the force at node 20 to check against our hand calculation.

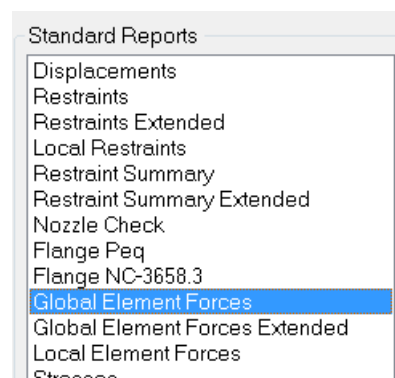
Select the load case (SUS) D1 and the Displacements standard report and click to show on screen:



The DY at Node 20 is -2mm, as we specified.

(1)Displacements				
CAESAR II 2011 SP1 Ver.5.30.01, (Build 110228)				
Job: D:\TRAINING\CAESAR II\FKAXIAL				
Licensed To: Seat -- ID #51				
DISPLACEMENTS REPORT: Nodal Movements				
CASE 1 (SUS) D1				
NODE	DX mm.	DY mm.	DZ mm.	
10	0.000	-0.000	0.000	
20	0.000	-2.000	0.000	

Now to check the force at node 20; view the Global Element Forces report.

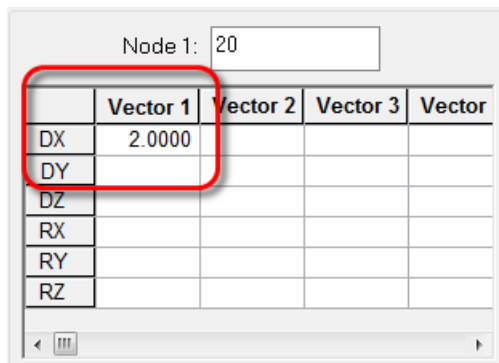


(1)Global Element Forces				
CAESAR II 2011 SP1 Ver.5.30.01, (Build 110228)				
Job: D:\TRAINING\CAESAR II\FKAXIAL				
Licensed To: Seat -- ID #51				
GLOBAL ELEMENT FORCES REPORT: Forces on Elements				
CASE 1 (SUS) D1				
NODE	FX N.	FY N.	FZ N.	
10	0	37	0	
20	0	-37	0	

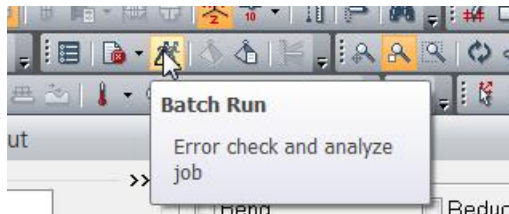
37 N as we calculated.

## Axial

We can repeat this exercise for axial forces. This is a simple change in the model changing the displacement from the Y to the X direction.



The analysis can be quickly re-run in cases where a change such as this has been made by using the Batch Run “Double Running Man” icon. This will run the error checker followed immediately by the analysis (providing there are no Errors).



The force should be as follows:

We are still using  $F = Kx$ , but we are using the Axial stiffness.

$$K = \frac{AE}{L}$$

$A$  = cross Sectional area ( $\text{mm}^2$ )

$E$  = modulus of elasticity ( $\text{N/mm}^2$ )

$L$  = Pipe Length

$$A = \pi \frac{D^2}{4} - \pi \frac{d^2}{4}$$

$$A = \frac{\pi}{4} (219.08^2 - 202.72^2)$$

$$A = 5419.76 \text{ mm}^2$$

Therefore:

$$K = \frac{5419.76 \times 203E3}{(10E3)}$$

$$K = 110,021 \text{ N/mm}$$

So for an axial extension of 2mm, the force required is



$$F = Kx$$

$$F = 110,021 \times 2$$

$$F = 220,043 \text{ N}$$

The CAESAR II results, Global Element forces report should verify this:

(1)Global Element Forces			
CAESAR II 2011 SP1 Ver.5.30.01, (Build 110			
Job: D:\TRAINING\CAESAR II\FKAXIAL			
Licensed To: Seat -- ID #51			
GLOBAL ELEMENT FORCES REPORT: Forces on Elem			
CASE 1 (SUS) D1			
NODE	FX N.	FY N.	FZ N.
10	-220423	0	
20	220423	0	

The forces calculated such as in the previous example produce bending moments throughout the piping system. Bending moment is produced when a Force is applied at a distance –  $M_b = F \times L$

Once the bending moment has been calculated, beam theory is used in order to calculate the stress at this point.

$$\frac{\sigma}{y} = \frac{M}{I}$$

$$\sigma = \text{Stress (N/mm}^2\text{)}$$

$$M = \text{Bending Moment (Nmm)}$$

$$Y = \text{Distance from Neutral axis to outer fibre of beam (mm)}$$

$$I = \text{Moment of Inertia (mm}^4\text{)}$$

$$\frac{\sigma}{y} = \frac{M}{I} \text{ rearranges to } \sigma = \frac{My}{I}$$

$\frac{I}{y}$  is the section modulus Z. So this reduces further to

$$\sigma = \frac{M}{Z}$$

The stresses are calculated using this basic theory and compared to the allowable stresses in the design codes. CAESAR II has many design codes available, all of which have evolved separately over time, thus the way the stresses are calculated for each specific code are slightly different. However, looking at one of the most common piping codes – B31.3 – it can be seen that the equations used are based on the basic bending as detailed above.

### B31.3 Chemical Plant and Petroleum Refinery Piping

#### **Sustained:**

$$S_l = \frac{F_{ax}}{A_m} + \frac{[(i_i M_i)^2 + (i_o M_o)^2]^{1/2}}{Z} + \frac{P d_o}{4t} \leq S_h$$

#### **Expansion:**

$$S_E = \frac{[(i_i M_i)^2 + (i_o M_o)^2 + 4M_T^2]^{1/2}}{Z} \leq S_A = f(1.25S_c + 1.25S_h - S_l)$$

As can be seen, the equations essentially use bending stress  $M/Z$ . The equations are a little more complicated than the basic cantilever example for the following reasons:

- To address piping systems in 3 dimensions
- To address areas in a piping system where particular geometry/components, such as at a branch connection or a bend, can increase the stress, and therefore the likelihood of failure. At these points, the stress is increased by a Stress Intensification Factor (SIF) known as  $i$ . The design codes contain formulae to calculate these SIFs.
- Stresses can also be caused by Pressure and Axial Forces
- The Stresses are categorised into Sustained, Expansion and Occasional, as detailed below.

**Sustained Stress:** This is primary stresses caused by primary loadings such as the weight and pressure of the piping system.

**Expansion Stress:** Expansion stresses are secondary stresses caused by secondary loadings such as the thermal expansion and applied displacements.

**Occasional Stress:** Combines sustained stresses with those produced by an occasional loading such as earthquake or relief valve operation. As these are occasional loads, the allowable can be increased by a scalability factor,  $k$ .  $k$  is usually dependant of the duration or frequency of the occasional load.