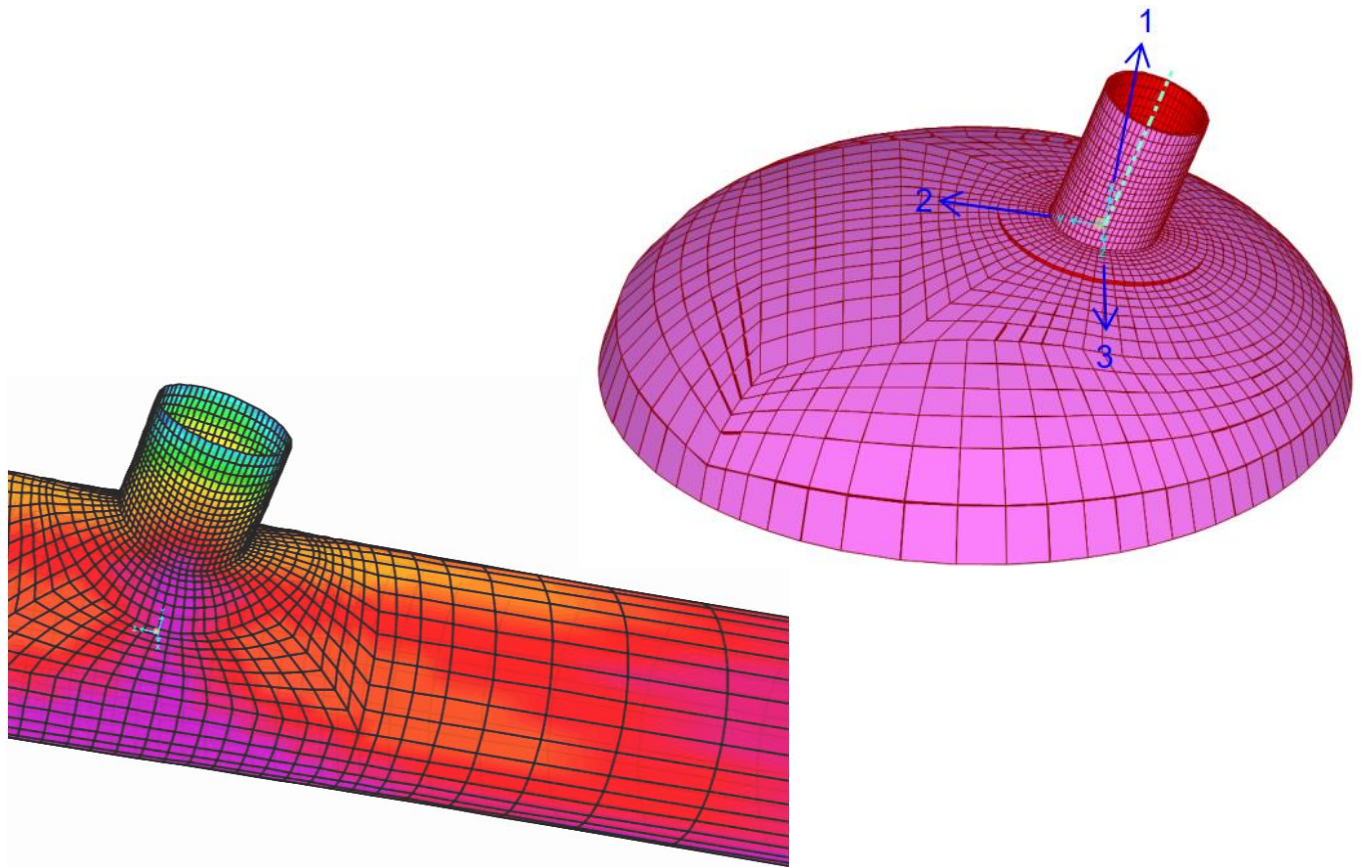




Nozzle FleX Plugin for SAP2000



User Manual

Version 2.0 September 2023

Contents

Introduction.....	3
Applicability	3
Installation	4
Trial license.....	4
Licensing the plug-in	5
Finite element model	6
Orientation of local axes	6
Cylindrical intersections	6
Nozzles on heads	8
User interface	10
Modelling of Cover Weld and Re-pad.....	13
Output stiffnesses.....	13
How stiffness is calculated	14
Stress intensification / stress concentration factors.....	15
Modelling in a pipe stress program	17
Approach 1 – B31J pipe branch with custom flexibility factors and SIFs	17
Approach 2 – Flexible Anchor with optional SIFs	20
CSiPlant.....	20
CAESAR II.....	24
Applying the stress intensification factors – flexible anchor model.....	24
Known Issues	27
Version history.....	27

Introduction

The Nozzle FleX plugin is designed to calculate stiffness and stress intensification of cylindrical intersections for metallic piping and vessels. The plugin has been created for use with SAP2000 v22.1 and onwards. It may be possible to run it with earlier versions – please contact us for assistance. For this or any other queries, please contact support@xtrados.com.

SAP2000 is a registered trademark of Computers and Structures, Inc (CSI).
Xtrados is not affiliated with CSI.

Applicability

The plugin is for fabricated branches, stubs, trunnions and nozzles. The branch may be offset from centre of head / header (hillside nozzle), and may be angled at between 20 degrees and 90 degrees from the cylinder longitudinal axis. The intersection may have a reinforcing ring of constant width.

The following characteristics and limitations apply

- intersection of two cylinders, or cylinder and head
- each has constant thickness
- the intersection does not have a radius of curvature (such as a sweeplet or welding tee)
- material is isotropic
- circular reinforcement pad or no reinforcement

Therefore the plug-in does not currently cater to

- welding tee fittings
- extruded tees
- trunnions on elbows
- branch on reducer or conical shell
- not recommended for tank nozzles near tank bottom (use API 650 Appendix P).

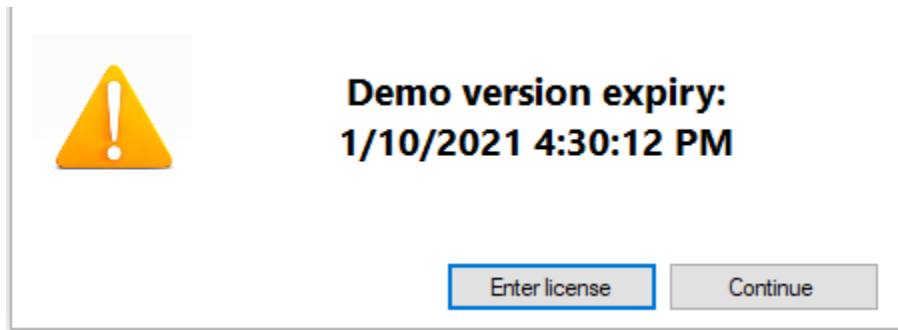
Installation

This is a self-extracting executable zip file - save this to My Docs or any folder of your choosing, then double click to unzip it. This creates a folder called 'Nozzle_flex' containing the plugin files. Note: thanks to Windows' paternal instincts, you may need to Unblock the file first - right click on Nozzle_flex.exe, choose 'Properties', tick 'Unblock' and OK. Any troubles, please contact your IT support.

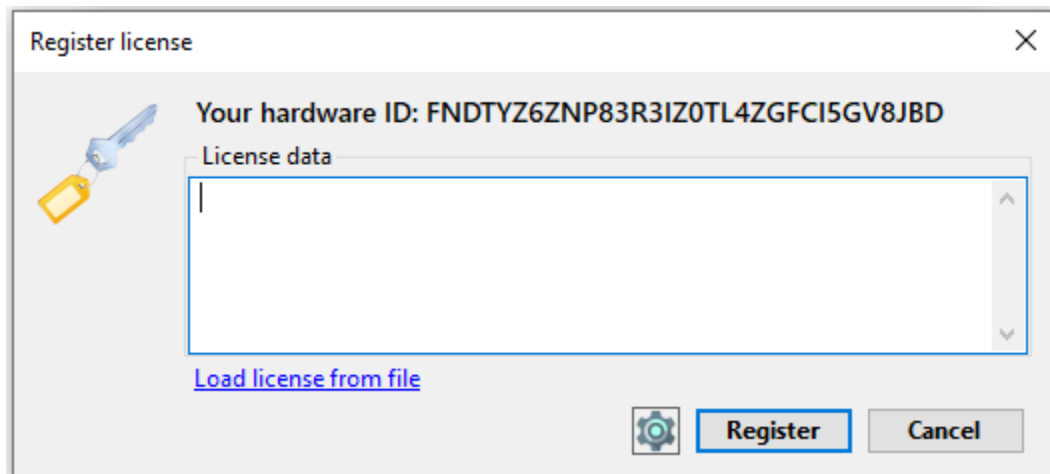
Install the plug-in by first choosing Tools -> Add/Show Plugins from the SAP2000 main menu. Browse to where you unzipped the plugin, select the file 'SAP2000Plugin_Xtrados_Nozzle.dll', then 'Add' and 'OK'. If successful the plugin will now be available directly under 'Tools'.

Trial license

On start-up, you will need to license the plugin. [Email us](#) to receive a trial license. Save this license file somewhere easy to find. In the first dialog box, click on 'Enter license'.



Next, choose 'Load license from file' and browse to the license file. Then click on 'Register'.



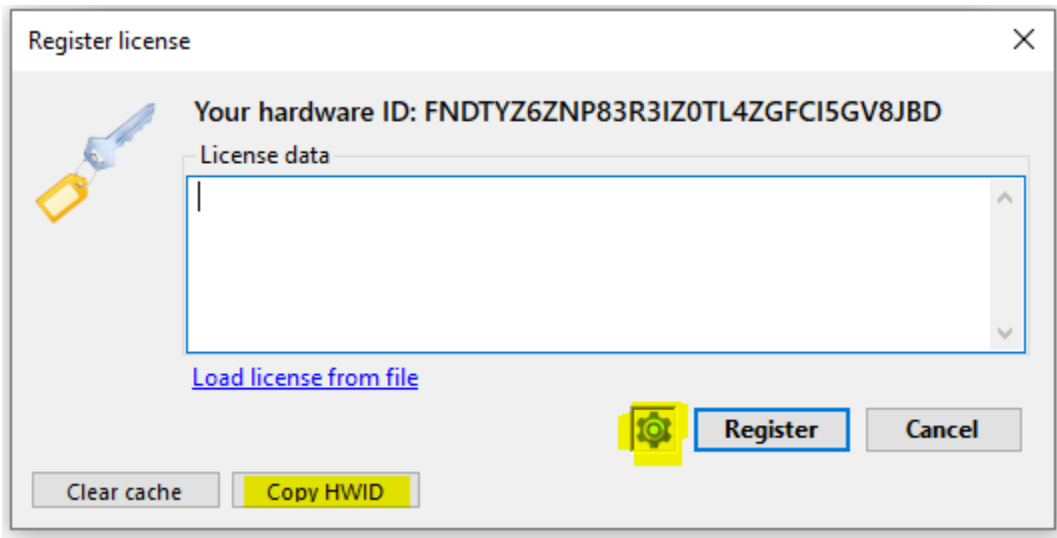
The plug-in trial should now be ready to go. Each time you re-start it you just click on the 'Continue' button in the first dialog.

Licensing the plug-in

If you or your organisation has purchased a license, click on 'Enter license' in the first dialog. Then click on the gear wheel icon, followed by 'Copy HWID'.

Paste the license key into an email and [email to us](#) – we will send your license key by return email.

If you're obtaining licenses for multiple users, please either supply us with the relevant email addresses, or have your users email their HWID codes to you for collation.



We will email your license key back – save this key to a place you can easily find. Select 'Load license from file' in the above dialog box, and browse to the license file. Then click on 'Register'.

In some cases, an update to the plug-in version will require a new license file. In this case we will email new license keys if your license is current.

Finite element model

The finite element model uses SAP2000's thick shell elements. This is a 4-node shell element, which includes membrane and bending stiffness, thick shell / transverse shear displacements, quadratic in-plane displacement and 'drilling' rotational effects. In other words, its accuracy is good even when compared to typical 8-noded quadratic elements.

Note that the shell is modelled at the centre of the pipe wall. Therefore dimensions in the model do not coincide with the outer radius of pipe. Reinforcing pad widths are increased by half the branch thickness to compensate for this.

Orientation of local axes

When referring to nozzle flexibility, we are talking about the stiffness of the header (or vessel / tank) at the point of nozzle attachment. NozzleFlex calculates the following rotational and translational stiffnesses -

- Rotational stiffness calculated in three orthogonal axes
- Translational stiffness calculated in radial direction. Stiffness in the other two (shear) directions are not calculated (considered to be rigid).

The orientation of these axes is as described below.

Cylindrical intersections

Refer to Figure 1a. A right-angled co-ordinate system is defined at the intersection of the shell and the nozzle centreline. The local coordinate system axes are :

1. Radial to shell. Radial displacement occurs along this axis. Torsional rotation occurs about this axis. Often called Axial direction. However note that this direction is defined with respect to the shell, not the branch.
2. Tangential to shell. Longitudinal rotation (aka in-plane) occurs about this axis.
3. Longitudinal (parallel to shell centreline axis). Circumferential rotation (aka out of plane or out-plane) occurs about this axis.

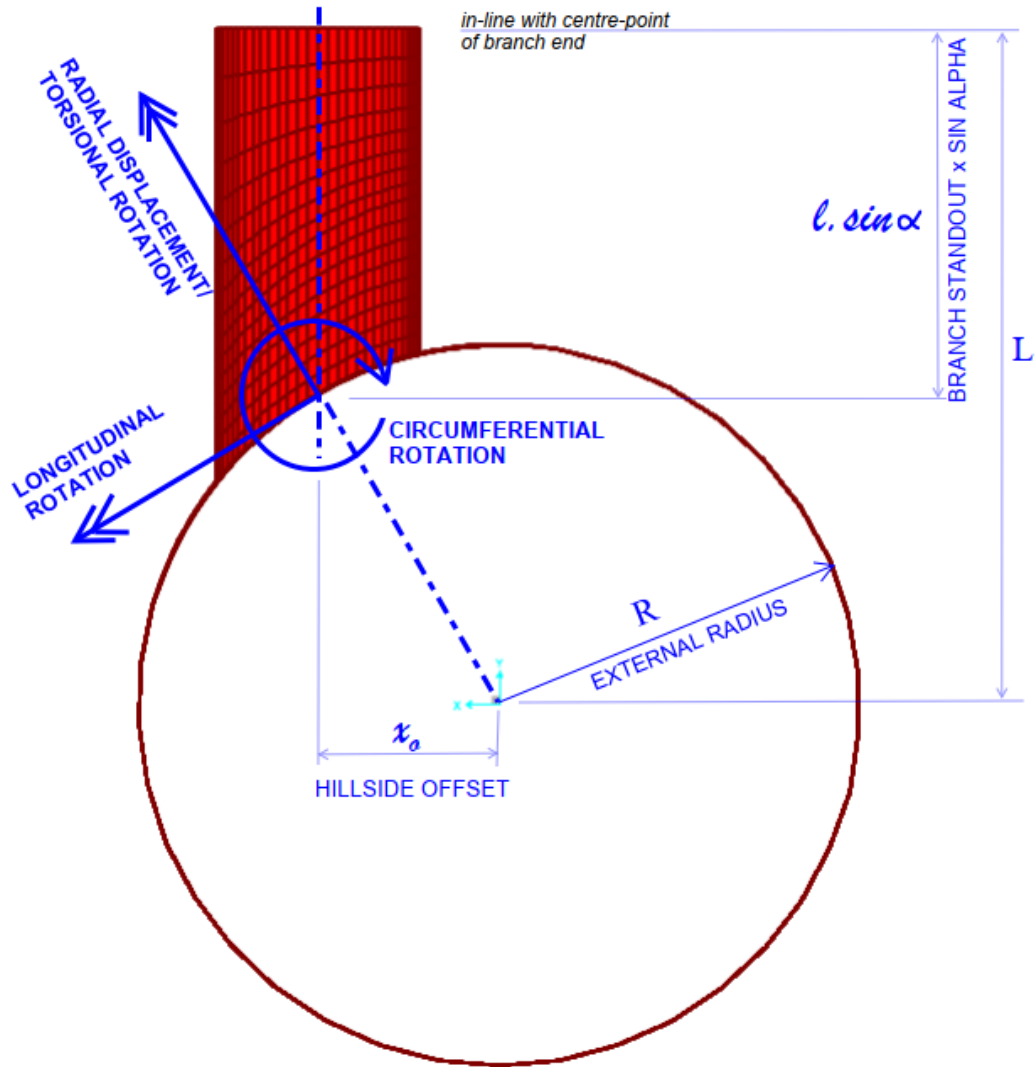


Figure 1a - Branch with hillside offset. Note: standout dimension 'l' is measured along the branch axis, as shown in figure 1b. For the model in Figure 1, branch angle 'alpha' is 90 degrees so $l \cdot \sin \alpha$ is equal to l . If the branch has a flange or other feature which stiffens it, do not include this length in the standout.

The plug-in takes the Branch Standout as an input. If you have dimension L instead, branch standout can be calculated as follows:

$$l = \frac{L - \sqrt{R^2 - x_o^2}}{\sin \alpha}$$

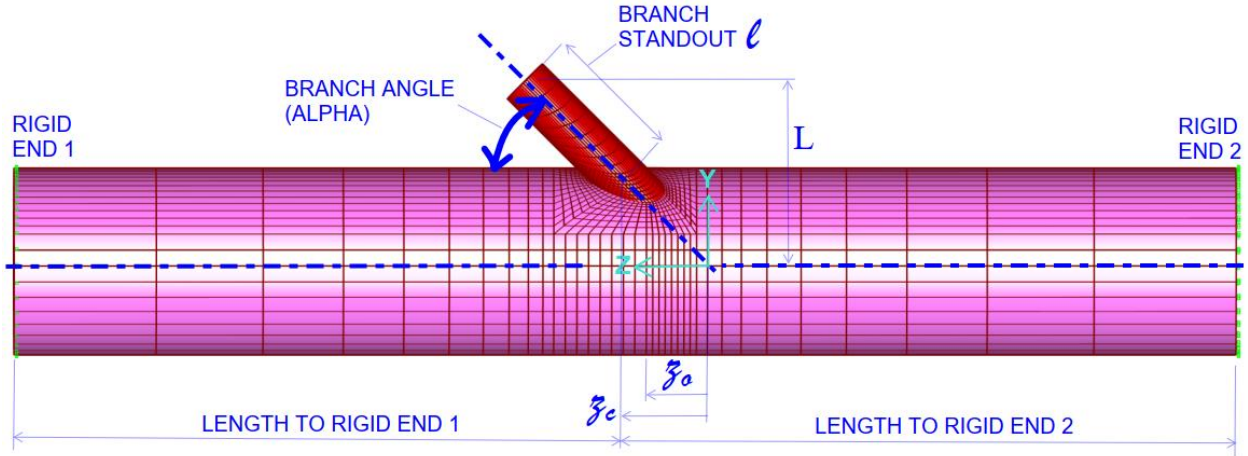


Figure 1b – elevation view of intersection

Rigid ends (1) and (2) are locations at which the shell / header is assumed to be rigid in all 6 degrees of freedom. This would coincide with the nearest flange, vacuum ring, head, large welded support or other feature which significantly stiffens the shell.

When the branch angle is 90°, the lengths to rigid ends (1) and (2) are the (positive) distances from the centre of the branch. For branch angles of less than 90°, these are measured with respect to the theoretical branch centreline / shell intersection at z_c .

For reference, the lowest intersecting point on the branch (point with lowest Y-coordinate) corresponds to the mesh line with constant z-coordinate z_o . These values are calculated as

$$z_o = \frac{\sqrt{r_h^2 - (x_o + r_b)^2}}{\tan \alpha}$$

$$z_c = \frac{\sqrt{r_h^2 - x_o^2}}{\tan \alpha}$$

-where r_h and r_b are the header outside radius and branch outside radius respectively.

The branch standout is measured along the branch length from the theoretical centreline intersection at z_c . This is also the origin point for the local coordinate system seen in Figure 1 a.

Nozzles on heads

Refer to Figure 2. A right-angled co-ordinate system is defined at the intersection of the shell and the nozzle centreline. The local coordinate system axes are :

1. Radial to shell. Radial displacement occurs along this axis. Torsional rotation occurs about this axis. Often called Axial direction. However note that this direction is defined with respect to the shell, not the branch.
2. Tangential to shell and pointing toward centre. Circumferential rotation occurs about this axis.

3. Tangential to shell and orthogonal to axes 1 and 2. Longitudinal rotation occurs about this axis.

Note: the designations “longitudinal” and “circumferential” are not generally used for nozzles on heads. We use them here just to identify which axis is being referred to. For a central nozzle the stiffness and SCF will be the same for both axes.

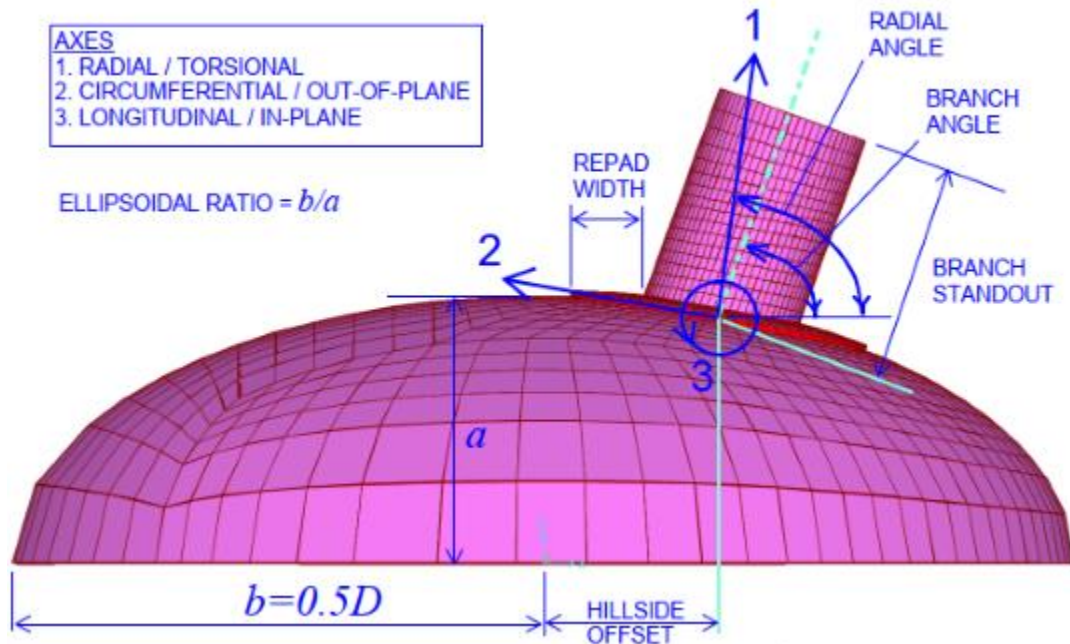


Figure 2 – Nozzle on semi-ellipsoidal head with hillside offset. Note: branch standoff is measured along the branch axis (dotted line). If the branch has a flange or other feature which stiffens it, do not include this length in the standoff.

Reinforcing pads

The width of the reinforcing pad is measured along the curve of each mesh line going outward from the branch. Reinforcing pad widths are increased by half the branch thickness to compensate for the shell model being at the midpoint of the branch wall.

User interface

Note:

1. When the plug-in is open, SAP2000 will be non-responsive (you will get the spinning wheel or equivalent icon when your mouse is over the application). This is expected – you cannot operate SAP2000 when the plugin is open. After closing the plugin, wait until a dialog box from SAP2000 pops up saying 'Plugin closed successfully', and you can again use the program.
2. When the plug-in is running an analysis, both the plug-in and SAP2000 will be non-responsive.

Menu Options

Option	Action or meaning
File -> Save RTF	Generate .rtf file with inputs and outputs. Open with any word processor software.
File -> Close	Close the plugin and return to SAP2000
Options -> Units -> US Units	Use inches, psi, lb/in, ft-lb per unit of rotation
Options -> Units -> Metric	Use mm, MPa, N/mm, Nm per unit of rotation
Options -> Output SIF or SCF? -> SIF	Generate Stress Intensification Factors for pipe stress analysis. Incorporates the user-entered weld SCF (aka Fatigue Strength Reduction Factor) to incorporate the effect of imperfections at the toe of the weld. Note that SCF and SIF are related as follows: $SIF = 0.5 * SCF * (weld\ SCF)$
Options -> Output SIF or SCF? -> SCF	Generate Stress Concentration Factors for use in a separate vessel local stress analysis. Does not include weld SCF (this can be applied in the separate analysis). Currently the plugin outputs the SCF for membrane plus bending only.
Options -> Rotation -> Degrees	Generate rotational stiffnesses per degree
Options -> Rotation -> Rotations	Generate rotational stiffnesses per radian
Options -> About	General information about the plugin
Options -> Help	Open the help file (this document)
Options -> License	Show the license status and option to remove

Graphic User Interface - Cylinders

The screenshot shows the 'SAP2000 Nozzle Flex Plugin' window. It contains a 'File' menu and a list of input parameters for cylinder analysis. Red arrows point from text boxes to specific fields in the GUI.

Parameter	Value	Annotation
Geometry type	Cylinder	Choose Cylinder, SE head, torispherical head or hemi head
Nozzle angle	Custom	
Shell OD (mm)	406.4	
Shell thk. (mm)	9.53	
Branch OD (mm)	219.1	
Branch thk. (mm)	8.18	
Cut through shell?	Y	enter N to model an attachment rather than a nozzle
Re-pad width (mm)	0	reinforcing pad width and thickness
Re-pad thickness (mm)	0	
Branch angle (degrees)	60	angle from 20 to 90
Hillside offset (mm)	0	Distance from centre
Fillet weld leg (mm)	0	Leg length of cover weld
Weld SCF	1.35	Stress conc factor for covering weld*
Distance to rigid end 1 (mm)	1000	Leave blank for automatic flexible lengths; or enter distance to rigid feature like flange
Distance to rigid end 2 (mm)	1000	
Branch standout (mm)	300	
Young's modulus (MPa)	206850	
Poisson's ratio	0.3	
Mesh size (1-5)	3	Defaults to 3. Enter an integer from 1 (finest) to 5 (coarsest). Generally minor impact - use 1 for most accurate SIFs
Run		
Radial stiffness: 166729 (N/mm) SIF_ab: 14.04 In-plane stiffness: 5792400 (Nm/rad.) SIF_ib: 3.51 Out-plane stiffness: 1181694 (Nm/rad.) SIF_ob: 10.98 Torsional stiffness: 40744463 (Nm/rad.) SIF_tb: 1.14		Output stiffnesses and branch SCF's or SIF's

* only used when stress type is SIF

Figure 3a – user interface (cylinders)

Graphic User Interface – Nozzle on Head

The screenshot shows the SAP2000 Nozzle Flex Plugin window with the following input fields and annotations:

- Geometry type:** SE head (dropdown) → Choose Cylinder, SE head, torispherical head or hemi head
- Nozzle angle:** Custom (dropdown) → Choose Custom, Vertical, or Normal to shell
- Shell OD (in.):** 80
- Shell thk. (in.):** 0.375
- Branch OD (in.):** 8.625
- Branch thk. (in.):** 0.322
- Re-pad width (in.):** 0 → reinforcing pad width and thickness
- Re-pad thickness (in.):** 0 → angle from horizontal
- Branch angle (degrees):** 45 → Distance from centre
- Hillside offset (in.):** 0
- Fillet weld leg (in.):** 0.25 → Leg length of cover weld
- Ellipsoidal ratio:** 2 → Ratio of dia. to height
- Branch standout (in.):** 12 → Leave blank for automatic flexible length; or enter distance to rigid feature like flange
- Young's modulus (psi):** 30000000
- Poisson's ratio:** 0.3
- Mesh size (1-5):** 3 → Defaults to 3. Enter an integer from 1 (finest) to 5 (coarsest). Generally minor impact - use 1 for most accurate SIFs
- Run** button
- Output section:** Radial stiffness: 358504 (lb/in.), Longit. stiffness: 985353 (ft.lb/rad.), Circum. stiffness: 687932 (ft.lb/rad.), Torsional stiffness: 41845073 (ft.lb/rad.), SCF_ab: 19.5, SCF_ib: 10.17, SCF_ob: 12.1, SCF_tb: 1 → Output stiffnesses and branch SCF's or SIF's

Figure 3b – user interface (nozzle on head)

There are some minor differences in the input fields for each type of head. These are explained in the image which appears to the right of the input window (not shown in figure 3b). You can show or hide the image using the arrow button [>].

Modelling of Cover Weld and Re-pad

If a covering fillet weld leg is set to greater than zero, shell element thicknesses are increased in the weld zone as shown below.

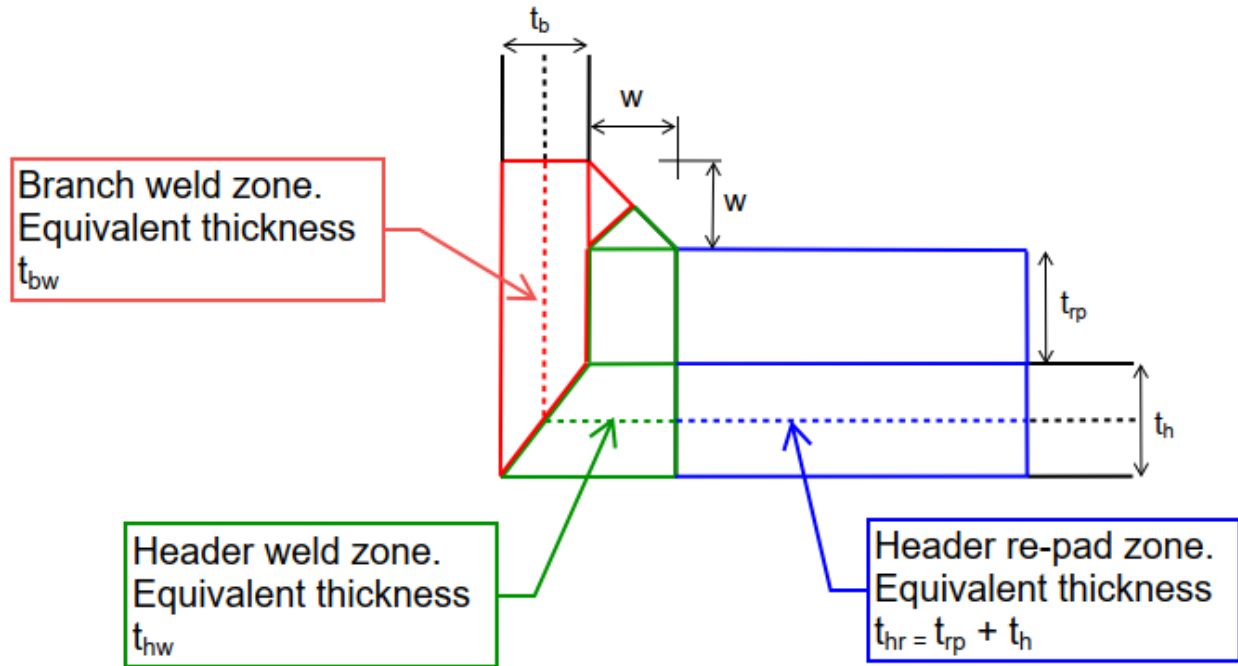


Figure 4– Intersection model cross-section

Equivalent thicknesses are set to give the same areas as those in the cross-section in Figure 4. This gives:

$$t_{bw} = \frac{0.25w^2 + t_b(w + 0.5t_h + t_{rp})}{w + 0.5t_h + t_{rp}}$$

$$t_{hw} = \frac{0.25w^2 + w(t_h + t_{rp}) + 0.5t_h t_b}{w + 0.5t_b}$$

The FEA model is not currently altered or affected by the fillet weld at re-pad edge.

Output stiffnesses

The plug-in calculates the **branch** stiffnesses using the axes shown in Figure 1. Stiffnesses are not calculated for bending of the header / shell itself. These stiffnesses can be used in a pipe stress program to model a nozzle with stiffness, or to define custom branch flexibility factors.

Branch flexibility factors as defined in ASME B31J can be computed from rotational stiffnesses as follows. These equations are for angle in radian units.

$$k_{ib} = \frac{EI_b}{K_{ib}d} \quad k_{ob} = \frac{EI_b}{K_{ob}d} \quad k_{tb} = \frac{EI_b}{K_{tb}d}$$

where:

$k_{(ib,ob,tb)}$ is the branch flexibility factor for in-plane bending, out-of-plane bending and torsion respectively

$K_{(ib,ob,tb)}$ is the branch rotational stiffness for in-plane bending, out-of-plane bending and torsion respectively.

E is Youngs Modulus of Elasticity.

I_b is the branch moment of inertia

d is the branch mean diameter

While B31J does not define an axial translation flexibility factor, one could in theory be defined similarly to above, using branch area A_b rather than I_b .

You can get the flexibility factors by generating the RTF output report from the File menu. This report can then be opened in MS Word or Wordpad.

How stiffness is calculated

To avoid including the flexibility of the beam elements to which the branch attaches, the first model created is a simple beam model of the intersection. Subsequently a shell model of the intersection is created. Deflections and rotations calculated in the beam model are subtracted from the equivalent shell model deflections and rotations before calculating stiffnesses.

Both rigid ends are fully fixed around their circumference for these calculations.

Stress intensification / stress concentration factors

The plug-in calculates factors for loads applied to the branch leg (not for loads applied to the vessel or header). Results are given based on the highest principal stress (for torsion we instead calculate the maximum shear stress). This stress may be either in the header or the branch – this does not affect the calculation.

As the shell-type finite elements are located in the centre of the branch and header walls, the plugin ignores stresses within one half-thickness of the intersection and within the weld zone (the red and green areas in Figure 4).

Stress concentration factors

$$i_i = \frac{2S_i I_b}{M_i D_b} \quad i_o = \frac{2S_o I_b}{M_o D_b} \quad i_a = \frac{S_a A_b}{F_a} \quad i_t = \frac{2S_t J_b}{M_t D_b}$$

where:

$S_{(i,o,a)}$ is the maximum principal stress calculated for pure in-plane bending, out-of-plane bending and axial (radial)¹ force respectively.

S_t is the maximum shear stress calculated for torsion

$M_{(i,o,t)}$ is the applied in-plane bending moment, out-of-plane bending moment, torsion respectively.

$i_{(i,o,a,t)}$ is the branch stress concentration factor for in-plane bending, out-of-plane bending, axial (radial)¹ force and torsion respectively. In the plug-in these are denoted as **SCF_ib**, **SCF_ob**, **SCF_ab** and **SCF_tb** respectively.

F_a is the applied axial (radial) force.

I_b is the branch moment of inertia

J_b is the branch polar moment of inertia, $=2I_b$

D_b is the branch external diameter

Currently the plugin calculates stress concentration factors for membrane plus bending stresses (not membrane stresses only).

Note:

Welds are known to increase fatigue susceptibility via the creation of small imperfections and stress raisers. For instance, see ASME VIII Division 2 Tables 5.11 and 5.12 for values termed Weld Surface Fatigue Strength Reduction Factors.

If analysing vessel stresses (eg in a manual calculation or spreadsheet) you would apply a separate factor to account for this effect as applicable.

¹ The B31 code terminology 'axial' is used here. Note that the SIF and stiffness calcs are with respect to the direction radial to the vessel or header wall at point of branch to shell intersection. This does not necessarily coincide with the branch axis.

Stress intensification factors

Stress intensification factors as used in B31 codes are based on one half of the true stress concentration factor. Thus the plugin uses the following formulae to calculate the SIF's, with loads applied in four separate load cases:

$$i_i = \frac{S_i I_b k_w}{M_i D_b} \quad i_o = \frac{S_o I_b k_w}{M_o D_b} \quad i_a = \frac{S_a A_b k_w}{2 F_a} \quad i_t = \frac{S_t J_b k_w}{M_t D_b}$$

where:

k_w is the weld stress concentration factor. Welds are known to increase fatigue susceptibility via the creation of small imperfections and stress raisers. The FEA model does not account for these stress raisers. To generate a suitable SIF for use in a pipe stress program, we therefore include the weld stress concentration factor input field. The factor defaults to 1.35, which is a typical value used in industry. For additional conservatism or for vessel calculations you may wish to consider higher values such as those given in ASME VIII Division 2 Tables 5.11 and 5.12.

other symbols – as defined in the prior section.

In line with the physical testing methods outlined in ASME B31J, only one end of the header is fixed for SIF calculations on cylinders. Under the naming terminology in Figure 1b, Rigid end 2 is fixed while Rigid end 1 is free. However, note that the cross section at the free end is still constrained to deform in a uniform manner, ie the end is rigid but unrestrained.

The user can consider the effect of nearby flanges or other stiffeners on a cylindrical header by setting the 'Distance to rigid end X' as required. A rigid or stiffened section in proximity to the intersection on one or both sides tends to reduce stress intensification while increasing stiffness.

When adopting the default flexible length (leaving 'Distance to rigid end X' blank), the end is generally far enough away to give conservative SIFs. To see the length values assigned in such cases, generate the RTF output report from the plug-in menu.

Note that ASME B31J provides SIF formulae for cylindrical intersections which are suggested where the following are true:

- *Branch pipe is within 5 degrees of perpendicular to run*
- $R/T \leq 50$
- $r/t \leq 50$

FEA solutions such as Nozzle Flex are appropriate for intersections outside (or within) these parameter ranges, and can account for the effects of hillside branches, re-pad width, cover weld size and varying distance to rigid ends / stiffened sections.

Modelling in a pipe stress program

We suggest one of the following two approaches to use the plug-in outputs in a beam-type pipe stress analysis program. The appropriate method may depend on the program's capabilities.

1. Model as an ASME B31J piping branch with custom flexibility factors and SIFs
2. Model as a flexible anchor with optional SIFs

Approach 1 – B31J pipe branch with custom flexibility factors and SIFs

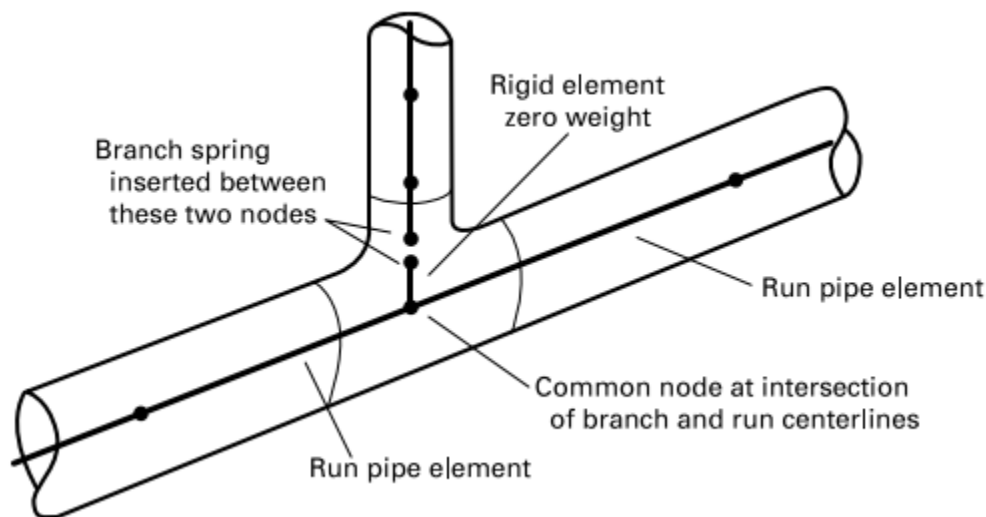


Figure 5 – B31J model with branch flexibility

In this figure taken from ASME B31J (2017), spring(s) are inserted by the program at the point where the branch meets the header. These spring stiffnesses are normally calculated based on the flexibility factors given in B31J. Some pipe stress programs will let you override the flexibility factors. Typically these factors are available for the 3 rotational directions only.

You can get the flexibility factors by generating the RTF output report from the plug-in menu.

Enter only branch flexibilities – do not alter the run flexibility (unless calculated using other means). Assuming the stress program is using the B31J approach, the appropriate springs will be inserted at the branch/shell intersection point.

If you wish to consider radial stiffness, you would also need to manually add a short link or internal anchor on the branch with the appropriate axial stiffness (set to rigid in all other directions).

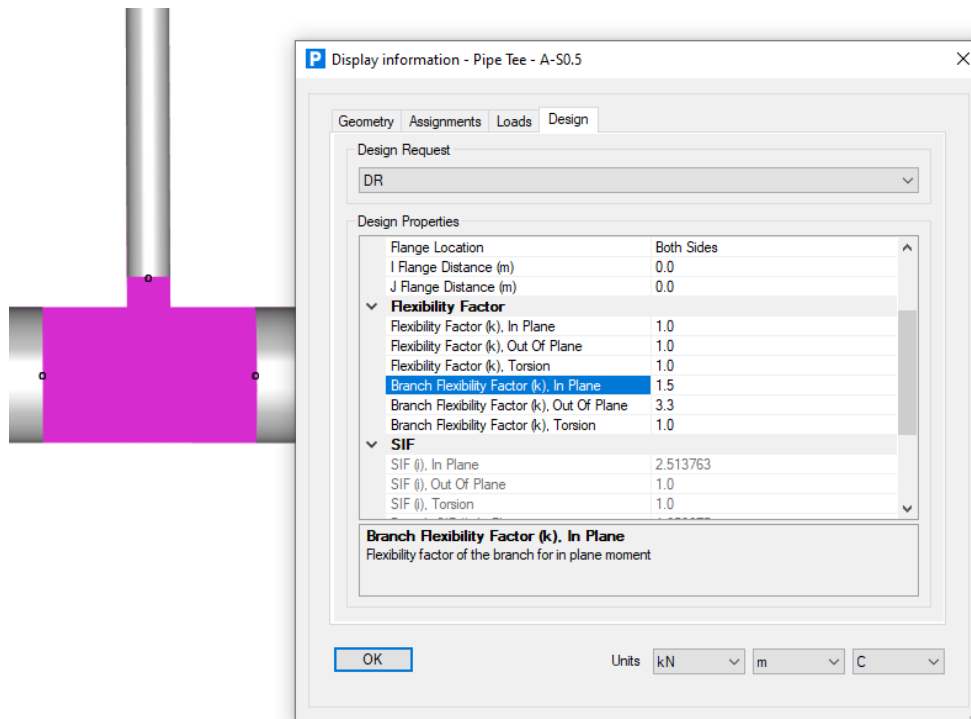


Figure 6 - Defining custom branch flexibility using CSiPlant

Angled or Hillside Branches

The plug-in calculates stiffnesses and SIFs with respect to the axes shown in figures 1a or 2. Therefore it is imperative that the branch be defined radially outward from the centre of header/vessel to the branch intersection point. For an angled branch on a cylinder, the starting point along the header axis is at ' z_c ' in figure 1b. The branch should protrude slightly past the point of flexibility insertion (e.g. the mid or outer radius of header) before turning to the branch pipe orientation. Do not include a bend where the branch pipe turns at this location.

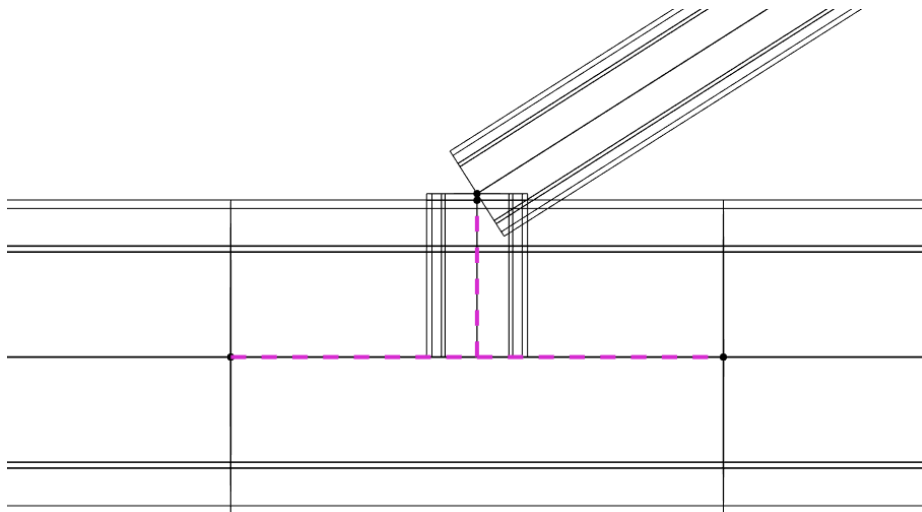


Figure 7 -Defining branch flexibility on an angled branch (CSiPlant)

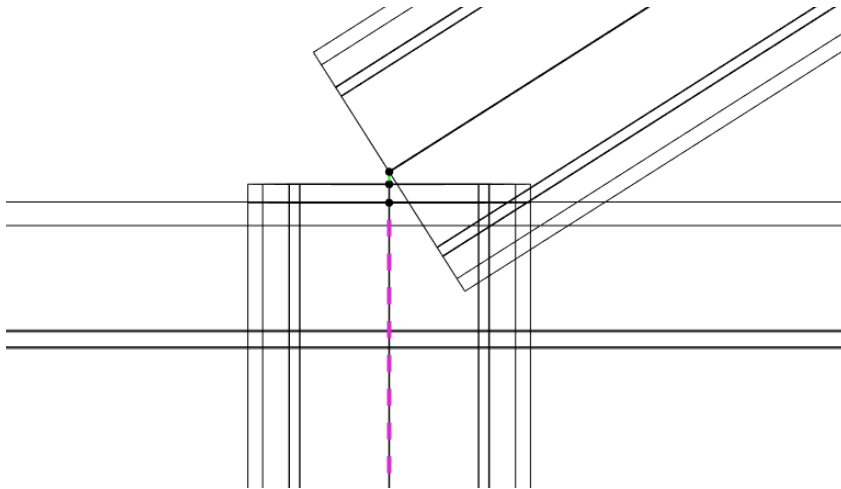
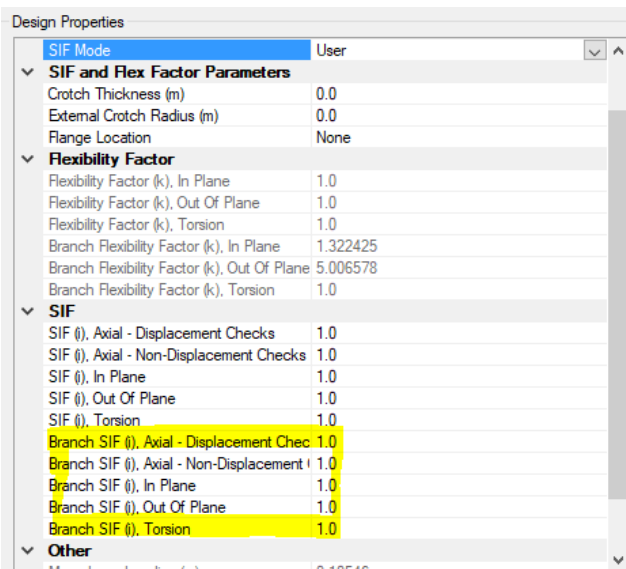


Figure 8 -Defining branch flexibility on an angled branch – with additional flexible link to capture radial stiffness (CSiPlant)

Applying the stress intensification factors – B31J style model

Most pipe stress programs will let you specify custom stress intensification factors. Specify these for the branch only – do not override the header SIFs unless otherwise calculated. The branch SIFs are intended for use with the section modulus $Z = 2I_b/D_b$, rather than the 'equivalent section modulus' Z_e as used in past editions of ASME B31.3 and B31.1.



User SIF entry in CSiPlant. For the 'non-displacement' axial SIF you can enter 0.75 times the axial SIF, or otherwise adjust as suggested in the B31J 2017 errata.

Just as with the branch flexibilities, SIFs calculated by the plug-in are with respect to the axis directions in Figure 1a. Therefore hillside and angled branches should be defined as discussed above.

Approach 2 – Flexible Anchor with optional SIFs

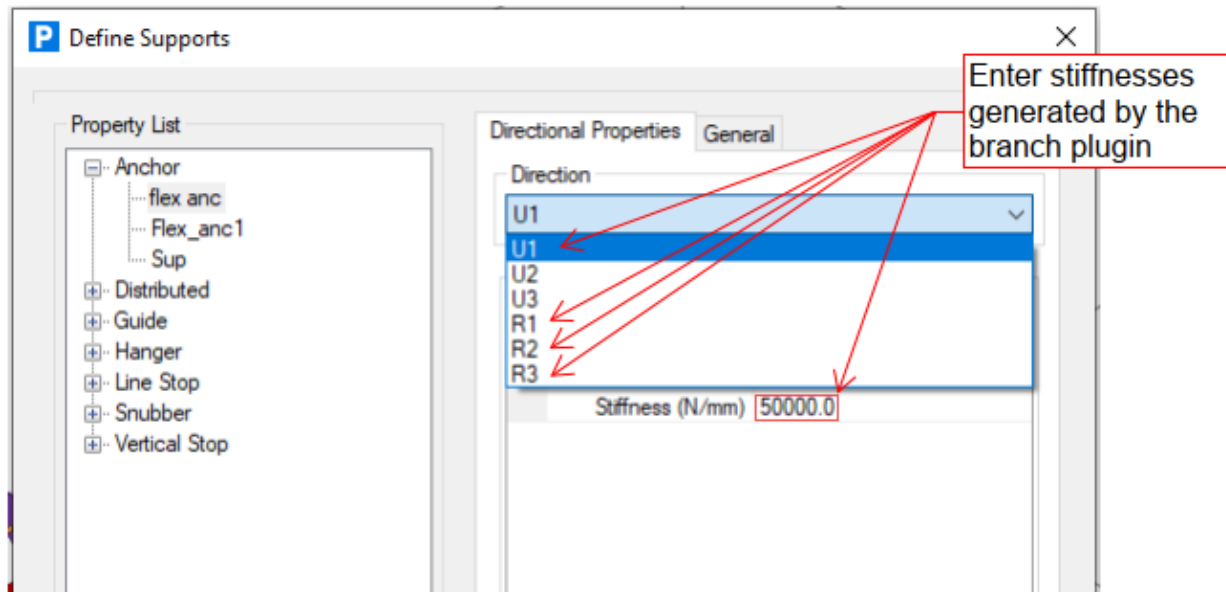
Under this approach, we model a rigid element from the centre of header to the midpoint of the shell wall at the centreline intersection. To ensure that the element is radial, it should start from the same axial coordinate as the centreline intersection (i.e. ' z_c ' in Fig. 1b). Examples follow for CSiPlant and CAESAR II.

CSiPlant

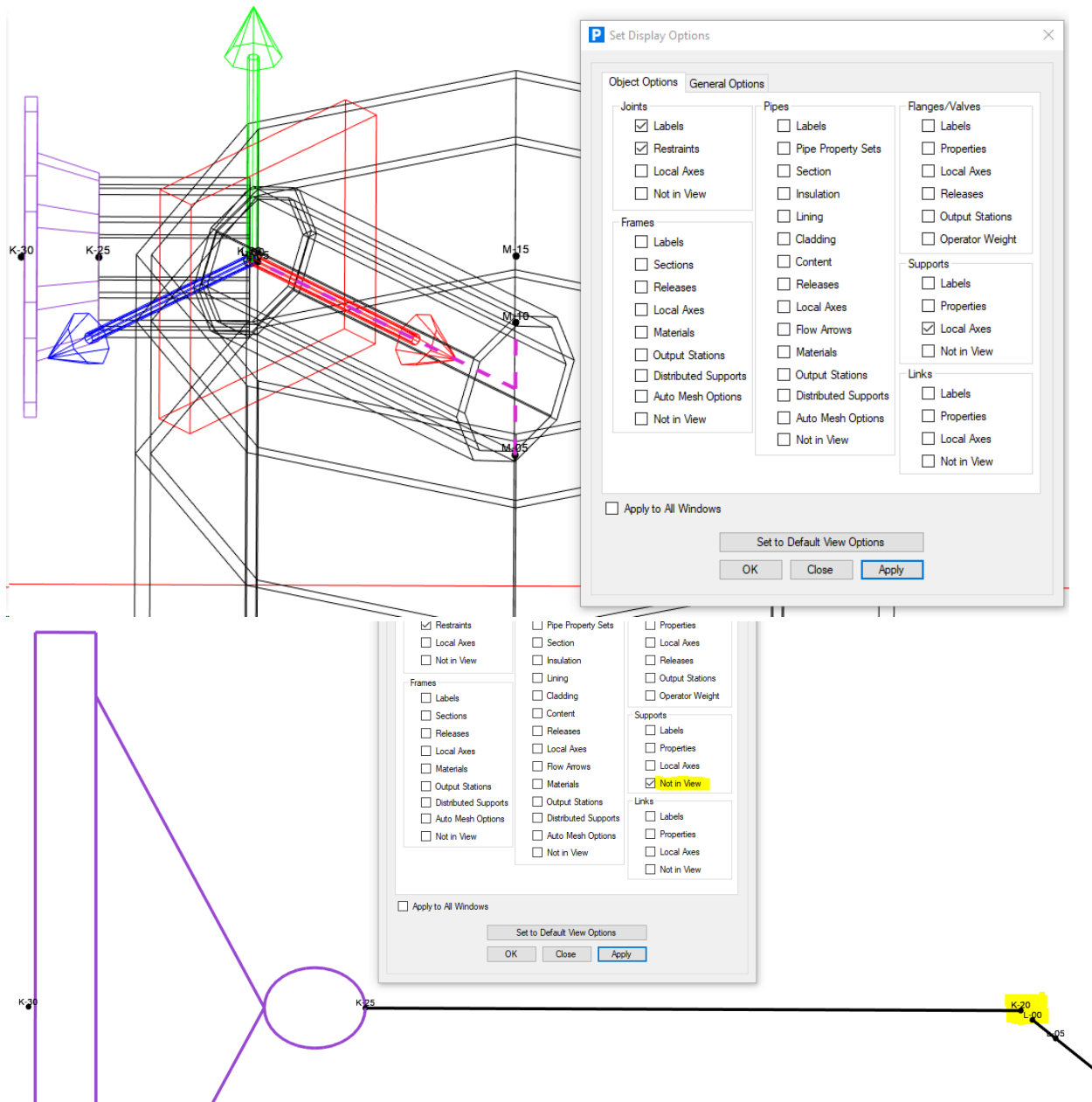
We created a video at the following link to show how to model a flexible nozzle in CSiPlant.
[Video tutorial - flexible nozzle in CSiPlant](#)

Note: this video was created using CSiPlant v6.2 with ASME B31.3 2018 Appendix D for flexibility factors. If you are defaulting to the use of ASME B31J in your model, be sure to override that setting for the vessel nozzle branch. Use a conventional method without flexibility at tees (eg ASME B31.3 Appendix D), to avoid potentially 'doubling up' on flexibility.

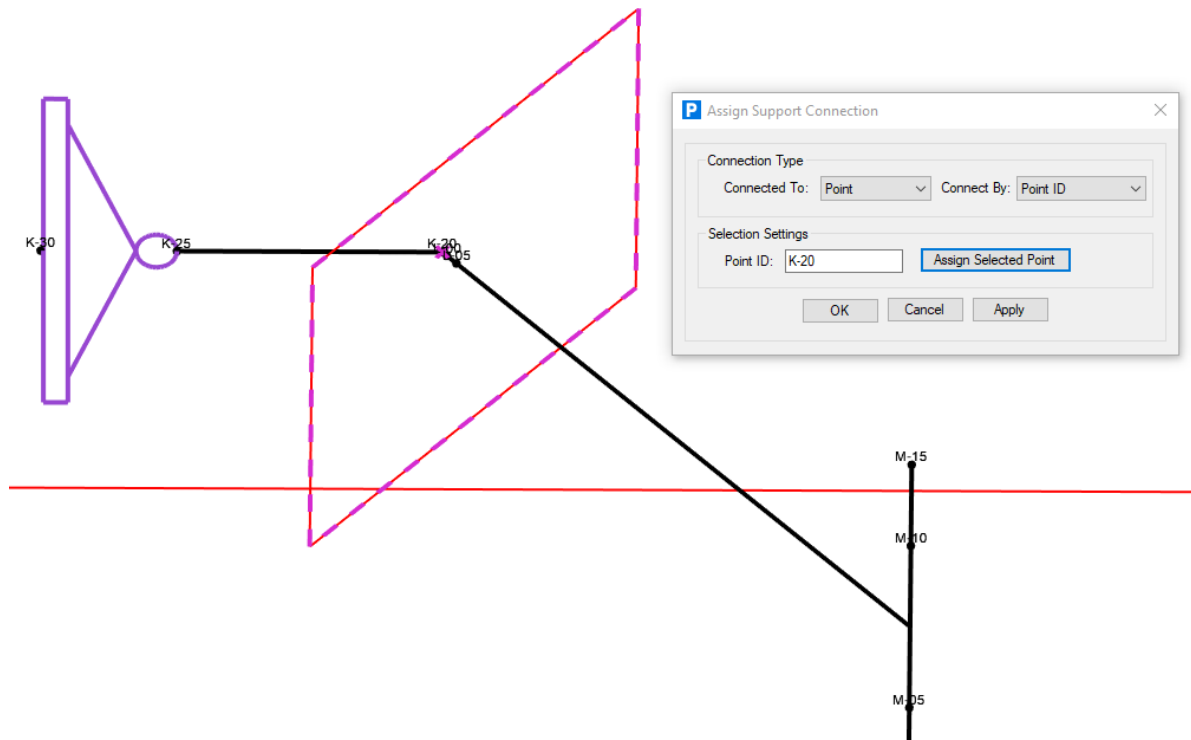
Start by defining a flexible anchor with stiffnesses taken from the plugin outputs. Generally U1 = radial stiffness, R1 = torsional stiffness, R2 = circumferential bending stiffness, R3 = longitudinal bending stiffness.



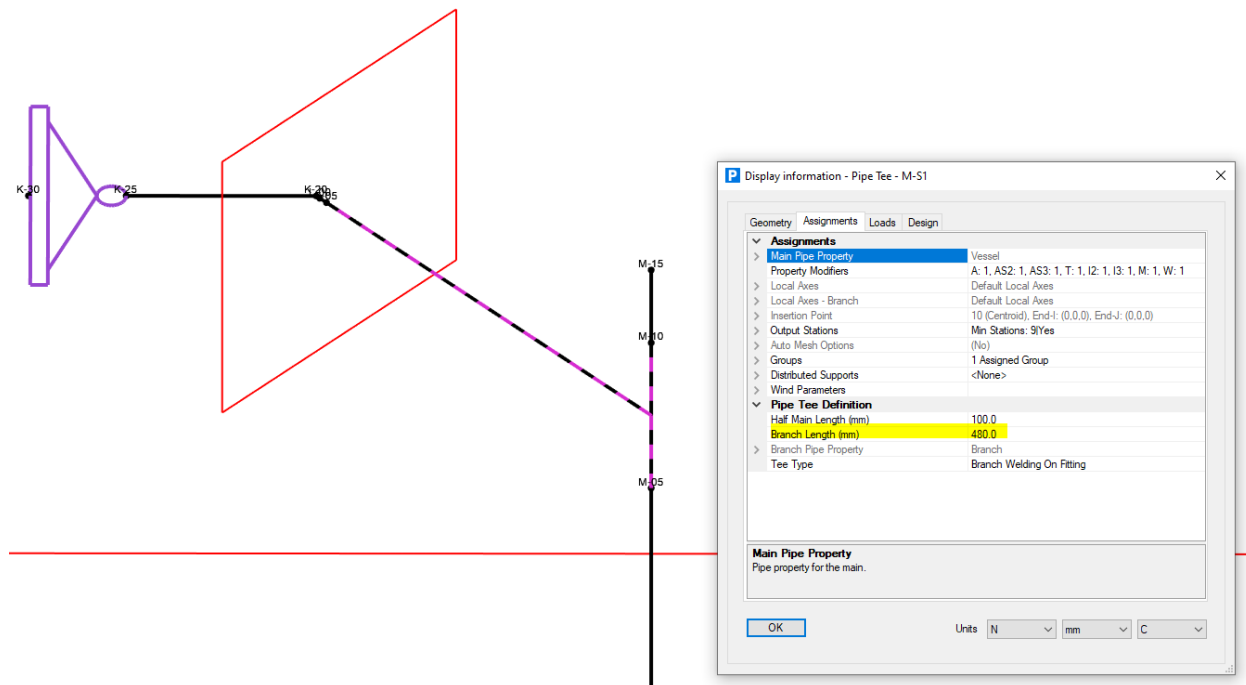
Once you've inserted the anchor you can inspect the axes as shown below to confirm that you have the directions correct. For the convention above, check that axis 1 (red) points radially inward or outward from the shell, axis 2 (green) points along the longitudinal axis and axis 3 (blue) is tangential to the shell.



CSiPlant – line view with supports not shown. Note the small gap between joints. The flex anchor should be assigned to joint L-00.



CSiPlant – Select the flexible anchor and the joint on the pipe side, and create a support connection.



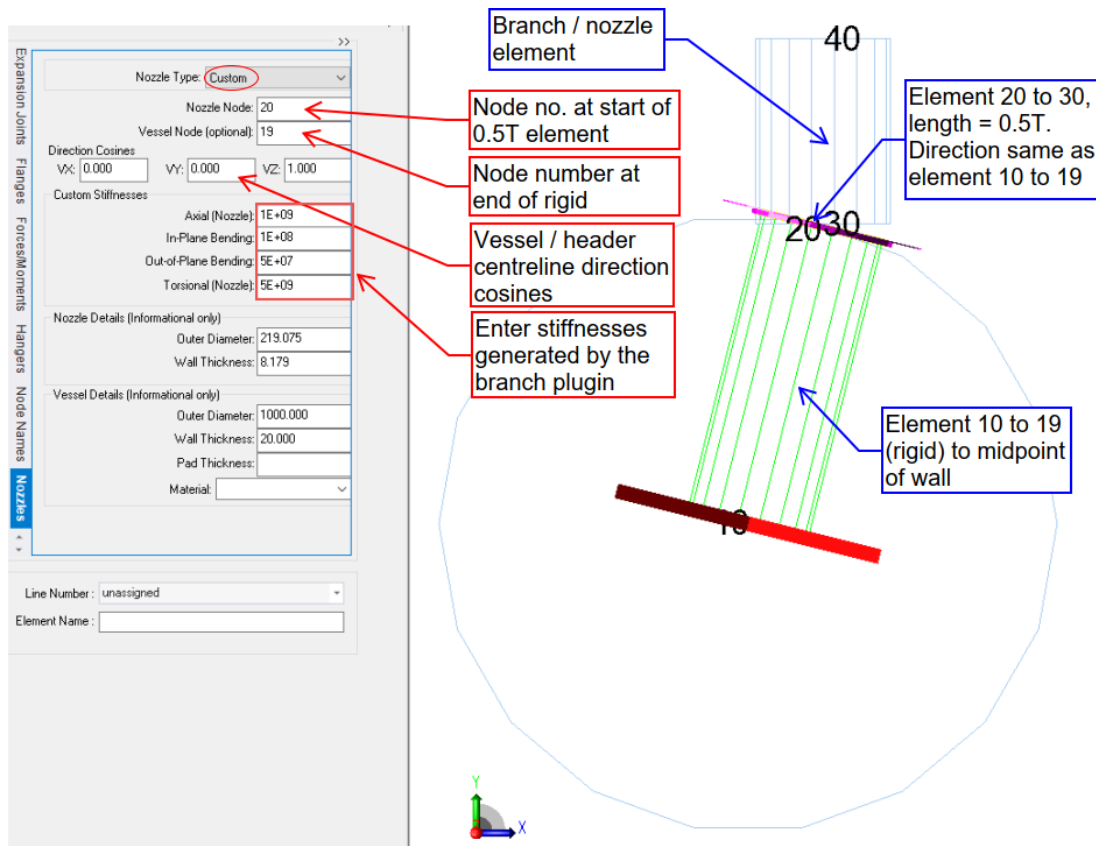
Note that the Branch Length inside the vessel / header pipe is treated as rigid by CSiPlant.

Do ensure you are not using B31J for flexibility of the vessel branch, to avoid potential additional flexibility being added at the shell intersection. All flexibility factors should be set to 1.0 under this approach.



Nozzle FleX Plugin for SAP2000

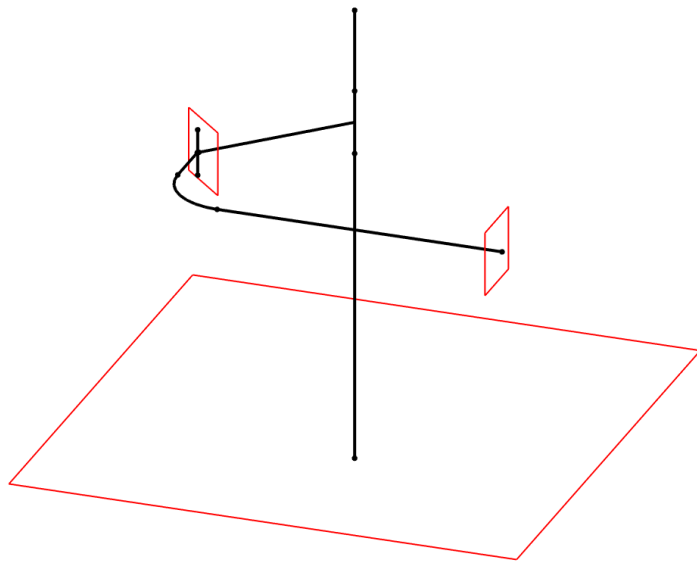
CAESAR II



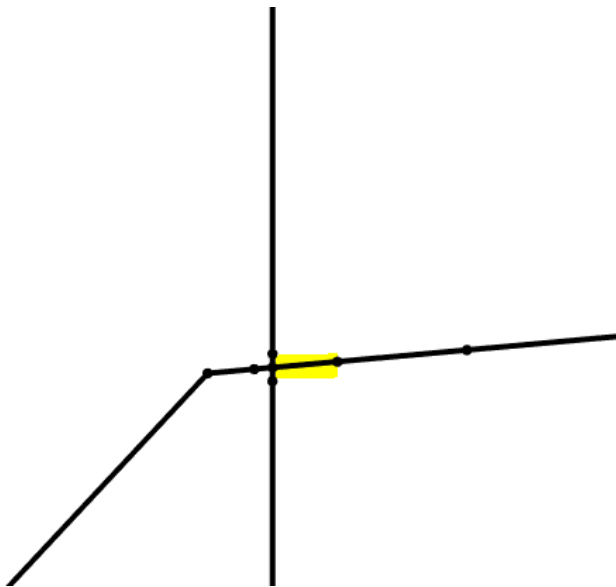
Example of entering stiffness values in CAESAR II

Applying the stress intensification factors – flexible anchor model

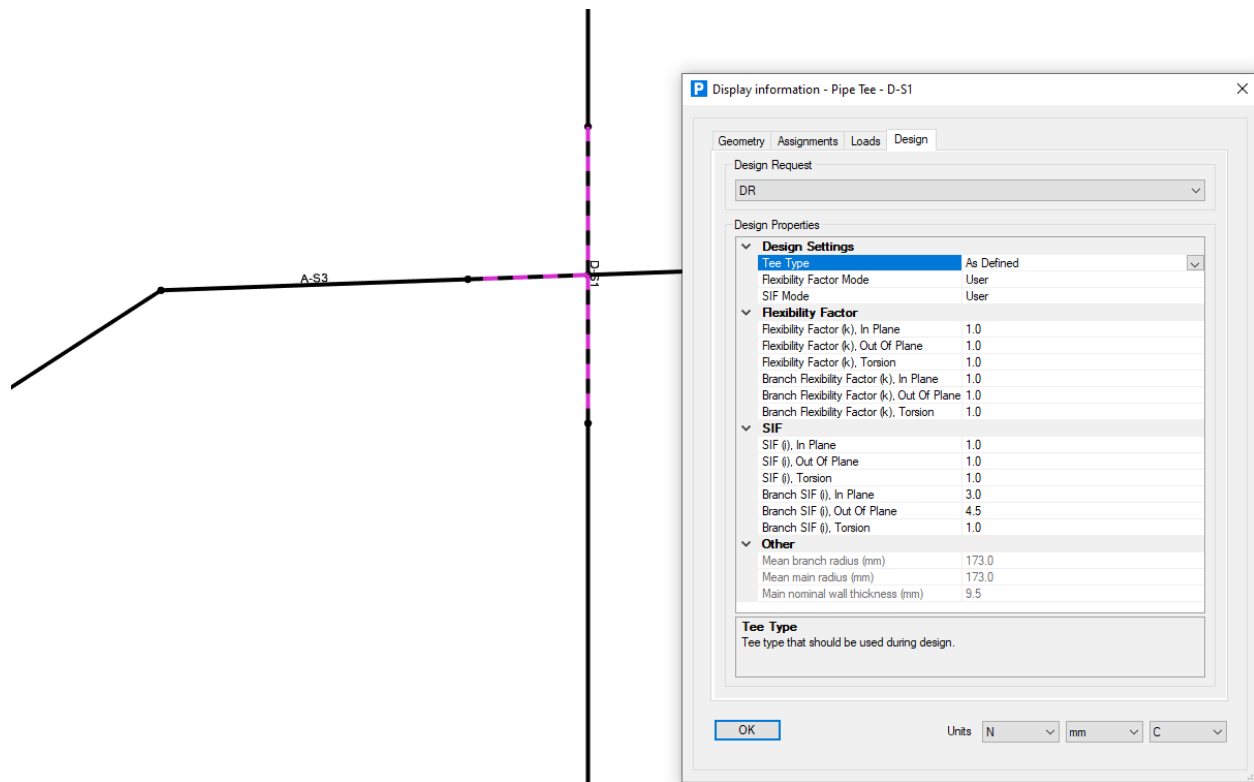
Most programs let you specify custom SIFs, but unless there is some way for the program to know what is 'in-plane' and what is 'out-plane', it must guess or make conservative assumptions. This may require you to model a 'dummy intersection' to let the program know the correct orientation, or you may opt to specify the larger of the in-plane and out-plane SIFs for both directions.



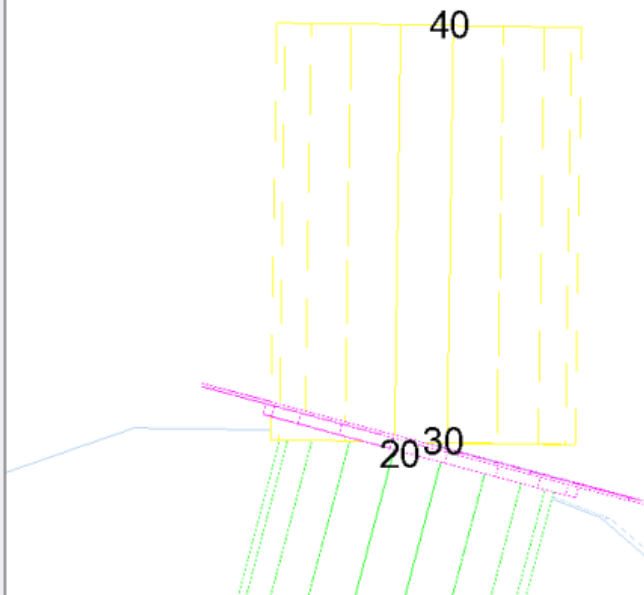
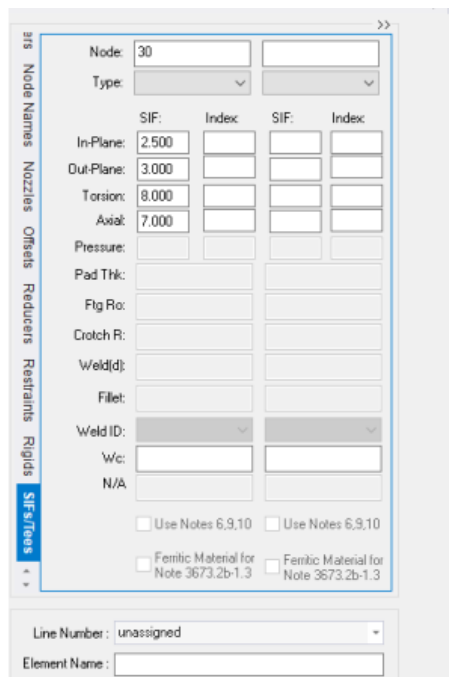
Example of modelling a dummy tee to correctly orient user-defined SIF's at the shell intersection in CSiPlant



Example of modelling a dummy tee to correctly orient user defined SIF's at the shell intersection in CSiPlant (zoomed in view of above model). The hilited element is a connection element to connect the pipe to the flexible anchor.



Defining user-defined SIF's in CSiPlant. SIF's calculated by the plugin are Branch SIF's not header SIF's, so do not alter the header SIF entries.



Entering SIF's in CAESAR II – in this example the program can't determine correct orientations for SIF's. A dummy tee can be constructed similar to that illustrated for CSiPlant above to correctly orient the SIF's.

Known Issues

Mesh for nozzles with low branch angles may need improvement.

Version history

Version	Changes
2.0	Added modelling of nozzle to head intersections (semi elliptical, torispherical, hemispherical). SIF calculation now uses principal stresses (max shear stress for torsional) rather than Von Mises. Option to output either SCFs or SIFs. Added ability to check license information. Explanatory image shown with each geometry type. Preferences for units, rotation and SCF or SIF retained after closing plugin.
1.11	Lengths used in FE model are now given in the output RTF file. Fixed incorrect flexibility factors when rotational stiffnesses were based on radians. Fixed problem for some models in English units where certain weld zone elements were not being generated.
1.10	Added option to specify covering weld SCF Added option to specify covering weld leg length Elements within weld zone have their thickness increased to account for weld size SIF calculation now ignores elements within weld zone SIF calculation now frees one end of header Default flexible lengths increased for both branch and header
1.01	Fixed mesh issue with fine mesh / small nozzles / short distance to end
1.0	First release