Table of Contents

FE/Pipe v4.5 Release Documentation

2007 Release: New Features

18 Degree of Freedom Beam Element

- **Permit element interaction of ovalization and warping effects**
- **Supports and loads may be attached to the outside pipe surface**
- **A pseudo-WRC 107 type stress calculation is made for all supports attached to the outside of the pipe.**
- **Improved thermal and dynamic solutions**
- **Modeling of stiffening rings**
- **Activated from 6dof model by single click of radio button**
- **Shell Stresses due to thickness changes**

API 579 Fitness For Service Evaluation of Local Thin Areas and Cracks

- **Crack Autosearch – Locates Critical Crack Location anywhere in the geometry**

ASME NH (Previously N47) High Temperature Local Stress Evaluation

- **High Temperature Material Database**
- **Creep-Fatigue Interaction**
- **Can Use Stresses Calculated by Any Pipe Stress Program**
- **Evaluate Short-Term High Temperature Excursions**

FE/Pipe & Nozzle/PRO Link to Latest ASME and B31.3 Material Database

- **Automatic look-up of allowable stress**
- **Direct access to allowable plots, High Temperature, Fatigue, and FFS calculators in Mat/PRO**

Integrated Beam Modeler in Nozzle/PRO

- **Any number of piping runs can be attached to the branch or header of any Nozzle/PRO model.**
- **6 or 18 dof beam elements can be used.**

ActiveX Component Licensing

- **Developers or End-Users can License PRG ActiveX Controls for FEA of Nozzles, 18dof Beams, and Other Technologies available through the NozzlePRO ActiveX Interface.**
- **FEA Nozzle Calculations can be performed through Excel**
- **User's Can Develop their own applications using FEA technologies.**

WRC 474 and BS 5500 Fatigue Analysis (Plus Extended VIII Div 2 App 5 Fatigue Curves, including latest published "Master Curve" coefficients)

Splash Automatic Modeling of Vessel Baffles

Splash Spectrum-to-Time History Converters and Spectrum Excitation Library

2007 Release: Program Updates

FE/Pipe Version 4.5

- **Fitness for Service (API 579) Reports (Evaluation of Cracks and Local Thin Areas)**
  - **Up to 15 flaws per geometry**
  - **Automatic Fracture Analysis Diagram construction and comparison**
  - **Crack growth rate calculations**
  - **Local thin area (LTA) or crack calculations**
  - **Output per API 579 Chapters 4, 5, and 9**
  - **User Flaw Locator**
  - **Ability to look at flaws at vessel/pipe supports or on support plates**
  - **Autosearch for critical crack location in the vessel or pipe**

- **Link to ASME and B31.3 Material Data Base for Automatic Property Lookup**

- **NH Reports (Creep Temperature)**
  - **Automatic creep/fatigue interaction calculation**
  - **Material Properties from API 579/ASME III Subpart NH/API 530**
  - **Actual hours at temperature and number of cycles considered**

- **Model Generator User Interface Improvements**

- **Automatic Stress Section Integration on Plane Stress and Plane Strain Element types with mechanical or thermal loading.**

Mat/PRO Version 2.0

- **Updated Fatigue Reports for latest ASME recommendations and PRG fatigue tests**

- **Updated ASME 2006 and B31.3 material database**

- **Updated Fatigue Calculation Wizard including 5 fatigue calculation methods**

- **Favorites interface to make storing commonly used materials easier**

- **ASME B31.3 materials database is now included**

- **Creep-Fatigue Interaction Diagrams**

- **Elastic-Plastic Stress Strain Curves for FE/Pipe**

- **Fatigue Curves Generated as a Function of Creep Temperature**

Nozzle/PRO Version 7.0

- **Gusseted Nozzles**

- **Piping Modeler to apply beam elements to the run or branch sections of intersection models.**
    - **Beams may be analyzed without shells (for simple pipe stress analysis).**
    - **Supports may tie one portion of the piping system to another**
    - **User labeling of elements**
    - **Element deactivation options**
    - **Beams may be automatically connected to header or brach shell model ends**
    - **One radio button changes from 6 dof to 18 dof beams**

- **Improved batch processor with import and export featues to Microsoft® Excel®**

- **Colorized Grid Presentation of Stress, SIF and Flexibility Results**

- **Fitness for Service Evaluation (API 579)**
    - **Cracks at repad edges**
    - **Local thin areas on straight sections**
    - **Local think areas at supports on the pipe interior**
    - **Corrosion on saddle plate or pipe shoes**

- **High Temperature (ASME III Subsection NH) Calculations**

- **Improved Automatic Saddle and Pipe Shoe Models**

- **Add-on options provide**
    - **Link to Mat/PRO material data bank for ASME Section II or B31.3 material property look-ups**
    - **Link to Mat/PRO for Enhanced Fatigue Analysis (Six different code approaches)**
    - **Link to Mat/PRO for High Temperature (NH) Analysis in the Creep Regime**
    - **Link to Mat/PRO and FFS for Level 2 and 3 Fitness for Service Calculations for nozzle geometries per API579 (local think areas, crack-like flaws, and general thinning)**

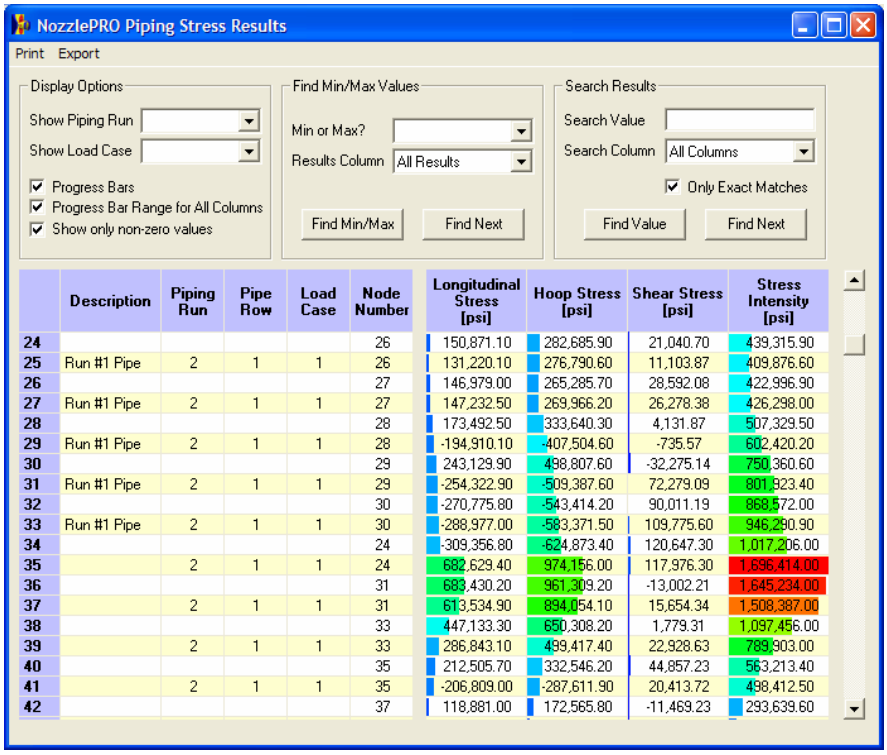
Nozzle/PRO features are described in the standalone Nozzle/PRO documentation.

Mesh/PRO Version 3.0

- **Surface regions may now be added to shell models. Shell regions can be used for a variety of purposes including applying stress concentration factors, defining identifiable regions of the model for ASME Code stress reports, and defining "no stress regions".**

- **Database joining capability has been added for shell models. Cylindrical ends of Mesh/PRO models can be joined using shell zipping or a single point join as done in all FE/Pipe database models.**

- **Addition of new capabilities for controlling surface normals.**



	Description	Piping Run	Pipe Row	Load Case	Node Number	Longitudinal Stress [psi]	Hoop Stress [psi]	Shear Stress [psi]	Stress Intensity [psi]
24					26	150,871.10	282,685.90	21,040.70	439,315.90
25	Run #1 Pipe	2	1	1	26	131,220.10	276,790.60	11,103.87	409,876.60
26					27	146,979.00	265,285.70	28,592.08	422,996.90
27	Run #1 Pipe	2	1	1	27	147,232.50	269,966.20	26,278.38	426,298.00
28					28	173,492.50	333,640.30	4,131.87	507,329.50
29	Run #1 Pipe	2	1	1	28	-194,910.10	-407,504.60	-735.57	602,420.20
30					29	243,129.90	498,807.60	-32,275.14	750,360.60
31	Run #1 Pipe	2	1	1	29	-254,322.90	-509,387.60	72,279.09	801,923.40
32					30	-270,775.80	-543,414.20	90,011.19	868,572.00
33	Run #1 Pipe	2	1	1	30	-288,977.00	-583,371.50	109,775.60	946,290.90
34					24	-309,356.80	-624,873.40	120,647.30	1,017,206.00
35		2	1	1	24	682,629.40	974,156.00	117,976.30	1,696,414.00
36					31	683,430.20	961,309.20	-13,002.21	1,645,234.00
37		2	1	1	31	613,534.90	894,054.10	15,654.34	1,508,387.00
38					33	447,133.30	650,308.20	1,779.31	1,097,456.00
39		2	1	1	33	286,843.10	499,417.40	22,928.63	789,903.00
40					35	212,505.70	332,546.20	44,857.23	563,213.40
41		2	1	1	35	-206,809.00	-287,611.90	20,413.72	498,412.50
42					37	118,881.00	172,565.80	-11,469.23	293,639.60

Nozzle/PRO Colorized Tabular Output and Search Options for Piping

Splash version 3.0

- **The interactive screen that appears during the CFD simulation of the free surface can be resized interactively. Viscosity can be added and adjusted for difficult to converge problems.**

- **Horizontal and vertical vessels with flat heads were added to the user defined construction list.**

- **Any number of perforated baffles can be added to any of three different horizontal vessel geometries.**

- **Tabular reports were updated to include more information regarding the runs.**

- **User defined time history or response spectra can be input.**

- **User defined response spectra can be input. Response spectra can be scaled in the frequency domain using response spectrum scaling or Power Spectrum Density (PSD) scaling to produce enveloping time history spectra for use with Splash.**

- **2D Spectrum plotting**

- **Model solution control refinement**

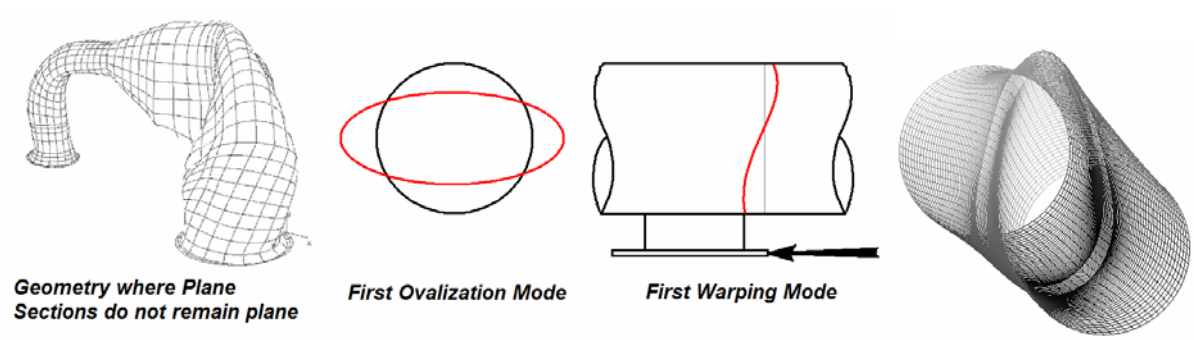
Section 1: 18 Degree of Freedom Beam Elements

General Discussion

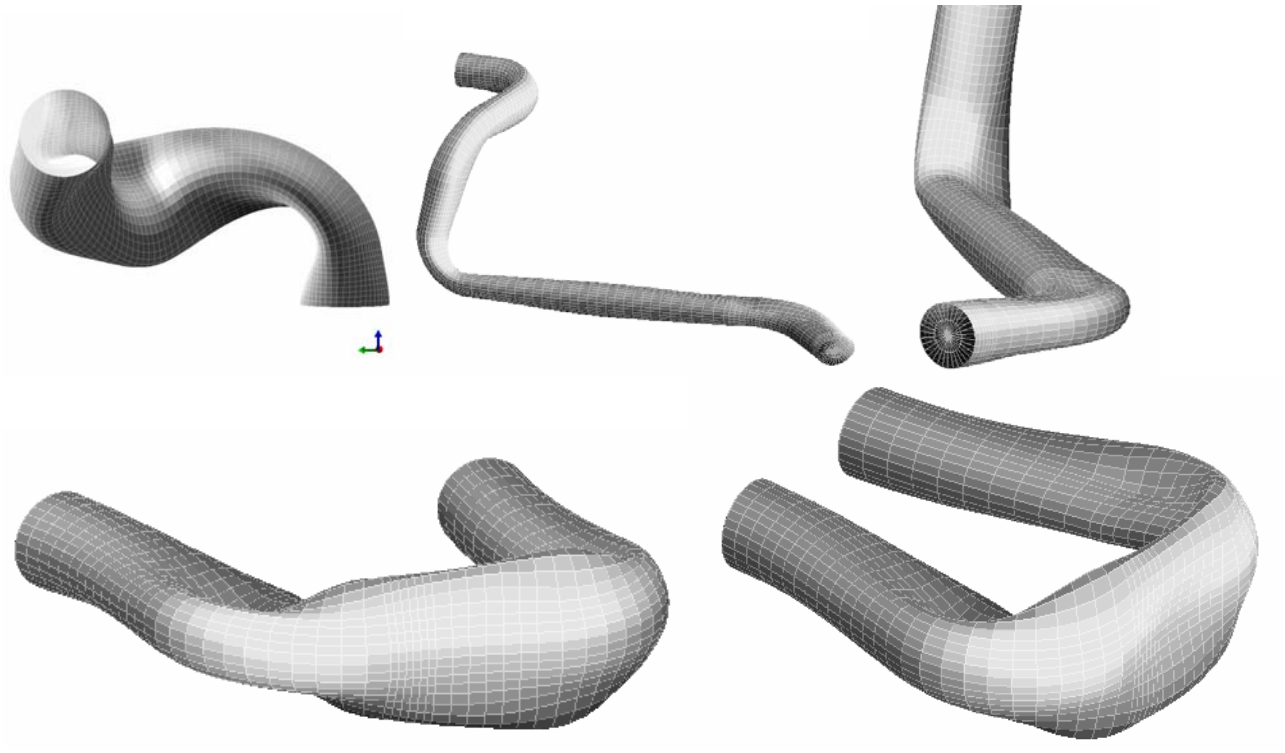
The 18 degree of freedom piping beam element is an industry first implementation for piping and pressure vessel analysis. The element formulation includes the effects of ovalization, dilation and warping that are not considered in the 6 degree of freedom classical elements found in traditional pipe stress programs. Functionality not included in a typical 6dof beam element includes:

1) Modeling of stiffeners (stiffening rings)
2) Simulation of flanges at any cross section in the model (not just at bends)
3) Loads and stiffnesses acting on the surface of the pipe
4) Interaction of ovalization between adjacent elbows
5) Differential thickness or thermal expansion between adjacent elements
6) Shell Stress Formulation to obtain more accurate stress calculations.
7) Inclusion in Dynamic analysis for more accurate shapes and frequencies
8) Correction of bending/torsional shear errors in most 18dof formulations

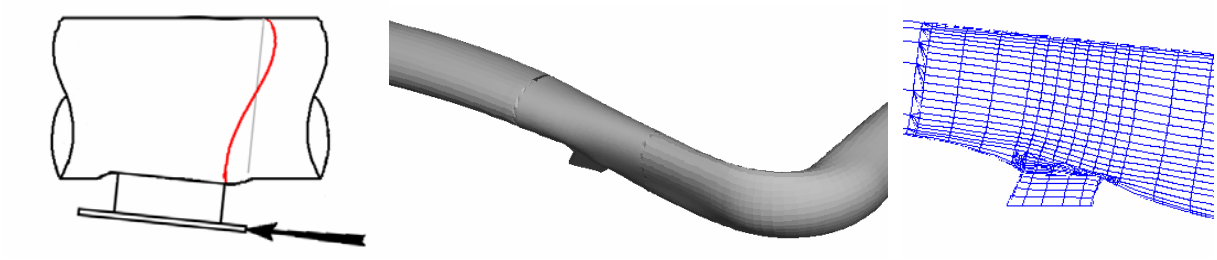
An example of ovalization and warping modes not explicitly included in typical 6dof beam elements are shown below:



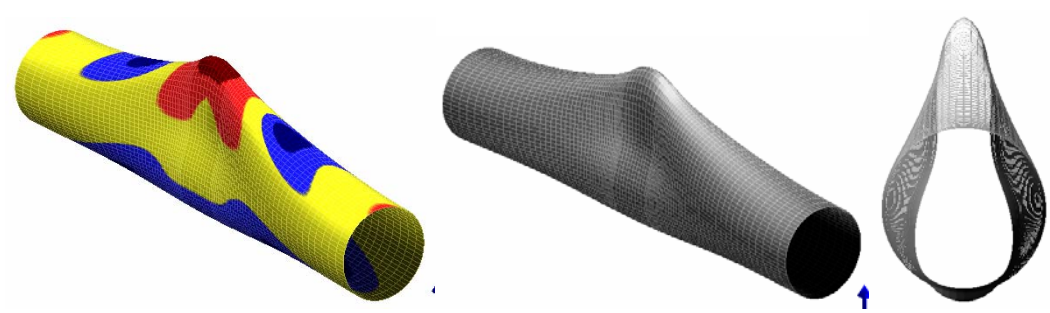
The strong interaction of ovalization modes between bends can cause one bend to either augment or retard another's ovalization modes, resulting in greater or less system stiffness depending on bend orientation. The distored plot below shows how bend cross sectional deformations can interact:



Surface loads or supports can also cause ovalization of straight or bend sections:

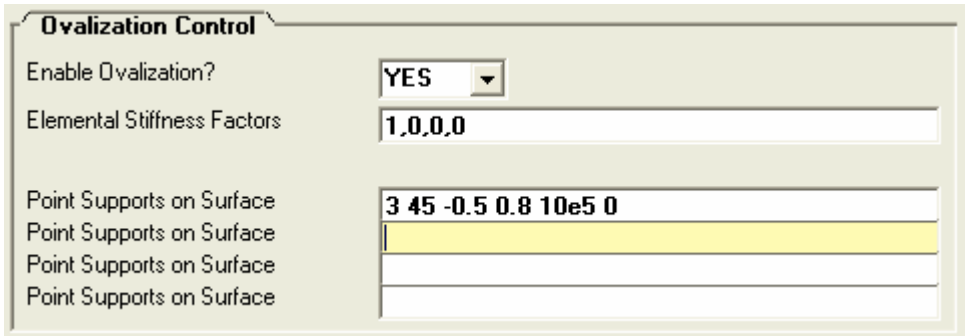


Local Applied Loads *and Stresses* can also be simulated. The shell model below shows the type of local loading *and stress* supported in the 18dof element, and how straight sections can ovalize when locally supported.



FE/Pipe Implementation

The 18 degree of freedom piping beam element is located in the “Beam Models” template. The controls for the element are found in the “Elements” panel grouped in the section labeled “Ovalization Control”.



Enable Ovalization

Select “YES” to allow ovalization effects for the element defined on this page. “YES” will change the element type from the standard 6 degree of freedom beam to the 18 degree of freedom element.

WHEN ENABLE OVALIZATION IS SET TO “YES”...
(1) ONLY A SINGLE ELEMENT SHOULD BE DEFINED PER PAGE WHEN STIFFNESS FACTORS ARE PROVIDED. THE USER WILL NOT BE ABLE TO PREDICT BEHAVIOUR WHEN TWO ELEMENTS ARE DEFINED PER PAGE

(2) ELEMENT SIZE. ELEMENTS NEXT TO GROSS STRUCTURAL DISCONTINUITIES SHOULD BE NO LONGER THAN [Rm t]$^{0.5}$. EXAMPLES: (A) A STRAIGHT ELEMENT THAT IS NEXT TO A BEND, (B) ANY ELEMENT NEXT TO A STIFFENING RING OR ANCHOR (C) NEXT TO A “POINT SUPPORT ON PIPE”

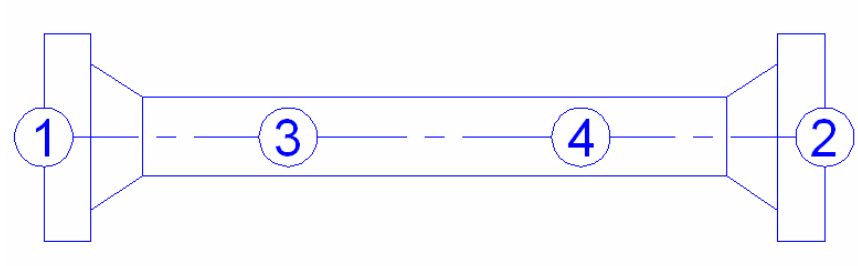
Elemental Stiffness Factor

Four inputs toggle ovalization and warping on or off at four equally spaced points along the element. Input “1” or “0” in the following order:

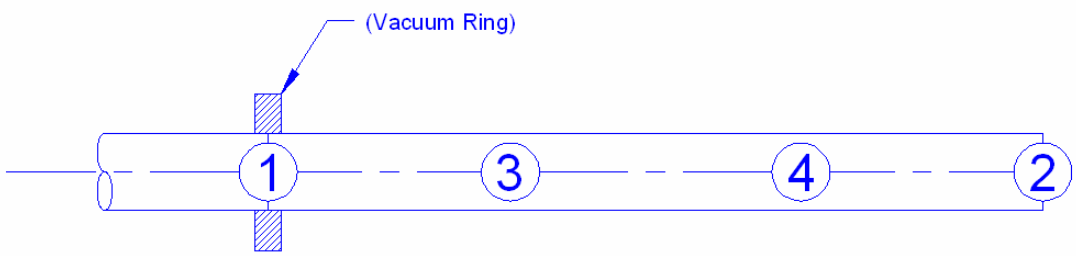
<first point>, <last point>, <internal nearest to first>, <internal nearest to last>

Where “1” restricts ovalization, warping and radial dilation (except that radial thermal expansion is unrestricted), “0” permits ovalization warping and radial dilation, “-1” restricts ovalization and warping but permits radial dilation, and any number> 1 adds a radial stiffness (eg. vacuum ring). Ovalization and warping are unaffected.

Example 1: the pipe segment shown below would be defined as 1,1,0,0 to describe the flange restriction to ovalization at both ends of the element.



Example 2: the pipe segment shown below would be defined as <K>,0,0,0 to describe the vacuum ring at the start of the element.



Where "K" is the radial stiffness of the stiffener in load per length per length of circumference. A good approximation to this stiffness is given by:

$$K = 4AE/d^2 \text{ where:}$$

K = stiffness value to input
A = Area of stiffener (the cross section revolved around the pipe centerline)
d = diameter to centroid of stiffener section

Point Supports on Surface

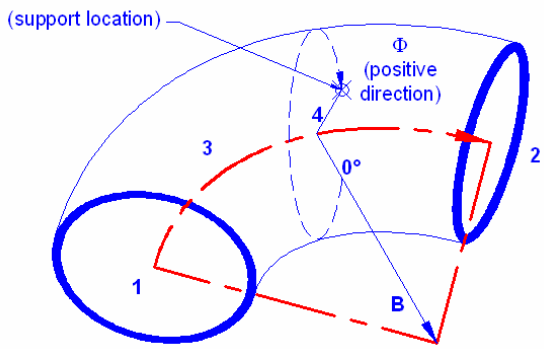
Stresses and deflections in real pipe are influenced by the location and type of pipe support. "Point Supports on Surface" controls the support location both along the length and around the circumference of the element.

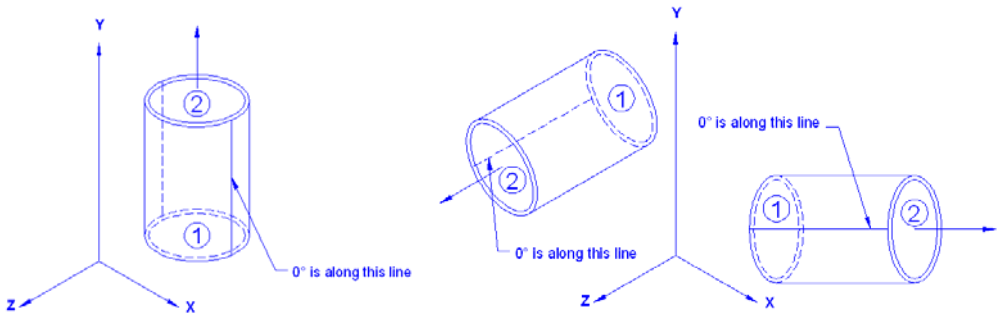
Up to four "point supports on surface" can be defined per surface. For each "point support on surface", the required inputs are:

Point#, hoop_angle, CosX, CosY, CosZ, Force, Stiff

Point#	One of four equally spaced locations along the element. The order of points is: "1" first, "2" last, "3" middle (closest to first) and "4" middle (closest to last). If Point# is equal to "5", the support is distributed along all four nodes.
hoop_angle	The angle around the circumference of the pipe (see figures below). Positive angles are defined by the orientation of points 1 to 2 and the "right hand rule". For bends, 0 degrees is the intrados. For straight elements two cases exist: (i) when the element is parallel to the Global

Y axis, 0 degrees is located on the Global +X side of the element. (ii) for all other orientations, 0 degrees is defined as the cross product of the element local x axis and the global "Y" axis. The hoop angle is designated with the symbol Φ in the figure shown below.



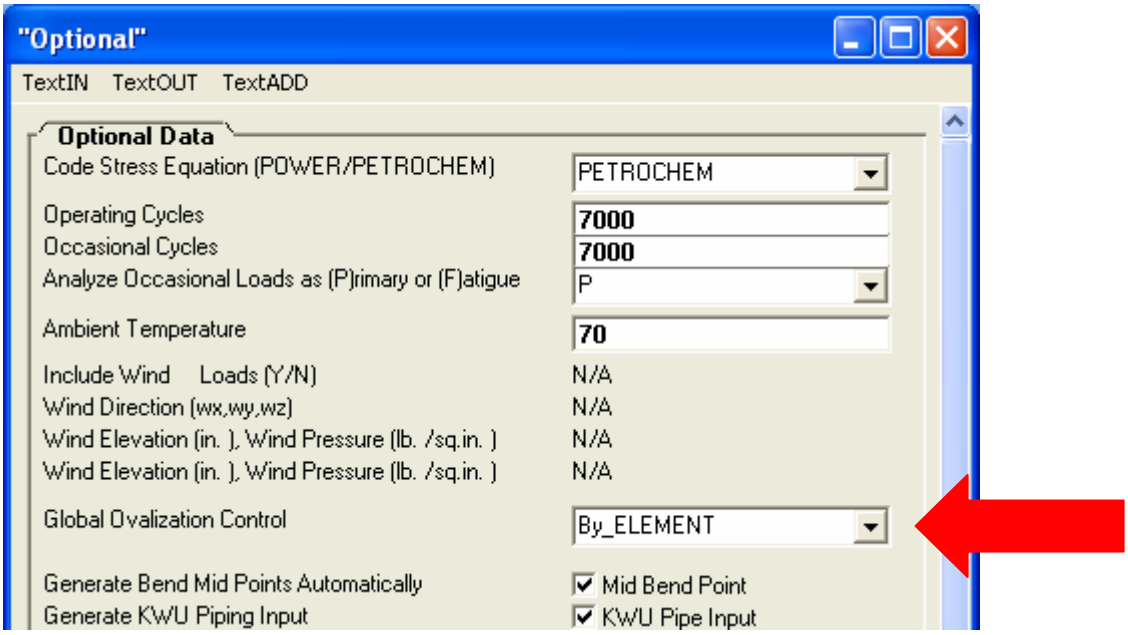


Straight Element Case (i)

Straight Element Case (ii)

CosX/Y/Z	The GLOBAL direction cosines of the force or restraint.
Force	Spring preload
Stiff	Restraint stiffness. Typical values for "rigid" restraints are 1E15 lb/in [2E14 N/mm]

The "Optional" panel includes an entry for the application of the 18 degree of freedom elements for the entire model.



Global Ovalization Control

Three options are available. They are GLOBAL_ON, By_ELEMENT, and GLOBAL_OFF respectively.

GLOBAL_ON	Ovalization and warping are activated for all the elements.
By_ELEMENT	Ovalization and warping depends on the input specified by “Enable Ovalization” entry in the Element pages.
GLOBAL_OFF	and warping are excluded for all the elements and the input in Enable Ovalization will be override.

Section 2: API 579 Fitness for Service Analysis

General Discussion

PRG Fitness for Service calculations can be used to evaluate crack-like flaws or local thin areas (LTAs) in accordance with the API 579 Guidelines. Conservative and yet realistic assumptions are used throughout and considerable control of the method is available.

Options exist for performing an AutoSEARCH for critical crack growth zones to give you an idea of where to concentrate inspection, and the size flaw should be considered critical for inspection. The AutoSEARCH calculation approach is described in a separate section below.

Fitness for Service Calculations are accessible from NozzlePRO, MatPRO and FE/Pipe.

The NozzlePRO fitness for service calculation is the easiest to use, requires the least level of expertise, and provides the most conservative solution.

MatPRO is equally easy to use, but you can only evaluate a single point at a time, and you must determine the membrane and bending stresses at the flaw before entering MatPRO (for example, from an Ansys®, FE/Pipe or NozzlePRO result).

FE/Pipe provides the most solution control but requires the most input. With FE/Pipe, you must locate the flaw and its size on the geometry of interest using a flaw influence sphere and radius. The flaw influence sphere provides an easy way for you to define local thin areas or cracks in arbitrary nozzle or plate-type shell geometries. Fitness for service options are available in the templates listed below.

- Nozzles-Plates & Shells (General Head, Cone or Cylinder Geometries)
- Unreinforced Fabricated Tee (Shells)
- Reinforced Fabricated Tee (Shells)
- Hillside Unreinforced, or Reinforced Fabricated Tee (Shells)
- Bends with Staunchions

Each of these input mechanisms, along with pertinent mechanical considerations, is described below.

Warning: You should be particularly careful when evaluating both loads and material properties when performing fitness for service evaluations.

All present operating or expected loads should be included in the evaluation. Evaluating the ductility and strength of the material and weld is of the utmost importance. There are a number of options in the API FE/Pipe calculator that let you adjust the results. Several are mentioned below:

- Probability of Failure is used for fatigue (crack) calculations. The value represents the percentage of parts that have a 50% likelihood of reaching a target, critical state. The default is 0.023, or 2.3% of the components have a 50% likelihood of reaching the target, critical state.

- The Primary Load Certainty is also given in the optional form and can be set as well known, reasonably, known, or uncertain. The influence on the calculation between well known and uncertain is observed in the partial safety factors, and is seen to be about 2.0 for certain factors when going from well known to uncertain. This should not give you the notion that unknown loads can be evaluated. When the loadings are unknown, someone very familiar with fitness for service should be consulted. It is not uncommon for calculated piping loads to be off by an order of magnitude. Improperly loaded supports, incorrect fit-up, maladjusted spring hangers, errors in valve and pipe weights, and overly simplified modeling are some sources for these errors. When a fitness for service evaluation is critical, all errors in the source of the loading should be evaluated carefully. The primary load certainty is used to adjust the partial safety factors for crack-type flaws only, and is NOT used for local thin areas (LTAs)

- The weld joint efficiency should be entered whenever the crack or LTA is in the weld, or the HAZ, or could likely grow into, or be very close to the weld or HAZ. This has a direct effect on the allowed primary loading and should be entered carefully. When welds in an LTA or cracked area have been examined carefully and satisfy requirements for joint efficiencies of 1, the joint efficiency of 1.0 can be used. Poor quality, embrittled or otherwise hardened welds can show a very low resistance to increased loads due to local metal loss or cracking and should be evaluated carefully.

- You may wish to ignore Partial Safety Factors. Depending on other parameters, the use of partial safety factors can result in a doubling of the required thickness. In some cases, users' will not want to use this extra conservatism.

A fitness for service evaluation does not provide the original safety factor used in the component design. Realistic evaluations of loads, thicknesses and future corrosion are used to determine that an adequate separation between the calculation and failure exists. Probability can be used to evaluate the scatter in fatigue test results, and can be used with flaw evaluation to produce SAFE probability of success, where SAFE implies a two standard deviation shift from the mean of the failure line, basically, if everything else is in line, and the FFS calculation is right at the limit of acceptability, then very roughly, 1 out of 100 would fail. For primary loads such as pressure, probability of failure is not used as readily, and so the FFS calculation, as applied, and taken right to the allowable limit, can be thought to provide a 1.4 times safety factor against a pressure boundary failure. This is increased by the actual strength vs. the minimum or analyzed strength. In certain instances these margins are not satisfied, but in general they do, and ultimate plant failure (if one occurs), will be due to a multitude of unfortunate events, all happening together.

Applications

There are several orientations of flaws that are of common interest in a pressure vessel and piping geometries.

- Edge flaw in plate
- Thru wall flaw in plate
- Surface flaw in plate
- Thru wall flaw in cylindrical pipe

- Surface flaw in cylindrical pipe
- Surface flaw adjacent to discontinuity

Surface cracks are of most interest in a pressure vessel and piping (PVP) geometry because these are the most commonly experienced and because these flaws can propagate quickly under load due to the triaxial state of stress experienced at the deepest point of the surface flaw.

The basic purpose of a fitness for service evaluation is to determine the proximity of the flaw to failure. As stress in and around the flaw increases the stress state will either be constrained or unconstrained.

Where leakage is a major hazard even small surface flaws can be important because these flaws can grow quickly in the thru-thickness direction resulting in leak.

A leak-before-break analysis merely qualifies the thru-wall flaw as stable or unstable. An unstable thru-wall-flaw will grow in size, uncontrollable releasing product. A stable thru-wall-flaw will not grow in size once the flaw reaches completely through the wall thickness, but rather will remain at a fixed size, spewing contents into the environment.

Generally, the desire is to prevent any surface flaw from becoming a thru-wall flaw, and this is the objective of the PRG surface flaw evaluators. Once a flaw has grown completely through the pipe or vessel wall, it is desired that the crack length remain stable, and this is the leak-before-break criteria.

It is thought that surface flaws in PVP geometries will grow predominantly in the thru-thickness direction. This is particularly true in a membrane stress field. In a predominantly bending stress filed the crack will take more of an elliptical shape as it progresses, its length as well as depth increasing, however with the depth increasing most rapidly.

Material Properties

The Advanced Form allows you to enter material properties that will override program defaults for the material of choice. All pertinent data is included in the output reports for you to review. Actual yield and tensile strengths, correct Charpy energies, etc. can all be used if available. This is particularly useful in cryogenic conditions where material properties can be considerably stronger than at room temperature, (i.e. 304 Stainless).

Fracture toughness can drop in metals at temperatures above freezing as shown in the plot below. Low temperatures and old or poor quality welds can demonstrate very low fracture toughness values. Pipe or vessels subject to spot X-ray, or single sided welds that cannot be inspected from the inside, may be particularly susceptible to low fracture toughness values. *Flaws in welds should be evaluated carefully.*

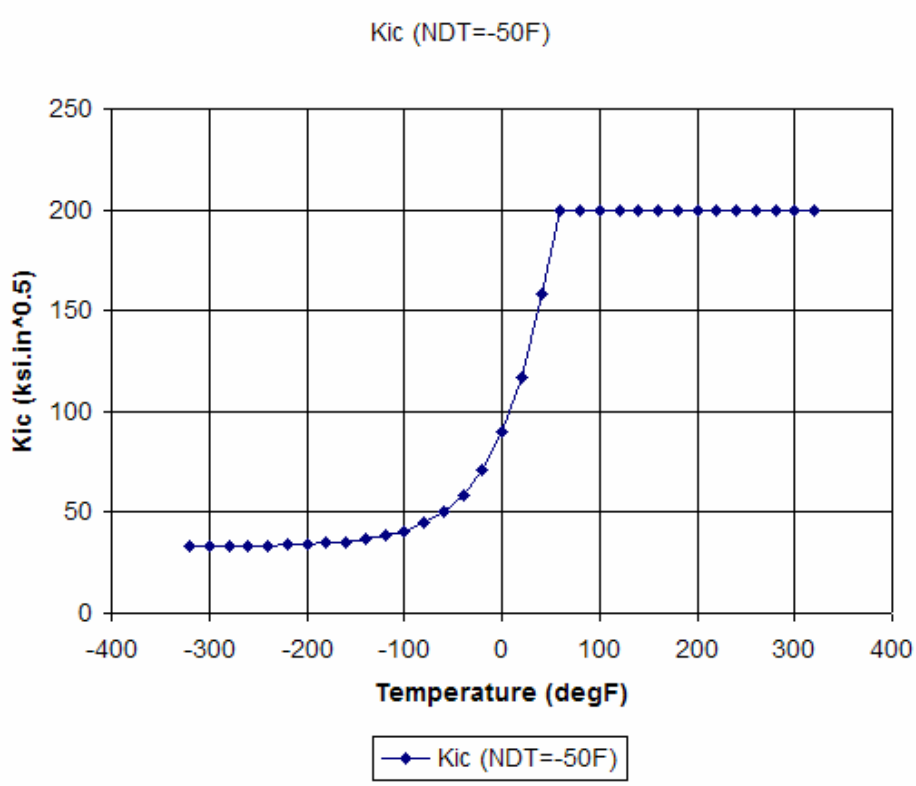


Figure 2-1: Critical stress intensity (K_{IC}) vs. Temperature

Axial strength on A53 Gr. A carbon steel pipe material at room and chilled temperature are shown below. Note that at a cryogenic temperature, the yield strength is higher, but the ductility at failure is nonexistent.

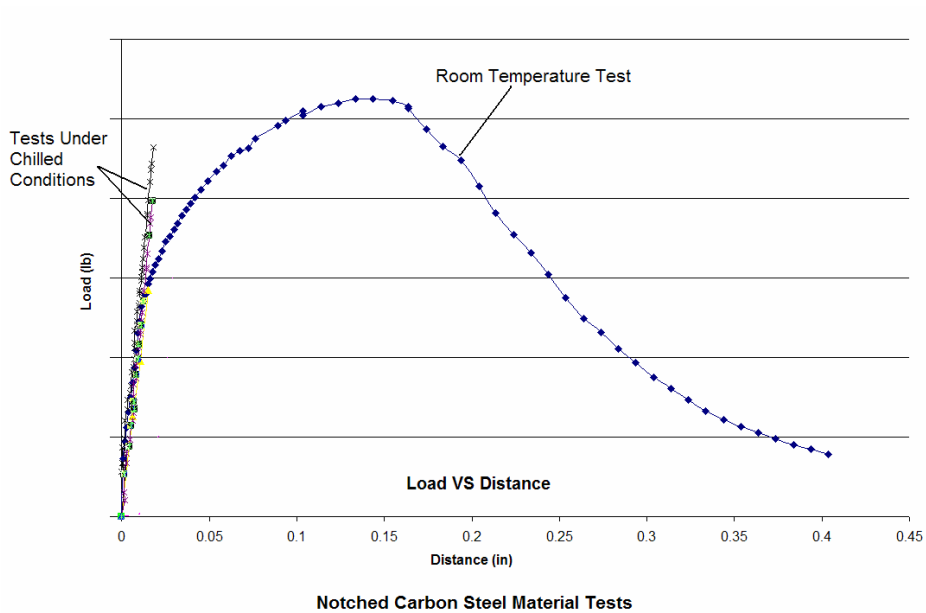


Figure 2-2 Stress Strain Diagram for Room and Cryogenic Temperatures (Carbon steel pipe)

The material yield and tensile stresses at room and operating temperatures are required for an FFS evaluation. These values are obtained from the MatPRO material data base or from the input. In either case, the values used in the evaluation are printed in the full reports, and you should check to be sure that correct values are entered.

The NDT, Nil Ductility Temperature, or the reference temperature (T_{ref}), as used in the FFS routine, and as defined in API 579, is the temperature corresponding to a Charpy value of 15 ft-lb for carbon steels and 20 ft-lb for Cr-Mo steels. This is not the Nil Ductility Temperature as defined in ASME NB 3200.

The critical stress intensity (K_{IC}) used in the calculations for cracks is found by going through the following steps.

1) User entered value is chosen over all others.
2) If stainless steel, then 200 ksi.in1/2 is used outside of a weld zone, and 120 ksi.in1/2 is used inside a weld zone.
3) If a Charpy value is provided at temperature, then KIC is calculated from the Charpy value and the yield stress.
4) If a user defined critical J Integral value is given, it will be used with the modulus and poisons ratio to find KIC.
5) If a user defined crack tip opening displacement is identified, then it will be used with the modulus and poisons ratio to find KIC.
6) If the user's reference temperature is entered, it will be used with the API 579 Appendix F.4.4.1 equation for KIC.
7) If no reference temperature is given, a reference temperature of 100F is used in the API 579 Appendix F.4.4.1 equation for KIC.

Carbon steel K_{IC} values for a given material reference temperature are illustrated below.

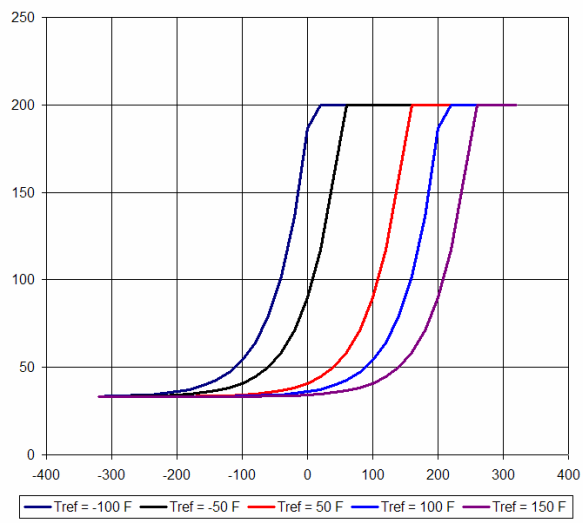


Figure 2-3: Effect of (T_{ref}) Nil Ductility Temperature on Fracture Toughness Curve

Local Discontinuities

Local discontinuities are considered structural items such as nozzles, platform clips, or other supports. In general these discontinuities are considered removed from flaws in both the longitudinal (meridonal), and circumferential direction when they are further away than $\beta(RT)^{1/2}$, where β is a number between 1.0 and 2.0. When a flaw is "*removed*" from a discontinuity, it is assumed that the local stress state due to the discontinuity does not affect the stress state at the flaw. For relatively thin geometries, the value $(RT)^{1/2}$ is often a small value, and so it is easily construed that discontinuities are frequently far enough away from flaws so that they do not affect them.

Often, loads at discontinuities can produce an ovalization of the vessel or pipe. This ovalization, and the shell bending stresses that accompanies it, is not limited to a zone defined by $(RT)^{1/2}$, and can extend much further away from the discontinuity and affect the stress state at a flaw.

A properly run finite element analysis should be able to evaluate the effect of discontinuities and determine when ovalization due to load or local weakness is present. You should be aware that boundary conditions can restrict this ovalization and should be far enough away from points of interest so that an artificial local stiffening of the shell does not occur. If you are unsure about the location of a boundary condition, a sensitivity study should be conducted, which basically involves moving the boundary condition further away (usually by more than an integer multiple of the diameter) and seeing if the solution is changed.

Using Crack AutoSEARCH

The Crack AutoSEARCH functionality is designed to help you understand which part of a component is particularly susceptible to a given flaw size and over what area. For example, if a large area of the circumference of a nozzle does not satisfy API 579 requirements when the flaw is 1.5 mm deep and if the component is susceptible to cracking, then careful inspection is warranted. If only a small percentage of the circumference of a nozzle does not satisfy API 579 requirements when the flaw is one-half the material thickness and no flaws are currently present, then less care is dictated and inspection resources can be directed to more critical areas.

AutoSEARCH should be used to develop inspection and criticality guidelines for key or highly loaded components in the piping or vessel system.

AutoSEARCH Recommendations

- Enter the following (4) flaws.

Table 2-1: Recommended AutoSEARCH Flaw Sizes

Flaw #	Flaw Depth	Flaw Length
1	0.06 in (1.5 mm)	Greater of $[3(RT)^{1/2}$, 1.5 in.]
2	0.06 in (1.5 mm) + $0.2t_{nom}$	Greater of $[3(RT)^{1/2}$, 1.5 in.]
3	0.06 in (1.5 mm) + $0.4t_{nom}$	Greater of $[3(RT)^{1/2}$, 1.5 in.]
4	0.06 in (1.5 mm) + 0.6 t_{nom}	Greater of $[3(RT)^{1/2}$, 1.5 in.]

- Describe the flaws using the “AUTOSEARCH” location option.
- Enter the operating loads that are most critical and use the best estimated of actual material thicknesses and properties. Any large occasional loads that can exist should be evaluated separately, once the operating conditions are evaluated.

AutoSEARCH Example

In the twin nozzle geometry shown below, there is concern about the level of inspection to be performed, and whether or not a failure would occur before inspection could reasonably find a flaw. It is also desired to map out the areas most susceptible to flaw initiation and growth.

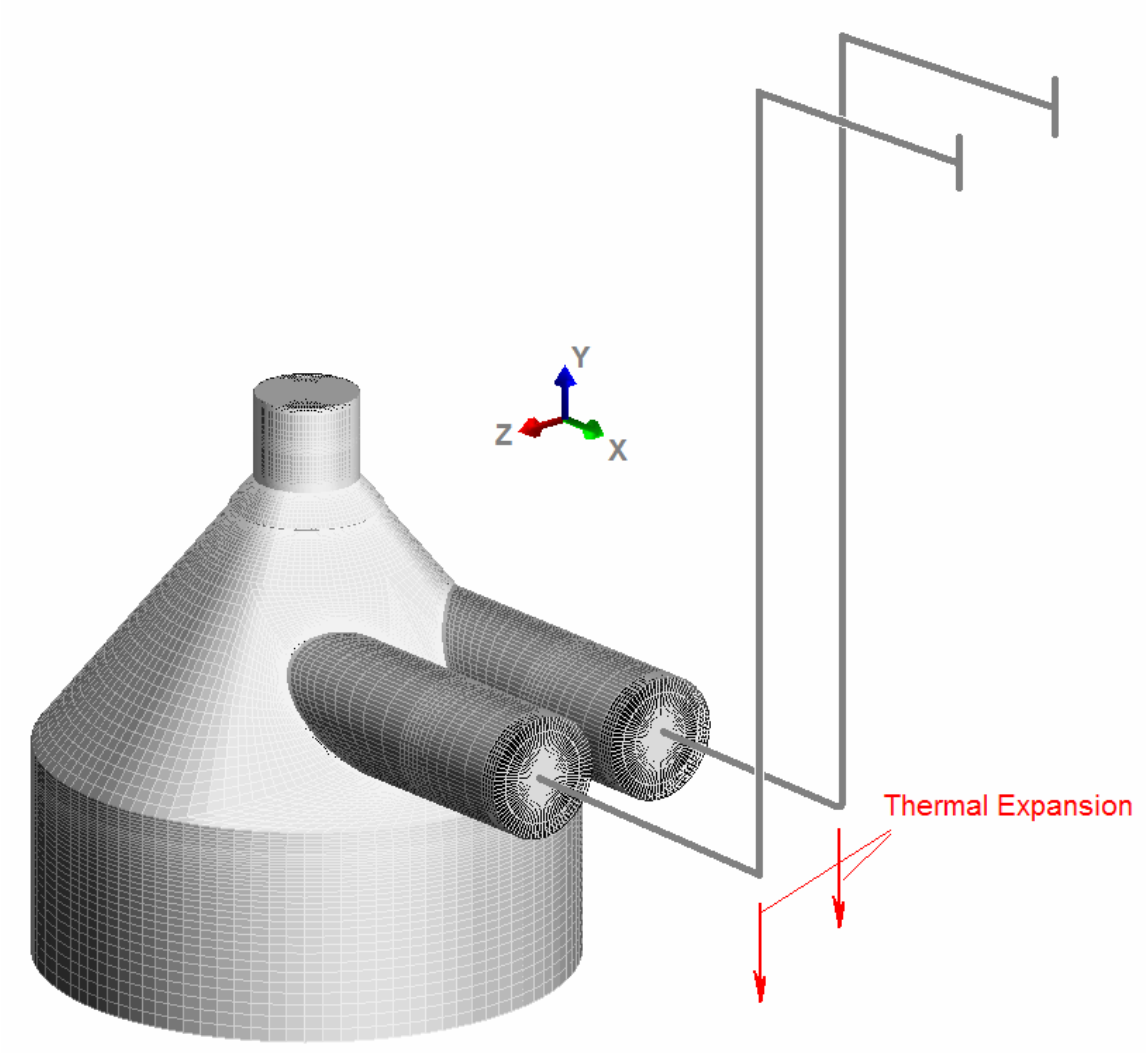


Figure 2-4: Crack AutoSEARCH Example

Whereas, it is always the engineer’s responsibility to select the parameters most important for a given situation, the following parameters are recommended for AUTOSEARCH:

Table 2-2: Recommended AutoSEARCH Flaw Parameters

Parameter	Setting	Reason
PWHT	YES	General tensile residual stresses at welds or attachments become compressive after a few load cycles.
Ignore PSF	YES	When ductile systems are evaluated, scatter bands are lower, and partial safety factors are not as applicable.
Proximity to Weld	Weld_HAZ	Even if the high stress is not immediately next to a discontinuity, at some point in the crack's life, it may cross a longitudinal or circumferential seam weld.
Nil Ductility Temperature	Set if Available and Applicable	When the material and behavior is ductile, failure predictions are much more accurate.
Charpy Test Energy at Operating Temperature	Set if Available and Applicable	A considerable reduction in strength exists if the weld or material is hard or embrittled. It is very important when performing an FFS examination, that brittle stress states (thick geometries), or embrittled materials are not present.

The four recommended AutoSEARCH flaw descriptions are shown below:

Table 2-3: Recommended AutoSEARCH Flaw Descriptions for Example Problem

	1	2	3	4
Flaw description	AutoSEARCH 0.06 in	AutoSEARCH 0.06in+0.2t	AutoSEARCH 0.06in+0.4t	AutoSEARCH 0.06in+0.6t
Evaluate this flaw?	YES	YES	YES	YES
General Location Option	AUTOSEARCH	AUTOSEARCH	AUTOSEARCH	AUTOSEARCH
Flaw depth (in.)	0.06	0.26	0.46	0.66
Flaw length (in.)	7	7	7	7
Midsurface radius at flaw (in.)	45	45	45	45
Local Nominal Thickness (in.)	1	1	1	1
Flaw Material Number	1	1	1	1
Flaw Average Temperature (Deg)	150	150	150	150
Pressure at Flaw (lb./sq.in.)	300	300	300	300
PWHT?	YES	YES	YES	YES
Marine?	NO	NO	NO	NO
Dynamic Load?	NO	NO	NO	NO
Ignore PSF?	YES	YES	YES	YES
Joint Efficiency at Flaw	1	1	1	1
Flaw Type	CRACK	CRACK	CRACK	CRACK
Flaw Profile	ELLIPTIC	ELLIPTIC	ELLIPTIC	ELLIPTIC
Probability of failure	LOW	LOW	LOW	LOW
Certainty of loads and dimensions	VERYCERTAIN	VERYCERTAIN	VERYCERTAIN	VERYCERTAIN
Proximity to Weld	Weld_HAZ	Weld_HAZ	Weld_HAZ	Weld_HAZ
Nil Ductility Temp	125	125	125	125
Charpy Test at operating Temp	33	33	33	33

When the analysis is completed, four flaw evaluation plots will be available with the regular FE/Pipe ASME Section VIII stress evaluations. These are shown as *"Results"* or *"3d"* menu options:

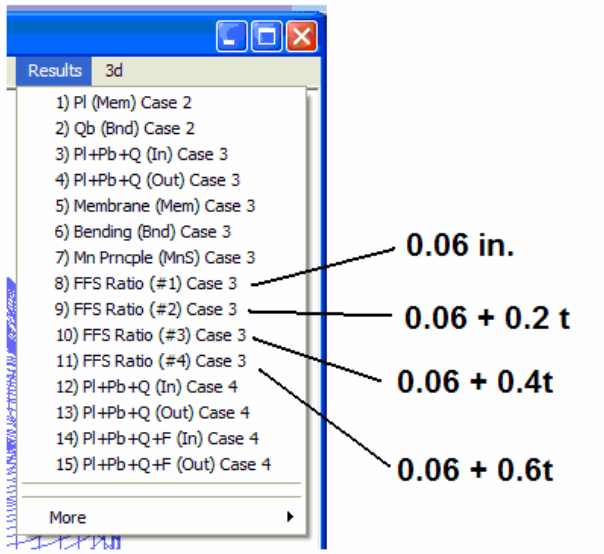


Figure 2-5: Plotted AutoSEARCH Results

Each plot shows the increasing criticality of detectable flaws (δ = flaw depth).

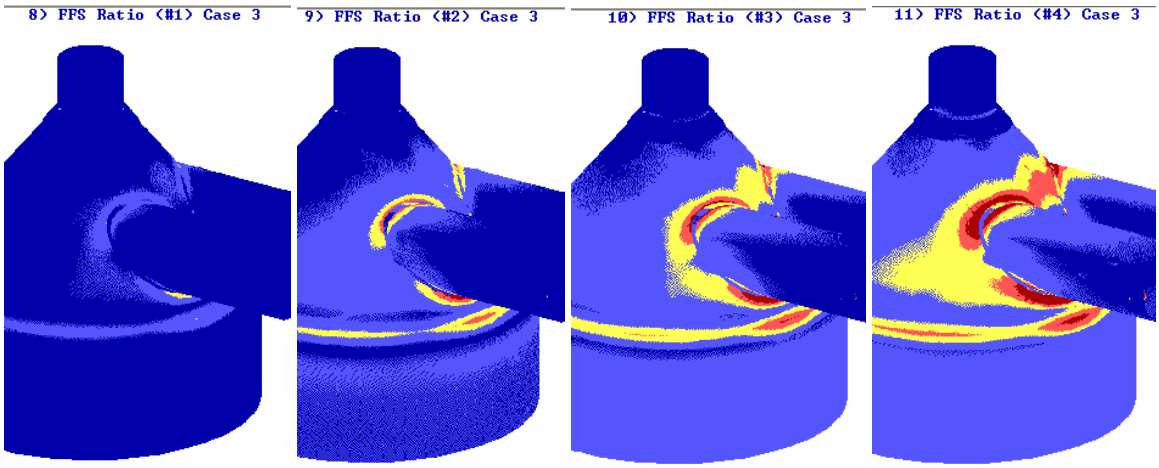


Figure 2-6: FE/Pipe Plots Illustrating Critical Crack Areas as a Function of Crack Depth

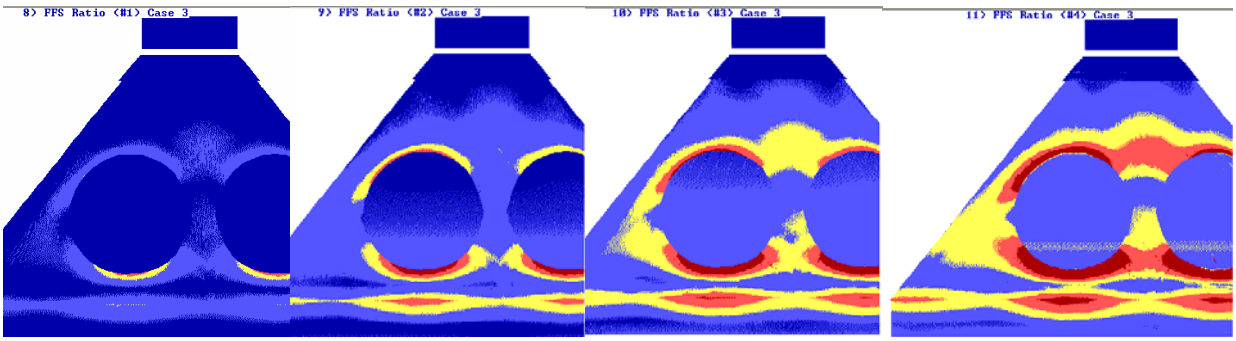
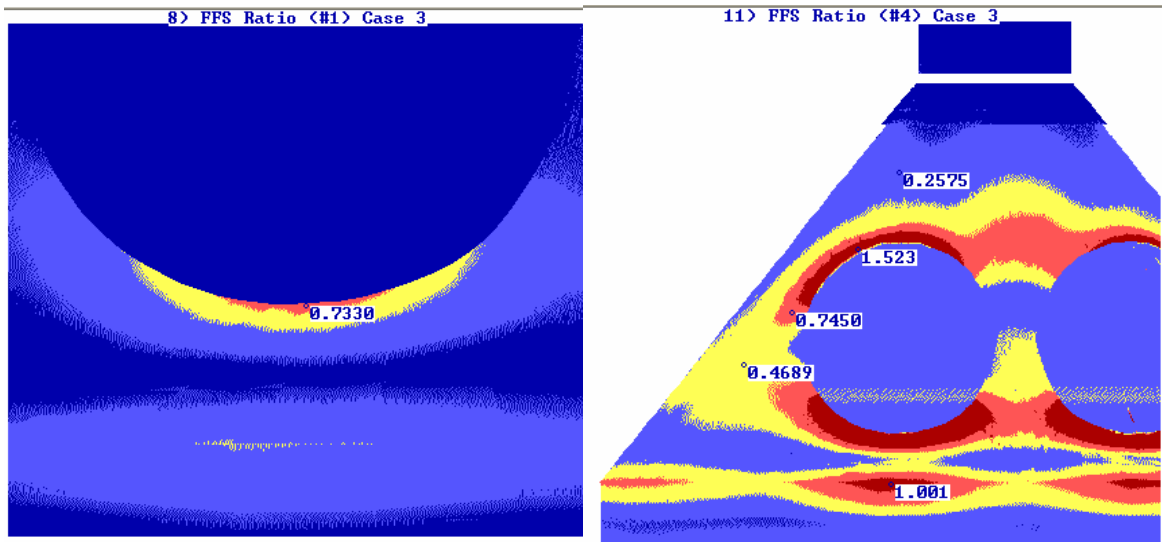


Figure 2-7: FE/Pipe Plots Illustrating Critical Crack Areas as a Function of Crack Depth (Front View)



δ = 0.06 in. (Limit of Objectionable Weld Flaw) **δ = 0.06 + 0.6t (60% of Nominal Wall Loss)**

Figure 2-8: Critical Crack Areas at Nozzles Rejectionable Initial Flaw and 60% of Wall Depth

There are two uses for the plotted results above. Any time during an inspection cycle that flaws are found, the flaw depth and location can be cross checked with the plots above for acceptability. Deep red areas show rejectionable flaws at the flaw depth given.

Additionally, the rightmost plot in Figure 2-8 above shows that a 60% thru-wall crack will not be critical in a ductile material in the light red, yellow and blue areas of the geometry. A 60% thru wall flaw has reached a rejectionable size in the dark red sections of the geometry. Inspection should focus on the deep red areas, including the circumferential weld just below the nozzles.

The leftmost plot in Figure 2-8 above shows that the system does not appear to be sensitive to rejectionable flaws. The material, geometry and load all show to be suitably resistant to typical fabrication defects.

When the AutoSEARCH option is used, the two plot ranges above are typical. The leftmost plot at a flaw depth of 0.06 inch should show no rejectionable areas. If the rightmost plot also shows no rejectionable areas, the material is in a very low stress state, and the user should be sure that all loads have been included. Inspection of the component may still be warranted if the system is in cyclic service or there is some other material degradation mechanism, but if all plots show no rejectionable areas, and no cracks have been found during inspection, then further inspection efforts can be focused on other, more critical items.

If cracks are found at any time during an inspection, the plots can be cross-checked with the flaw location and the depth of flaw evaluated for criticality.

Cracks can start from any point on the geometry, but in general start from the inside or outside, and at welds.

The conclusions drawn from these methods should always be reviewed by the appropriate fitness for service engineer, and they should verify:

- That loads used are accurate and conservatively estimated
- That go/nogo decisions are not tied to the sensitivity of the solution
- Where elastic follow-up or primary loads are present, extra concern should be taken.
- Extra care should be exercised when the ductility of the material is questioned. Some conditions where this may occur are:
    - Thick Sections
    - Creep damaged material
    - Poor quality (or very old) welds
    - Chemical embrittlement (SCC)
    - Highly cyclic conditions
    - Unknown corrosion rates
    - Uninspectable parts of the joint

As can be seen in the following plot, higher stresses exist in the parent material, and smaller stresses exist in the nozzle material. This does not mean that cracks will not appear in the nozzle material, only that the probability of a crack to appear in the nozzle material is smaller. The t/T ratio can often be used to get a quick estimate of which component is more susceptible to crack growth due to higher stress, where:

t = thickness of the branch
T = thickness of the parent/vessel/header

If t/T >> 1, then any cracking is generally expected in the parent/vessel/header material. If t/T << 1 then any cracking is generally expected in the branch.

In cases where it is difficult to inspect, Ultrasonic Testing (UT), as shown below, can estimate crack lengths, along with, in some cases eddy current, and the imaginative use of x-rays. Phased array (UT) is preferred to a fixed angle scan. The engineer responsible for the FFS evaluation should be comfortable that flaw sizes have been evaluated

accurately, and that material properties at the flaw are the same as those used in the calculations.

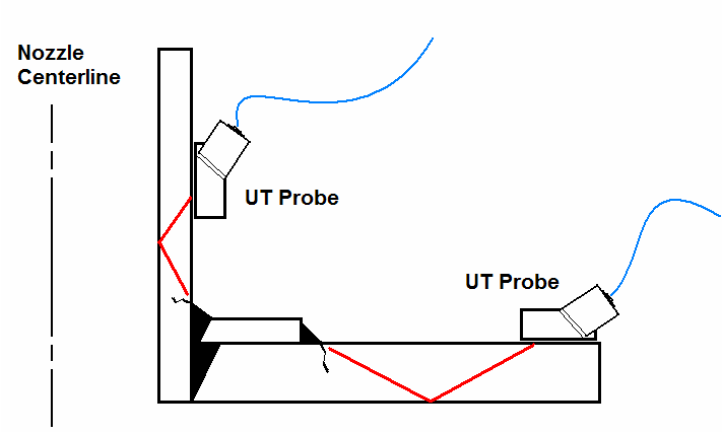


Figure 2-9: Inspecting and Measuring Cracks Starting from the Outside

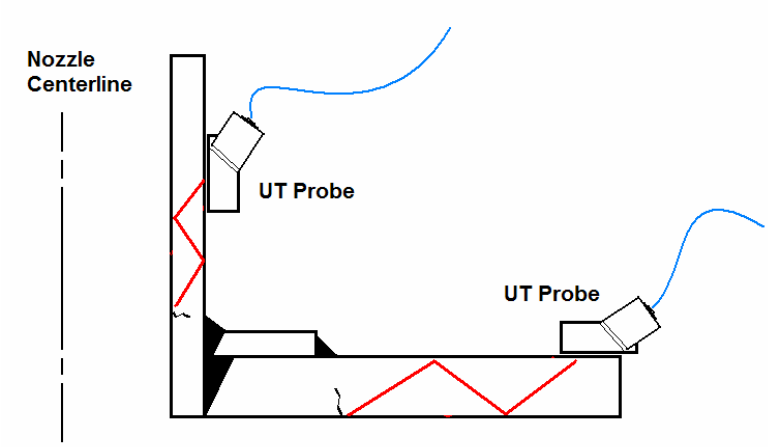


Figure 2-10: Inspecting and Measuring Cracks Starting from the Outside

When looking for cracks in new equipment, the rightmost plot in Figure 2-8 above should be used as a starting point to focus any inspection, and the FE/Pipe output fatigue reports indicate when a properly made component might expect failure. For nozzle 3 in the above example, the fatigue report is given below:

```
Shell Next to Nozzle 3

  Pl+Pb+Q+F              Sa      Primary+Secondary+Peak (Inner) Load Case 4
     29,753          41,997      Stress Concentration Factor = 1.350
        psi             psi      Strain Concentration Factor = 1.000
                                 Cycles Allowed for this Stress = 21,321.
              70%                "B31" Fatigue Stress Allowable = 50000.0
                                 Markl Fatigue Stress Allowable = 41701.0
                                 WRC 474 Mean Cycles to Failure = 110,267.
                                 WRC 474 99% Probability Cycles = 25,617.
                                 WRC 474 95% Probability Cycles = 35,566.
                                 BS5500 Allowed Cycles(Curve F) = 22,163.
                                 Membrane-to-Bending Ratio = 0.268
                                 Bending-to-PL+PB+Q Ratio = 0.789
                                 Sect VIII Ref: 4-112(l)(2),Fig.4-130.1,4-135
                                 Plot Reference:
                                 14) Pl+Pb+Q+F < Sa (EXP,Inside) Case 4
```

This report shows that in a typically welded geometry with acceptable initial flaws, the mean load life is 110,267 cycles. This means that after 110,367 cycles of load, 50% of the samples tested would have suffered a thru-wall crack and leak.

Taken together, these features provide a powerful tool to the inspector or plant engineer interested in evaluating flaws in major components.

Program Input

The FE/Pipe user can access the fitness for service calculations three ways:

1) From the FE/Pipe data screens in templates where fitness for service is installed.
2) From NozzlePRO
3) From MatPRO

Fitness for Service options are installed in the following FE/Pipe templates:

- Nozzles-Plates & Shells
- Unreinforced Fabricated Tee
- Reinforced Fabricated Tee
- Hillside Tee
- Bend with Staunchion

In FE/Pipe the user must specify the center and radius of the flaw or local thinned zone:

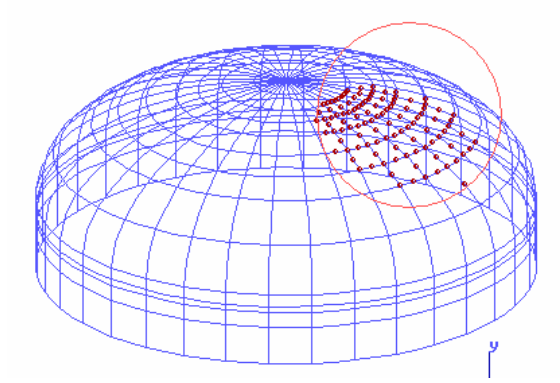


Figure 2-11: FE/Pipe Identified Local Thinned Area on a Head

In MatPRO the user must directly enter the membrane and bending nominal stresses at the flow or local thinned area.

The fitness for service buttons on most FE/Pipe “General” Screens appears: 

When using the fitness for service option in MatPRO, you must enter the membrane and bending stresses for the primary and secondary cases being analyzed. NozzlePRO and FE/Pipe look these values up automatically during the course of the finite element run.

The form below describes most of the FFS input used for FE/Pipe. The more self-explanatory of these are found in NozzlePRO, although the flaw location options in NozzlePRO are more generic, and are easier to use, but offer less control.

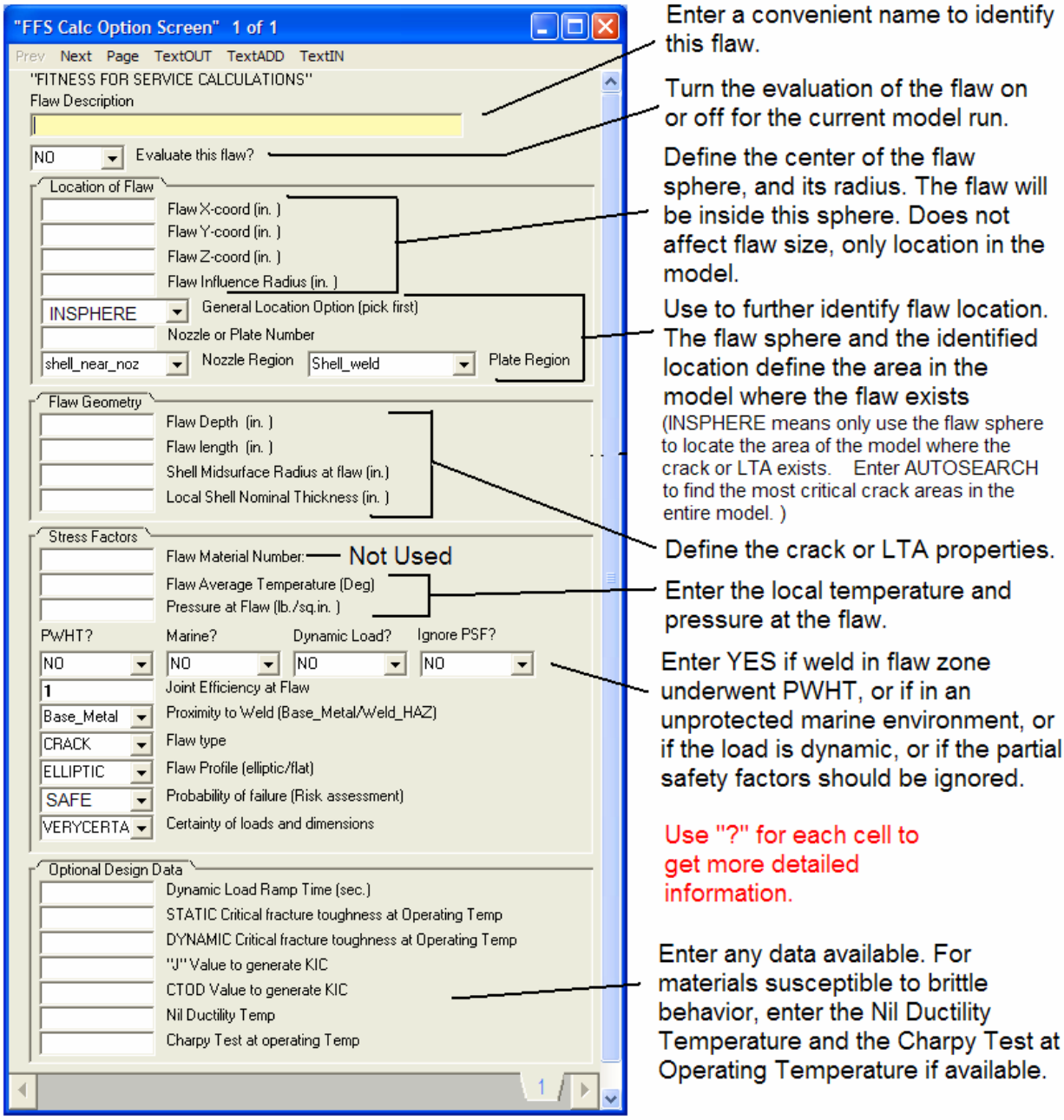


Figure 2-12: General FE/Pipe FFS Input Data

Nomenclature

CET Critical Exposure Temperature – the lowest metal temperature derived from either the operating or atmospheric conditions. For pressure vessels the CET is the lowest metal temperature at which a component will be subject to a general primary membrane tensile stress greater than 8 ksi (55 MPa).

CTP Critical Thickness Profile – the CTP in the longitudinal and circumferential direction is determined by projecting the minimum remaining thickness for each position along all parallel inspection planes onto a common plane. The length of the profile is established by determining the end point locations where the remaining wall thickness is greater than t_{min} in the longitudinal and circumferential directions.

FAD Assessment Diagram

FCA Corrosion Allowance

FFS Fitness for Service

LTA Local Thin Area (or Groove-Like Flaw)

MAT Minimum Allowable Temperature is the permissible lower metal temperature limit for a given material at a thickness based on its resistance to brittle fracture.

MAWP Maximum Allowable Working Pressure

MFH Maximum Fill Height

NDE Non Destructive Examination

NDT Nil Ductility Temperature

RP Recommended Practice

RSF Remaining Strength Factor

FE/Pipe FFS Input and Effect on Calculations

Proximity to Weld – Tells if the flaw is in the heat affected zone (HAZ), in the weld, or in the base metal. This flaw locator is not used for local thin areas, but is used for crack-like flaw evaluation. The effect of welds in local thin areas is included in the evaluation by the specification of the weld joint efficiency. For joint efficiencies of 1, the fact that the local thin area is in a weld has no effect.

Probability of Failure (POF) – If given in percent, then the number out of 100 specimens expected to fail. One standard deviation is equal to a POF of about 15%. Two standard deviations is equal to a POF of about 2.5%, and three standard deviations is equal to a POF of about 0.15%. The available options for POF in FE/Pipe and NozzlePRO correspond to approximately, two, three and four standard deviations: 0.023 (2.3%), 0.001 (0.1%), and 0.000001 (0.0001%). Most PVP Codes use a probability of failure between two and three standard deviations, and so the first or second option is reasonable. A POF of 2.3% is the

default, and is recommended for the conservative PRG approaches used. The POF is only used in the evaluation of crack-like flaws to determine the partial safety factors for crack length, stress and K_{IC} determination, and has no effect on the evaluation of local thin areas (LTAs).

Non-Factored Fracture Toughness – Value of K_{IC} reported in FFS output report for crack-like flaws that shows the material fracture toughness value before it has been modified by any partial safety factors. If the non-factored values are significantly less than the factored K_{IC} values, you should look closely at the POF used for the calculation and the uncertainty in the primary load. *Considerable caution should be exercised when flaws are evaluated in geometries where the primary loads are unknown*.

PWHT – Factor used for nozzle welds when they have been post weld heat treated. This factor is only used in the evaluation of crack-like flaws and will reduce the effect of residual stress when the flaw is in the proximity of a weld. FE/Pipe and NozzlePRO permit you to indicate that PWHT has or has-not been performed. If PWHT *has* been performed the residual stresses in the weld are reduced to 20% of their non-PWHT values.

Weld Joint Efficiency – Used for local thin areas only (LTAs), and is a direct multiplier on the allowable stress.

Ignore PSF – Partial safety factors (PSFs) as applied in API 579 are used to develop a desired confidence limit, and are developed from statistical considerations for load, material toughness and crack size, (PSFs – for load/stress, PSFk for toughness, and PSFa for crack depth. If, in the designer's opinion, the application of these safety factors is unnecessary because they provide a misguided level of confidence, then they may be removed. Partial safety factors are only used in the evaluation of crack-like flaws. Ignoring PSFs also results in a change to local thin area allowable, increasing it, equal to the flow stress, whereas otherwise it is based on 75% of the minimum specified yield stress.

Flaw Description – Each flaw can have user entered text that will be displayed when the flaw location is demonstrated and in the output. The description is not required but is recommended.

Evaluate This Flaw (Yes/No Combo Box) – User's can deactivate flaws if desired. It is not uncommon to enter the same flaw multiple times using different parameters to perform a sensitivity study to see what parameters affect the result most strongly. During the evaluation, flaw options that are not useful can be deactivated or deleted.

Flaw X,Y,Z Coordinate – Each flaw is evaluated as if they are in all or a part of the model geometry. If, in a part of the model geometry, the area where the flaw can be defined by either a flaw influence sphere, or by a geometric description, i.e. BRANCH, or by both, the Flaw X,Y,Z coordinates describe the center of the flaw influence sphere.

Flaw Influence Radius – The radius of the flaw influence sphere, centered about the Flaw X,Y,Z coordinates. The flaw will be located inside this sphere. You can enter an influence radius to be very large, i.e. 100000, to allow the FFS algorithm to investigate the effect of the flaw placed anywhere in an identified area of the model. (Use AutoSEARCH – to investigate the effect of the flaw placed anywhere in the model geometry.)

In Unprotected Marine Environment? (Yes/No Combo Box) – This option is used for crack growth rate calculations but does not affect the local thin area computations. Increases the crack growth rate for both stainless and carbon steels per API 579 F.5.3 by 4.4 times. Only used for crack-type flaws.

Dynamic? (Yes/No Combo Box) – This options is used when some portion of the operating load is applied dynamically. In this case, the K_{IC} value will be adjusted based on the temperature and the Dynamic Ramp loading time. You can override this calculation by entering the DYNAMIC Critical fracture toughness at operating temperature if a better value is available. Only used for crack-type flaws.

Flaw Type – The user can evaluate local thin areas (LTAs), crack type flaws, or both.

Dynamic Load Ramp Time (sec) – This is the time that any applied loading takes to get the secondary stress without concentrations to the material yield strength. Used if the Dynamic Load Combo Box is set to YES. This option has an effect on the calculated value of K_{IC}. Method can be found in API 579 Appendix F. Only used for crack type flaws.

STATIC Critical Fracture Toughness at Operating Temperature – K_{IC} value at operating conditions. This value will be estimated by the program based on the type of material input if left unspecified. Only used for crack type flaws.

DYNAMIC Critical Fracture Toughness at Operating Temperature – K_{IC} value at operating conditions for dynamic loadings. Only used if the Dynamic? loading combo box is set to YES. If not entered the program will calculate a dynamic K_{IC} based on loading time and temperature. Only used for crack-type flaws.

"J" Value to generate K_{IC} – If a J integral value is entered it will be used to compute the K_{IC} per API 579 Appendix F.4.2. Enter MPa-m, for metric units, and Ksi-in for English units. Only used for crack-type flaws.

CTOD Value to Generate K_{IC} – If a crack tip opening displacement value is available from a CTOD test of the material then this value may be entered as per API 579 Appendix F.4.2. Enter mm. for metric units, and in. for English units. Only used for crack-type flaws.

Nil Ductility Temperature (degF or degC) – Used as the "reference temperature", and defined as the temperature corresponding to a Charpy value of 15 ft-lb for carbon steels and 20 ft-lb for Cr-Mo steels. The Nil Ductility Temperature is not used for stainless steels. Only used for crack type flaws.

Charpy Test Energy at Operating Temperature – (N.m for metric units and ft.lb. for Imperial units). Enter the Charpy energy at operating temperature if available. This value can be converted into the K_{IC} value to be used in crack-type flaw evaluations.

For a given local thin area or crack, a Folias or "bulging" factor is used. This factor is more conservative whenever a smaller radius is given. When the user has an option to enter the local radius at the flaw location and there is more than one possible entry, the smallest value should always be used.

Locating Flaws using FE/Pipe

FE/Pipe provides you with robust control of flaw location. Note that NozzlePRO users can get access to this control by clicking in the optional form checkbox “Use FEPipe Editor”. The FE/Pipe user selects a generic location from a dropdown option list, and specifies a sphere of influence and center. The nodes in the model that fall within both groups are included in the FFS determination. This concept is shown in the figure below.

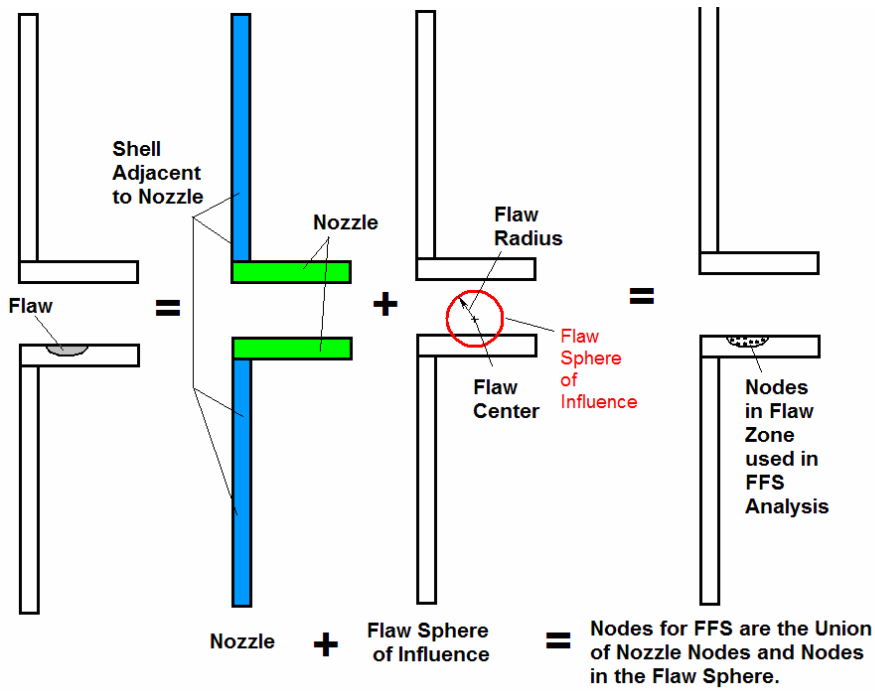


Figure 2-13: Flaw Zone Definitions in FE/Pipe

A model whose nodes have been identified in a similar manner is shown below.

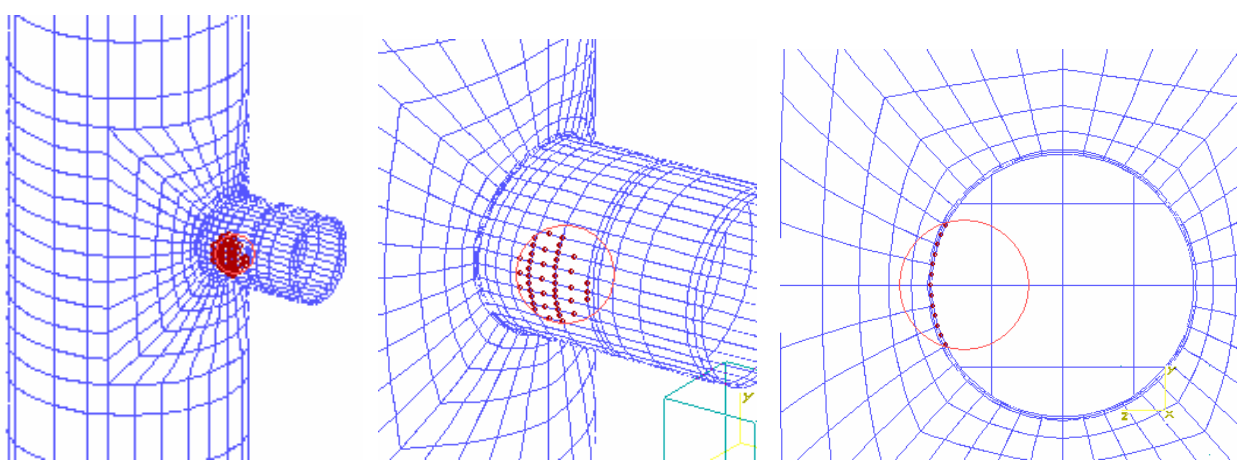


Figure 2-14: Nodes Analyzed in Flaw Zone

The FE/Pipe flaw description for the above nodal area is shown below:

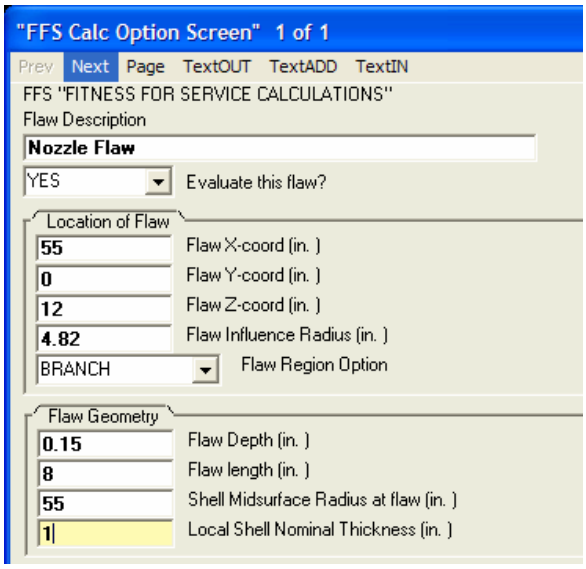


Figure 2-15: Sample Flaw Influence Radius and Branch Region

In all cases, you should make sure that the nodes in the area of the model containing the flaw are highlighted when you select the flaw name from the FFS menu when the model is plotted. The FFS menu is shown whether the model is “Plotted” or “Prepared” as shown below:

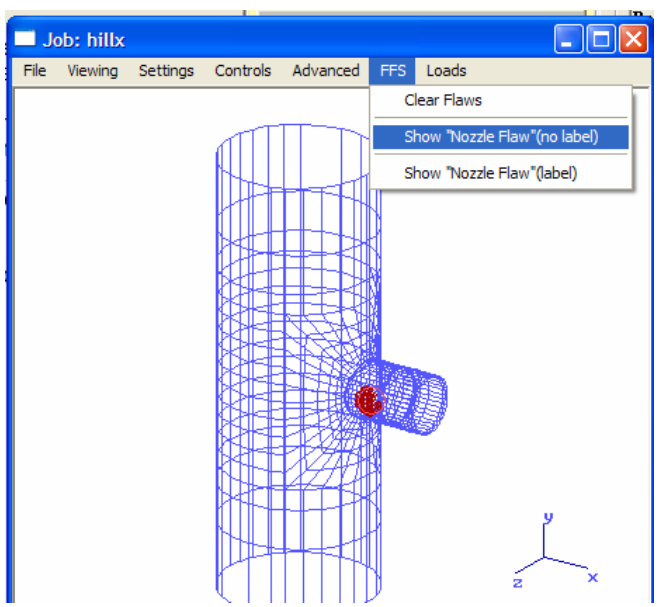


Figure 2-16: Flaw Zone Display Option

The benefits of this approach include:

- Easy to identify flaw zone
- Conservative FFS evaluation.
- All operating loads – including pressure, weight and operating loads are included in the analysis.

More comprehensive evaluations can be performed. The stress orientation display panel is shown below with results:

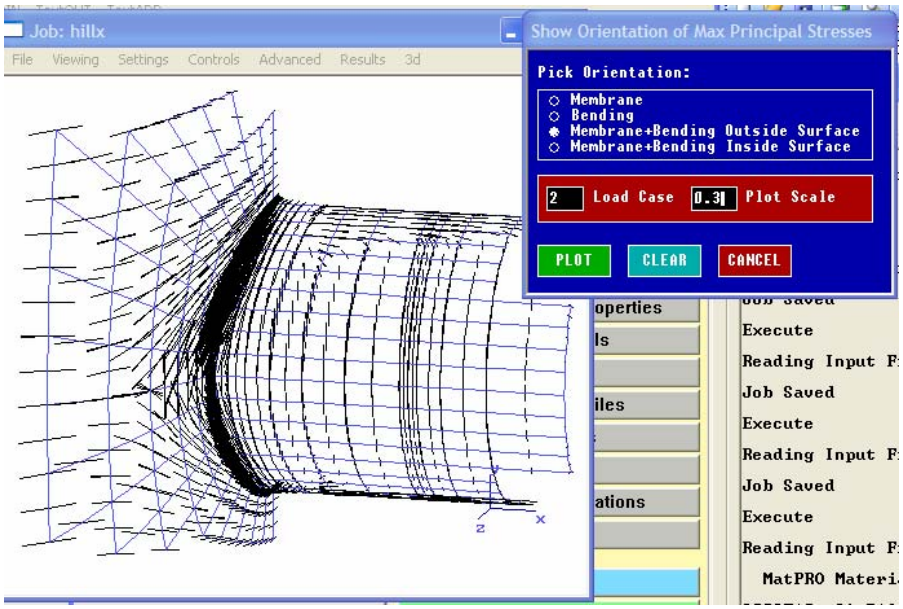


Figure 2-17: Bending + Membrane Outside Surface Orientation Plot

The local stress panel and options are shown below:

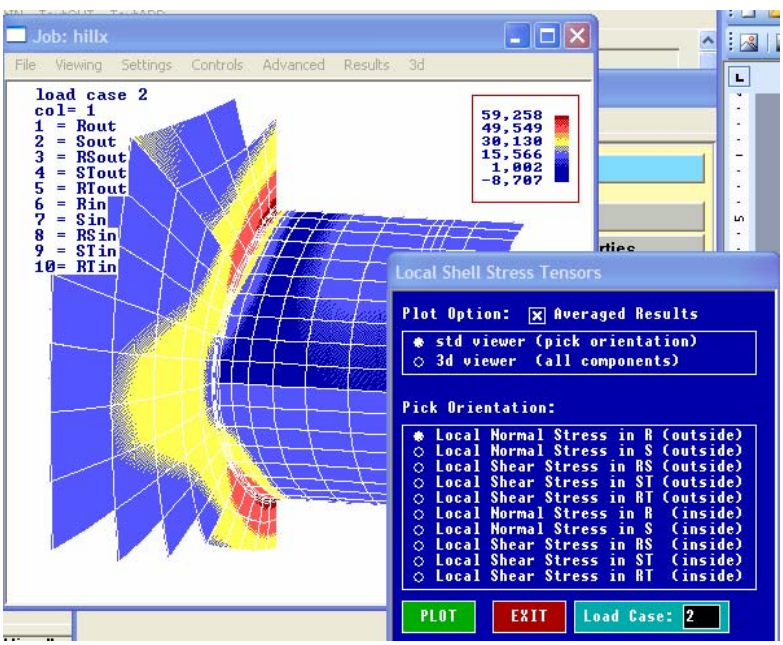
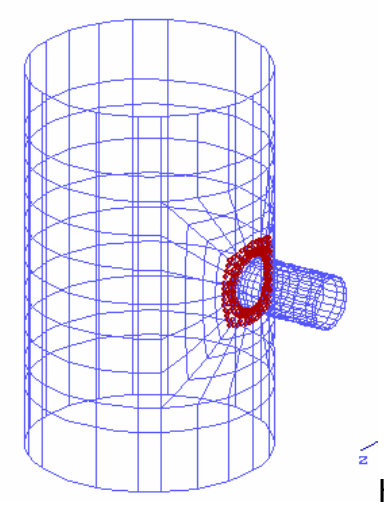


Figure 2-18: Local Stress Contour Option Menu

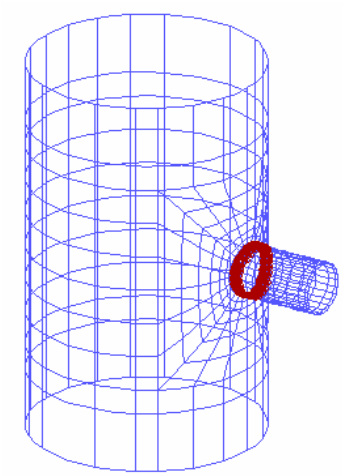
FE/Pipe FFS Templates and Defined Nodal Areas for Flaws

FE/Pipe templates have predefined areas for flaws as indicated below for each template.

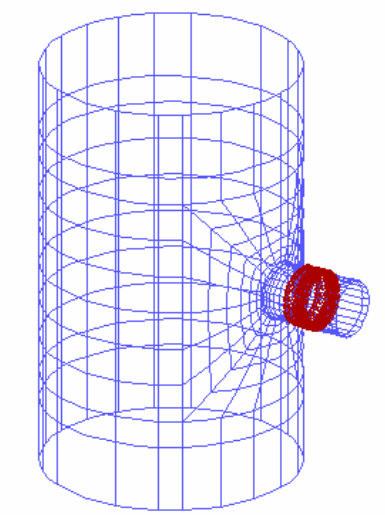
Unreinforced Fabricated Tee



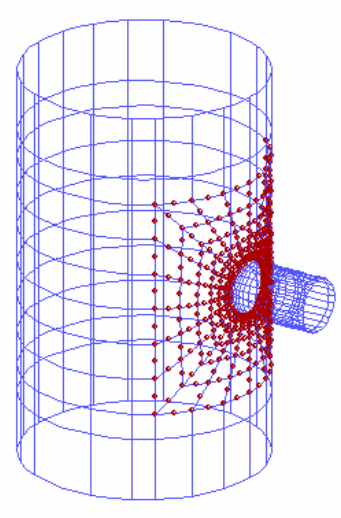
HDR_AT_WELD



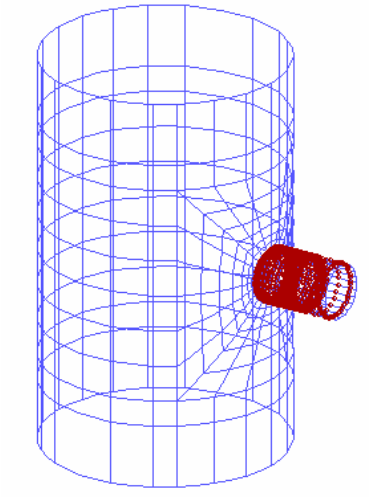
BR_AT_WELD



BR_TAPER – Used when there is a thickness or diameter transition some distance along the nozzle

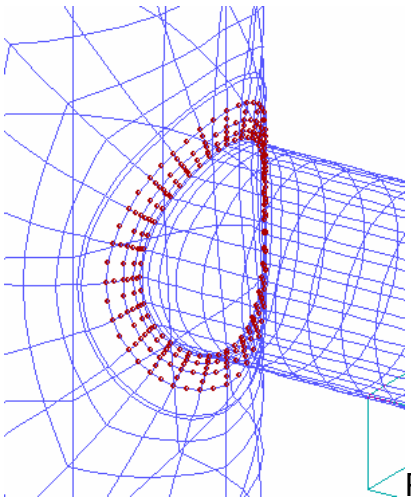


HEADER

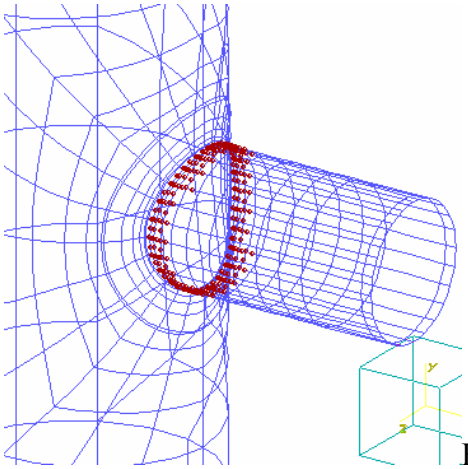


BRANCH

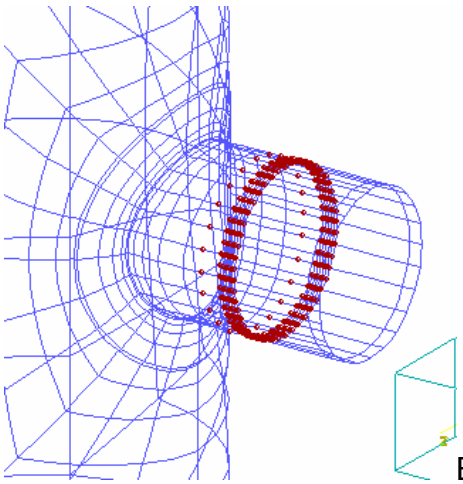
Reinforced Fabricated Tee (Repad)



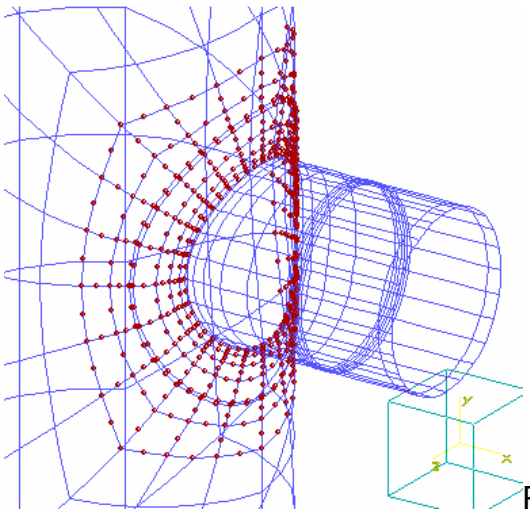
PAD_BRWELD



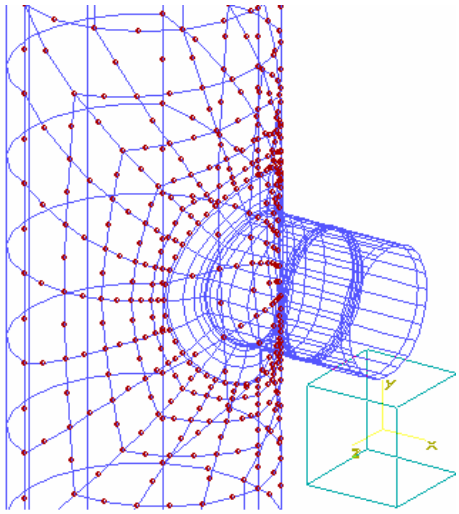
BR_WELD



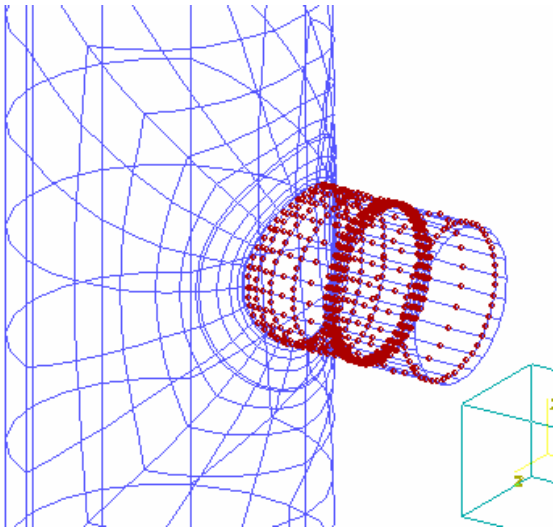
BR_TAPER



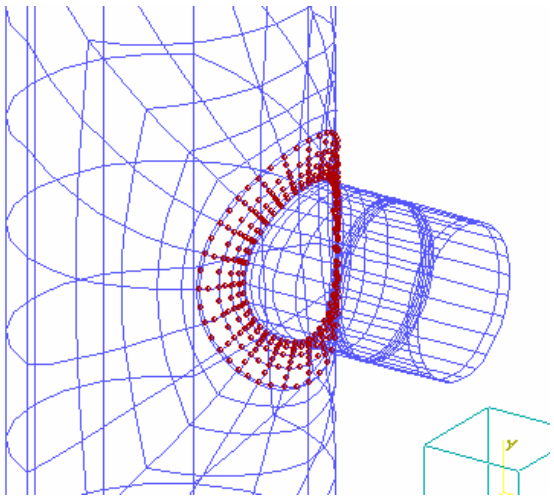
PAD_HDRWELD



HEADER



BRANCH



PAD_ONLY

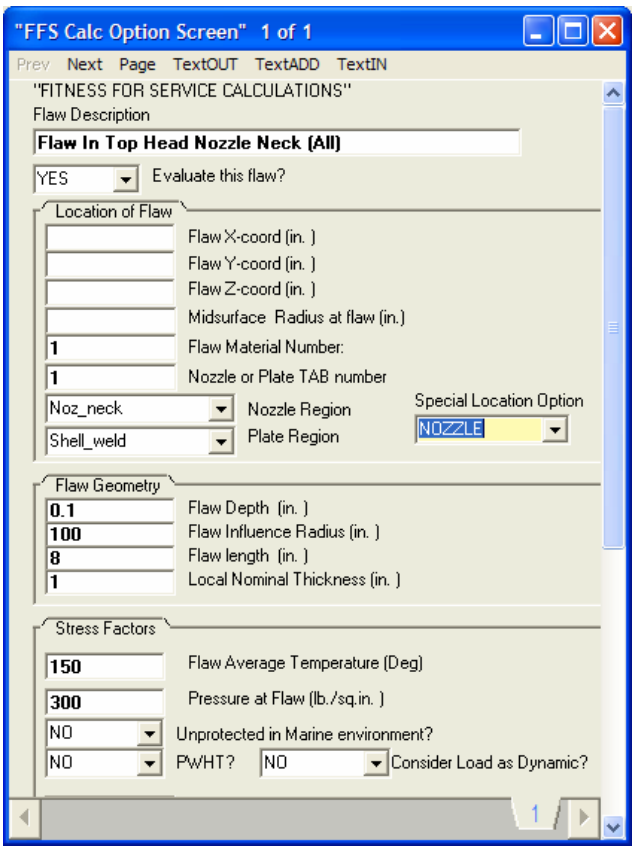
Nozzles-Plates and Shells

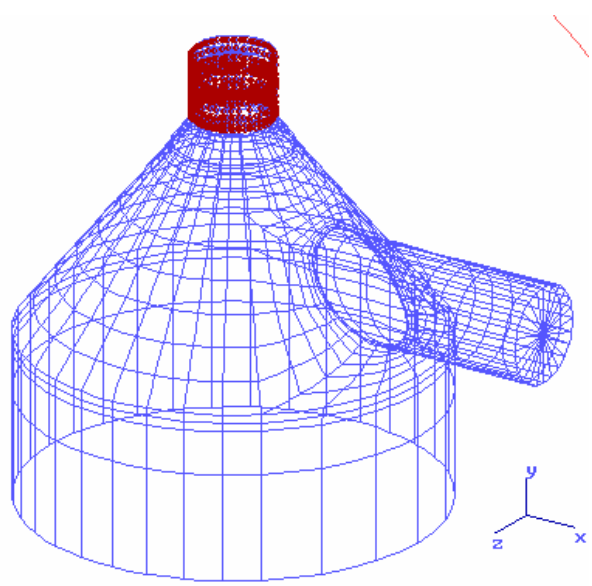
The Nozzles-Plates & Shells template user has more options than all other templates because of the varieties of geometries that can be constructed. The user can request, “AUTOSEARCH” and the algorithm will inspect the entire geometry for the specified flaw size and orientation.

The “INSPHERE” option can be used to use only the flaw influence sphere to find any nodes within the sphere, or the user can specify an influence sphere, and a specific location in any nozzle or plate.

The “ALL_PLATES_RGN” option can be used to inspect all plates structures in the model for the flaw size specified.

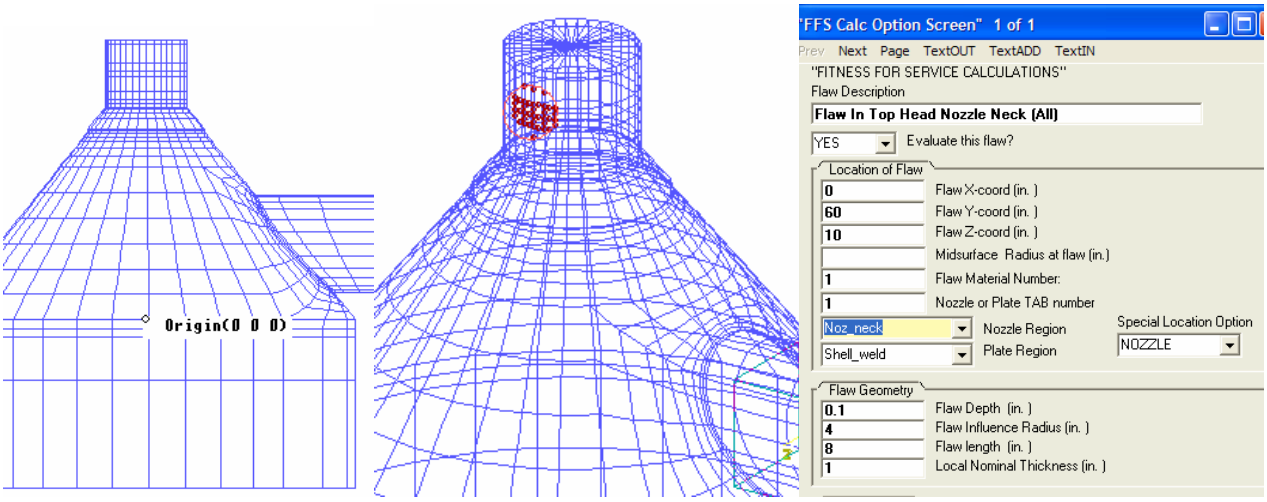
Various combinations are demonstrated below for several geometries.

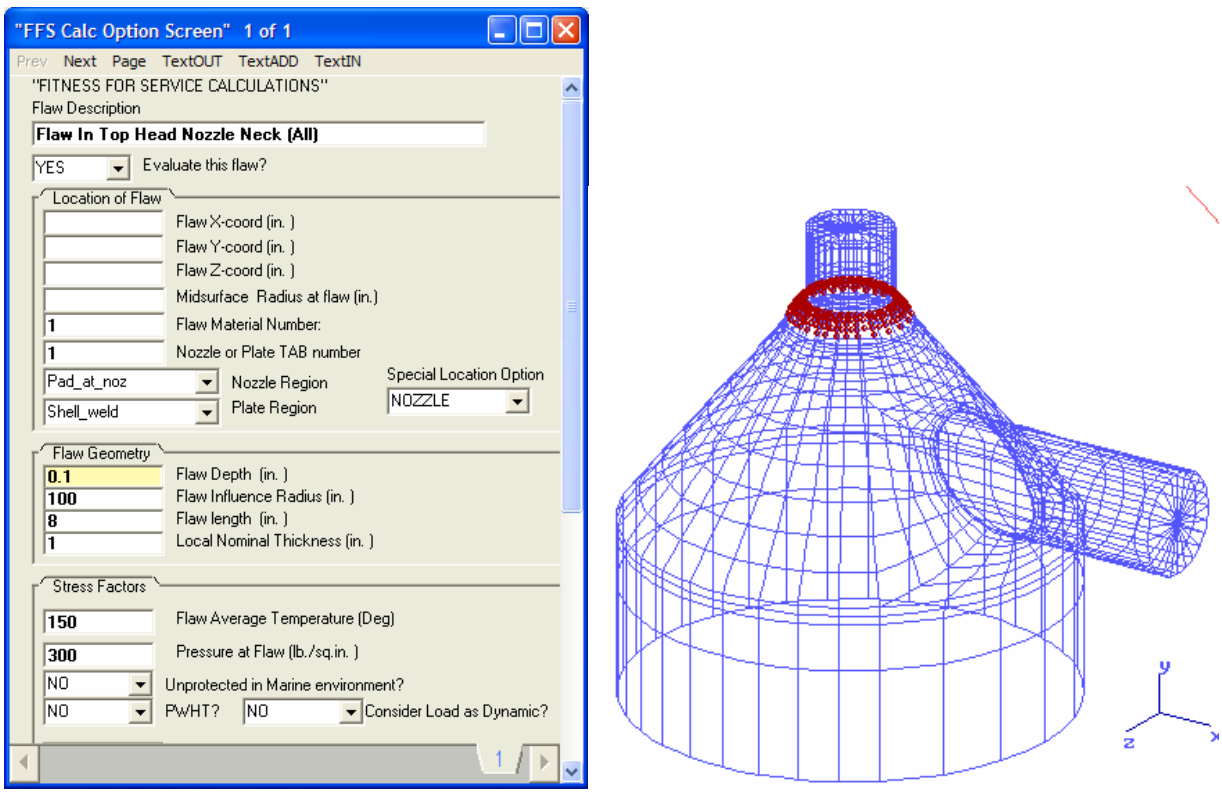




User wants the entire top nozzle evaluated for a 0.1” deep by 8 inch long flaw. The “Special Location Option” is set to “NOZZLE”, the Nozzle Region is set to “Noz_neck”, and the Nozzle Number is set to the top nozzle number, which is 1.

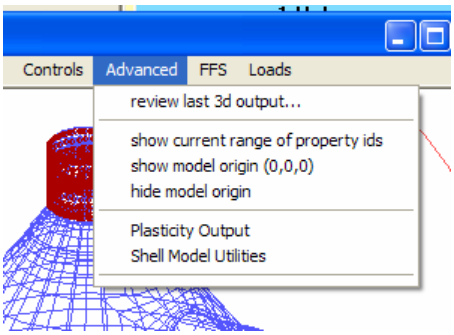
To check a small corroded zone on the side of the top nozzle, the user would specify the nozzle number, the Special Location Option “NOZZLE,” and the Nozzle Region “Noz_neck”.

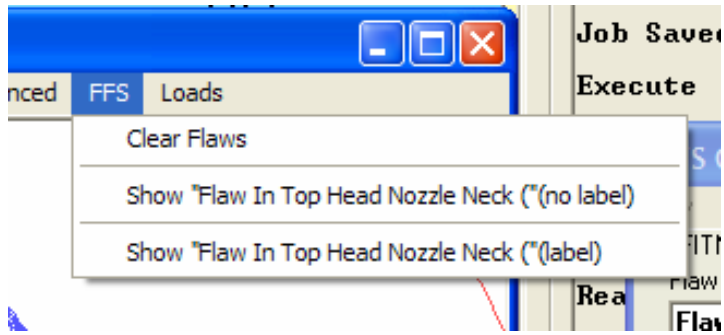




The “Pad_at noz” the node region is shown in the figure above. If the user is unsure what definitions refer to particular parts of the geometry, he is encouraged to make an educated guess, and then to plot the model, and display the flaw zone. From visual inspection the user will know if they have selected the correct part of the model. If no nodes appear for a particular flaw, the “Flaw Influence Radius” should be increased to a dimension that is about an order of magnitude larger than the biggest model dimension. If no nodes appear after this change is made, then the “Special Location Option”, “Nozzle Region”, “Plate Region”, or “Node Number” are entered incorrectly. Usually a few iterations are required to get exactly what is desired.

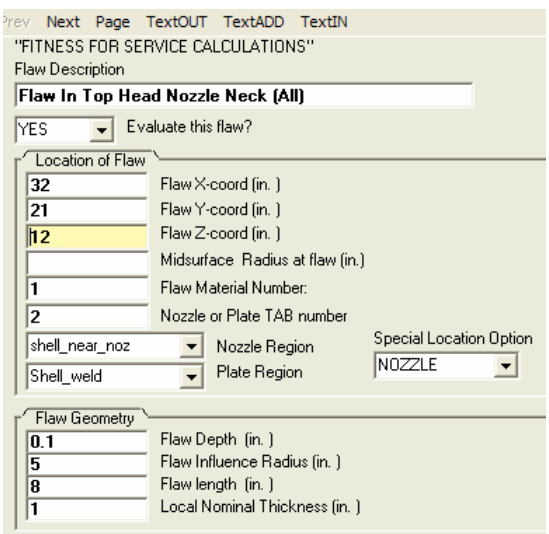
Under the “Advanced” menu option there is the possibility to show the model origin. It is from this origin that the user must define the center of the flaw influence cylinder. Under the “FFS” menu option, the user will find two flaw lists. The first draws each node in a particular region, but does not label the region. The second, draws ach node in a particular region, and labels the region for reference.

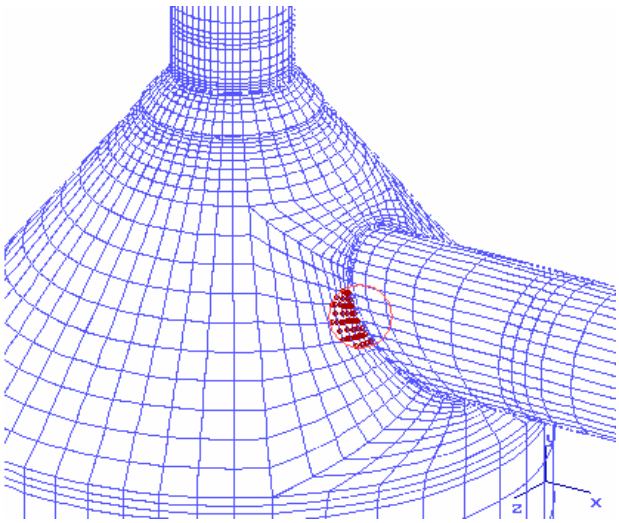




The recommended way to focus on small areas is to use either the “INSPHERE” option, where every node inside the “flaw influence sphere” is selected,

The input to evaluate a small local thinned area adjacent to the horizontal nozzle #2 in a conical head is shown below:





Example Problems and Discussion

The following example problems were developed from a number of sources. Each source reference is provided at the end of the example. In some cases the examples have been expanded and clarified where sufficient information was not provided in the original documentation.

The examples demonstrate the realistic conservatism provided in the Fitness For Service method as applied in the PRG Suite of Fitness for Service tools.

API 579 Problem 4.11.1 Example Problem 1 – Localized Corrosion

A region of localized corrosion has been found on a pressure vessel during a scheduled turnaround.

Design Conditions = 300 psi @ 350F
Inside Diameter = 48 in.
Fabricated Thickness = 0.75 in.
Uniform Metal Loss = 0.0 in.
FCA = 0.1 in. (Future Metal Loss [Corrosion Allowance])
Material = SA 516 Grade 70
Weld Joint Efficiency = 0.85

The corroded area was NOT in the circumferential weld seam.
Effective Longitudinal Flaw Length = 9.75 in.
Effective Circumferential Flaw Length = 9.0 in.
Minimum thickness = 0.45 in.

The local metal loss area passes through a longitudinal weld seam.

Thickness Measurements for the local metal loss are given below. There are 8 measurement points in the longitudinal (C) direction, and 7 measurement points in the circumferential (M) direction.

									Circ
	C1	C2	C3	C4	C5	C6	C7	C8	CTP
M1	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75
M2	0.75	0.48	0.52	0.57	0.56	0.58	0.60	0.75	0.48
M3	0.75	0.57	0.59	0.55	0.59	0.60	0.66	0.75	0.55
M4	0.75	0.61	0.47	0.58	0.36	0.58	0.64	0.75	0.36
M5	0.75	0.62	0.59	0.58	0.57	0.48	0.62	0.75	0.48
M6	0.75	0.57	0.59	0.61	0.57	0.56	0.49	0.75	0.49
M7	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75
Long.									
CTP	0.75	0.48	0.47	0.55	0.36	0.48	0.49	0.75	

Yellow Values are the minimum values for each row.

From the measurement grid, it is desired to find the most critical flaw. This is the longest and deepest flaw. If it is not clear whether the longitudinal or circumferential flaws are deepest, then both should be analyzed. For a cylindrical geometry subject to pressure loads only, a 2" long longitudinal flaw is more critical than a 2" long circumferential flaw because the longitudinal flaw is opened by the hoop pressure stress PD/2t, while circumferential flaw is opened by the longitudinal pressure stress PD/4t. When external loads are applied, and the longitudinal + torsional stress is greater than PD/4t, then a 2" long circumferential flaw becomes more critical than a 2" long longitudinal flaw.

NozzlePRO has a “measurement grid” processor that makes the flaw length and average depth calculations. This calculator is shown below for the flaw described above.

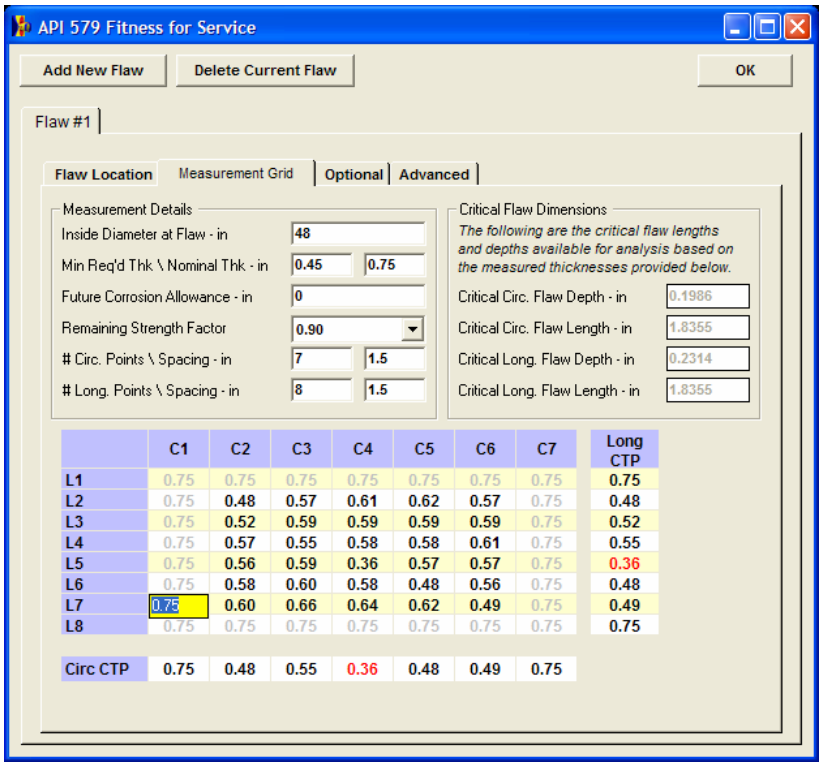


Figure EX1-1: NozzlePRO Measurement Grid Calculator

When using the NozzlePRO Measurement Grid Calculator, NozzlePRO always uses the maximum length, whether it is circumferential or longitudinal, and the maximum depth, whether it is in the longitudinal or circumferential directions. This may be excessively conservative, and the user not wishing to make this assumption is free to alter these dimensions on the NozzlePRO main flaw page, or when entering the flaw data into MatPRO.

NozzlePRO reports the both the critical longitudinal and circumferential flaws for the user to evaluate, but as stated above, will use the most conservative flaw length and depth from either if the user does not override the flaw size.

Calculations:

Average from Longitudinal CTP = 0.54125
Average from Circumferential CTP = 0.5514

The longitudinal flaw size is longer and in the most critical stress state and so will be evaluated at 0.54125 in.

The nominal hoop stress is PD/2t = (300)(48) / (2)(0.75-0.1) = 11,077 psi
The nominal longitudinal stress is PD/4t = (300)(48) / (4)(0.75-0.1) = 5,538 psi.

The hoop stress is NOT within the allowable (as also reflected by the API 579 level 1 and level 2 assessments.)

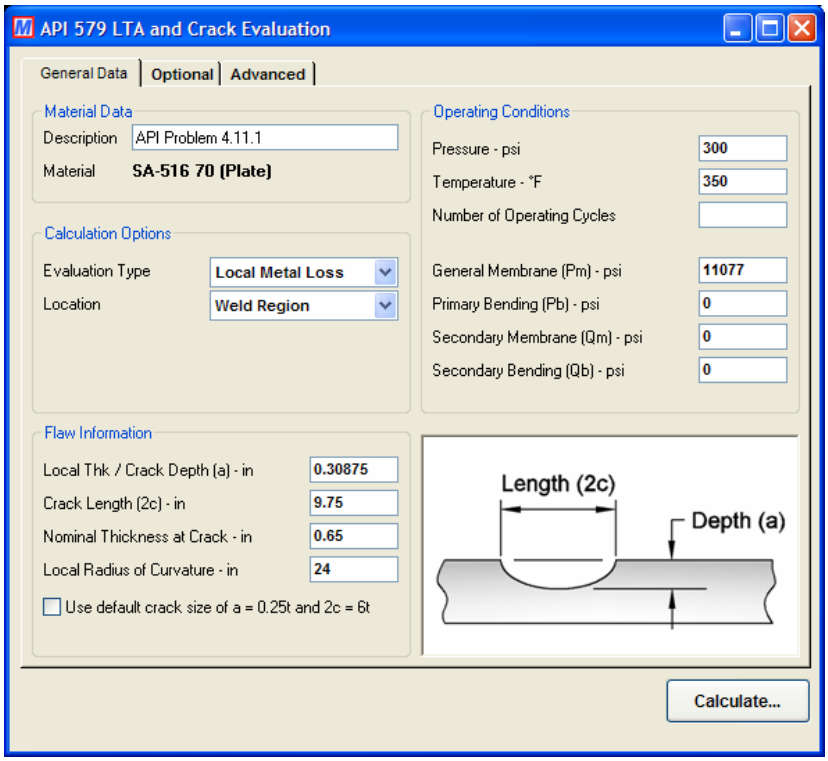


Figure EX1-2

Note that the weld joint efficiency is on the optional screen and defaults to 0.7. For this problem the weld joint efficiency is 0.85 and should be changed. The optional data form is shown below.

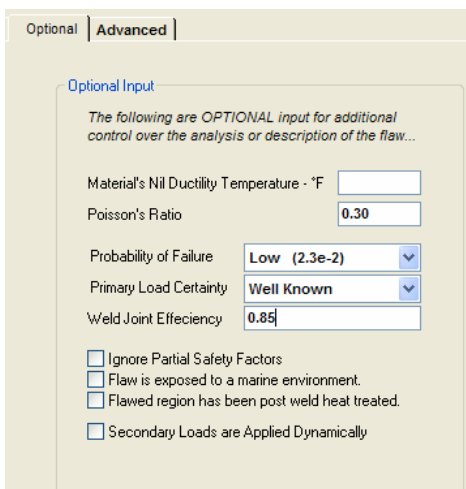


Figure EX1-3

```
MatPRO 1.0 - PAULIN RESEARCH GROUP     HOUSTON, TX
==========================================================================================
```

```
Time Stamp : 1/27/2007  2:41:36 PM

Materials Database : "ASME II-D, Table 2A" (2006)

 API 579 Fitness for Service Evaluation
 ----------------------------------------------

 Conservative assumptions were made when implementing the
 fittness for service rules of API579 Sections 5.0 and 9.0.
 It is the users responsibility to review and check the
 results printed herein to verify that they apply and are
 valid for the particular problem studied.


 Material = SA-51670 Carbon steel Plate

 Yield Stress at Room Temperature                =      38.000 ksi      262.010 MPa
 Yield Stress at Operating Temperature           =      33.050 ksi      227.880 Mpa
 Flow Stress at Operating Temperature            =      53.825 ksi      371.120 MPa
 Flow Stress at ROOM Temperature                 =      54.673 ksi      376.973 MPa
 Modulus of Elasticity at Room Temperature       =   29400.000 ksi   202713.000 MPa
 Modulus of Elasticity at Operating Temperature  =   28100.000 ksi   193749.500 MPa

 Internal Pressure                               =     300.000 psi        2.069 MPa
 Operating Temperature                           =     350.000 degF      176.667 degC

 Local Primary Membrane Stress in Area of Flaw   =      11.077 ksi       76.376 MPa
 Local Primary Bending Stress in Area of Flaw    =       0.000 ksi        0.000 MPa
 Local Secondary Membrane Stress in Area of Flaw =       0.000 ksi        0.000 MPa
 Local Secondary Bending  Stress in Area of Flaw =       0.000 ksi        0.000 MPa

 Initial Crack Depth                             =       0.309 in.        7.842 mm
 Initial Crack Half-Length                       =       4.875 in.      123.825 mm
 Component Wall Thickness at Flaw                =       0.650 in.       16.510 mm
 Component Inside Radius at Flaw Location        =      24.000 in.      609.600 mm

 Flaw is in an area that contains a weld or HAZ.

 Longitudinal Weld Joint Efficiency              =       0.850
 Probability of Failure                          = 0.023000000

 Coefficient of Variation                        =       0.100
 (Primary Loads and Stresses are
  computed and well known.)

 Poissons Ratio used in this analysis            =       0.300


 API 579 Section 5.0 Assessment for Local Metal Loss
 ---------------------------------------------------

 5.54 Membrane  Stress due to Primary Loads      =      34.328 ksi      236.693 MPa
 5.54 Allowable Stress due to Primary Loads      =      24.788 ksi      170.910 MPa

 Primary Membrane Stress at Flaw EXCEEDS limit   =     138.490 %

 5.54 Membrane  Stress due to Secondary Loads    =       0.000 ksi        0.000 MPa
 5.54 Allowable Stress due to Secondary Loads    =      49.575 ksi      341.820 MPa

 Secondary Membrane Flaw Stress WITHIN allowable =       0.000 %
```

Iterating through the calculation permits a re-rated pressure to be developed. In this example the pressure effect on stress is assumed to be linear, and so a 200 psi pressure (reduction from 300 psi) would result in the following membrane stress:

(11077)(200/300) = 7,384 psi.

and this value can be seen to result in an acceptable API 579 evaluation.

Problem 4.11.2 Example Problem 2 API 579 – USING FE/PIPE

A localized region of corrosion on a 2:1 elliptical head has been found during an inspection. The corroded region is within the spherical portion of the elliptical head.

Design Conditions	= 2.068 MPa @ 340 deg C (300 psi @644 degF)
Head Inside Diameter	= 2032 mm (80 in.)
Head Outside Diameter	= 2070 mm (81.5 in.)
Nominal Thickness	= 19mm (0.748 in.)
Metal Loss	= 0 mm
Future Corrosion	= 3 mm (0.118 in.)
Material	= SA 516 Grade 70
Joint Efficiency	= 1.0 (Seamless Head)

The grid and inspection data are included in the chart below. The grid spacing is 100 mm.

									Circ
	C1	C2	C3	C4	C5	C6	C7	C8	CTP
M1	20	20	19	20	20	19	20	20	19
M2	20	20	20	19	19	19	20	20	19
M3	19	19	19	19	19	19	19	20	19
M4	20	19	19	17	17	18	19	19	17
M5	19	19	19	17	14	15	19	19	14
M6	19	19	20	17	15	16	19	19	15
M7	20	20	19	19	20	19	19	19	19
M8	20	20	19	18	19	19	20	19	19
Meridonal CTP	19	19	19	17	14	15	19	19	

The average CTP thickness in the meridonal direction is: 17.625 mm (0.6938 in.)
The average CTP thickness in the longitudinal direction is: 17.625 mm (0.6938 in.)

The length of the flaw is (8)(100) = 800 mm. (31.5 in)

The nominal wall for analysis should be 19-3 = 16 mm (0.63 in.)
The flaw depth “a” is found from:

Average thickness with FCA removed: 17.625 – 3 = 14.625 mm (0.575 in.)
Flaw Depth “a” = 16 – 14.625 = 1.375 mm (0.05413 in.)

The FE/Pipe model of this geometry with the flaw identified is shown below:

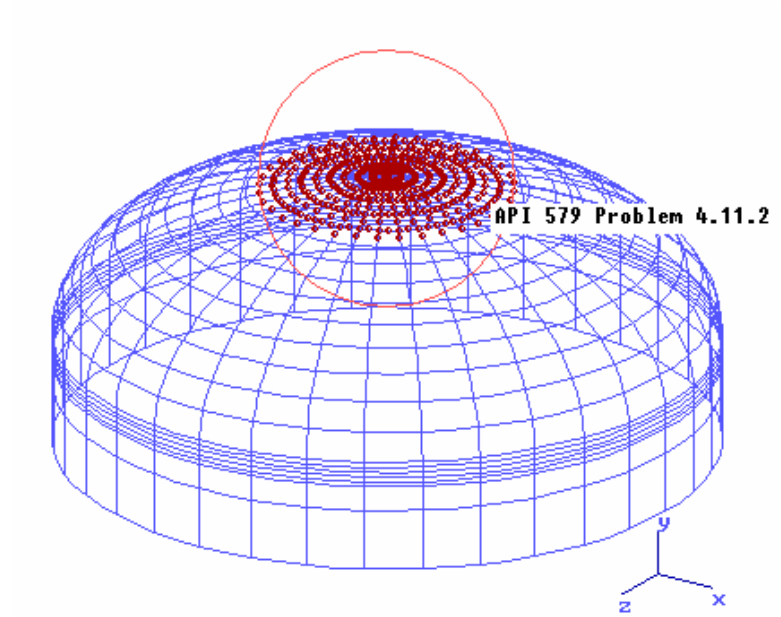


Figure EX2-1

Note that you can shift the flaw area to any geometric location as shown in the figures below:

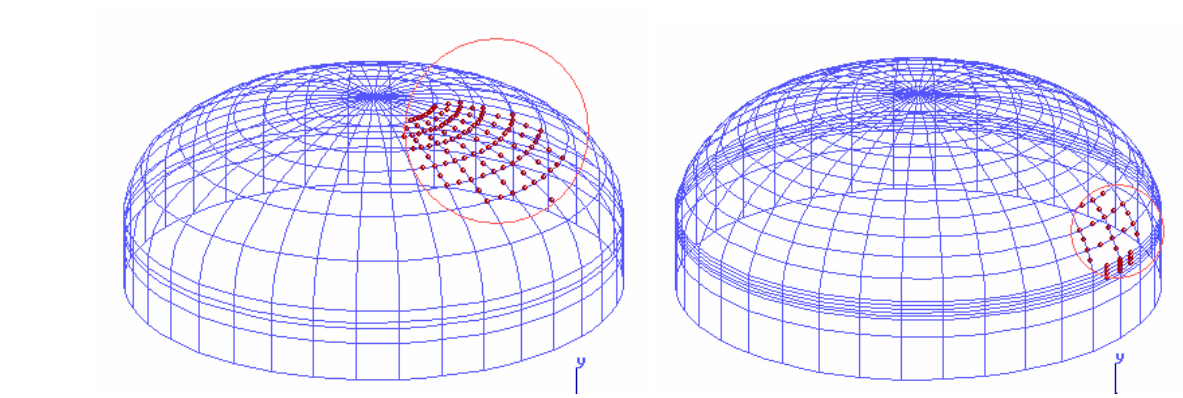


Figure EX2-2

When showing FFS points on a plot, a red circle is shown that defines the sphere specified in the FFS data form. The flaw shown on the right in the figure above is shown in three views below. Note how the red circle in each view defines the region where the flaw exists. The user is encouraged to estimate the flaw zone and then iterate through various plots like those shown above and below to be sure that at least a single node in the flaw zone will contain the highest stress in the area where the crack or local thin area occurs. The FE/Pipe FFS processor will trap the highest nodal membrane and bending stresses in this zone and assume the worst flaw orientation possible to perform the FFS evaluation. User's wishing to take a less conservative approach can compute the orientation of the stress relative to the precise flaw direction and enter membrane and bending stresses directly in MatPRO.

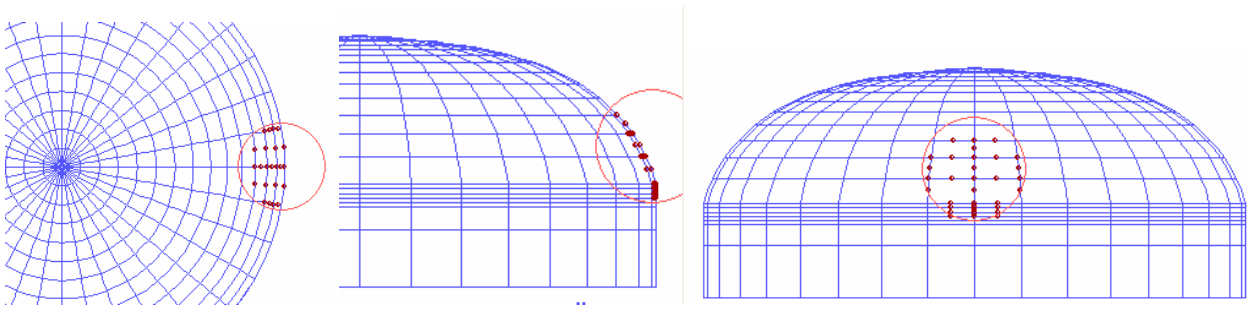


Figure EX2-3

The first step is to identify the flaw. This is done by selecting the FFS button from the Nozzles-Plates&Shells *General* Screen and then entering the approximate coordinate for the center of the flaw. This may be found by first plotting the origin of the model geometry from the "Advanced" screen as shown below.

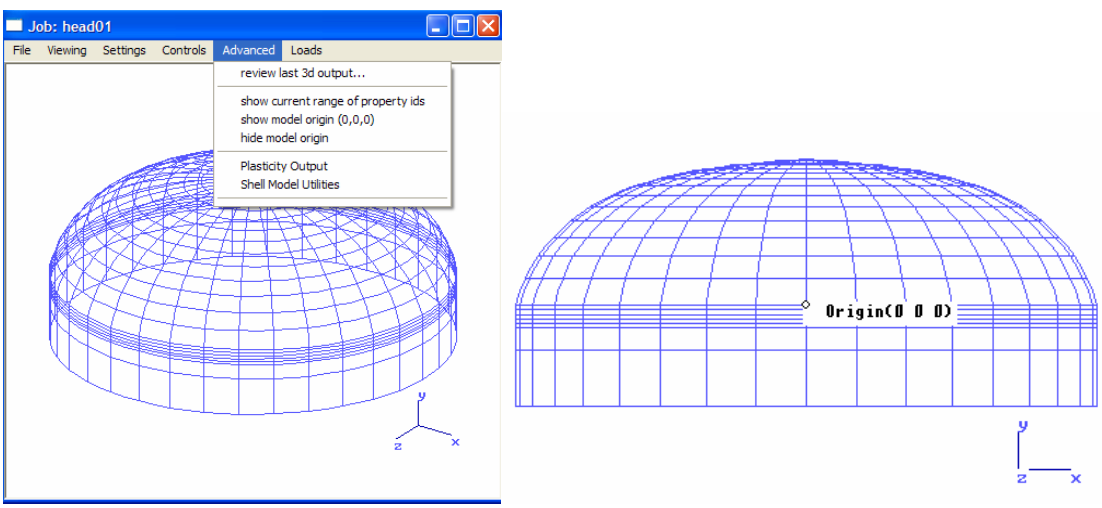


Figure EX2-4

Since the flaw is in the spherical portion of the elliptical head, and the head ID is 80", the flaw will be approximately 20" up from the origin. The midsurface radius of the shell at the flaw location must be entered so that the Folias local bulging factor can be calculated. In the spherical portion of the elliptical head, the radius of the shell will be approximately 80 inches. This input and output for this porblem is shown below:

Both Level 1 and Level 2 assessments in API 579 were NOT passed per API 579. The procedure implemented in FE/Pipe shows a 127% violation.

```
FFS for Flaw# 1 for Region:Elliptical Head

API 579 Fitness for Service Evaluation
-------------------------------------------------

Conservative assumptions were made when implementing the
fittness for service rules of API579 Sections 5.0 and 9.0.
```



```
It is the users responsibility to review and check the
results printed herein to verify that they apply and are
valid for the particular problem studied.

Descr:  API 579 Problem 4.11.2

Yield Stress at Room Temperature                =      38.000 ksi
Yield Stress at Operating Temperature           =      28.308 ksi
Flow Stress at Operating Temperature            =      38.308 ksi
Flow Stress at ROOM Temperature                 =      48.000 ksi
Modulus of Elasticity at Room Temperature       =   29400.000 ksi
Modulus of Elasticity at Operating Temperature  =   26059.998 ksi

Internal Pressure                               =     300.000 psi
Operating Temperature                           =     644.000 degF

Local Primary Membrane Stress in Area of Flaw   =      22.662 ksi
Local Primary Bending Stress in Area of Flaw    =       1.444 ksi
Local Secondary Membrane Stress in Area of Flaw =       0.000 ksi
Local Secondary Bending  Stress in Area of Flaw =       0.000 ksi

Initial Crack Depth                             =       0.054 in.
Initial Crack Half-Length                       =      15.750 in.
Component Wall Thickness at Flaw                =       0.630 in.
Component Inside Radius at Flaw Location        =      80.000 in.

Flaw is in base metal removed from welds.

Probability of Failure                          = 0.000001000

Coefficient of Variation                        =       0.100
(Primary Loads and Stresses have
 signifiant uncertainty due to random
 loading or modeling approximations.

Poissons Ratio used in this analysis            =       0.300


API 579 Section 5.0 Assessment for Local Metal Loss
---------------------------------------------------

5.54 Membrane  Stress due to Primary Loads      =      26.898 ksi
5.54 Allowable Stress due to Primary Loads      =      21.231 ksi

Primary Membrane Stress at Flaw EXCEEDS limit   =     126.694 %

5.54 Membrane  Stress due to Secondary Loads    =       0.000 ksi
5.54 Allowable Stress due to Secondary Loads    =      42.462 ksi

Secondary Membrane Flaw Stress WITHIN allowable =       0.000 %
FFS for Flaw# 1 for Region:Elliptical Head
```


Example 4.11.3 Example Problem 3 API 579 – USING FE/PIPE

A region of corrosion on a 12" Class 300 # long weld neck nozzle has been found during inspection. The corroded region includes the nozzle bore and a portion of the vessel cylindrical shell (see inspection data).

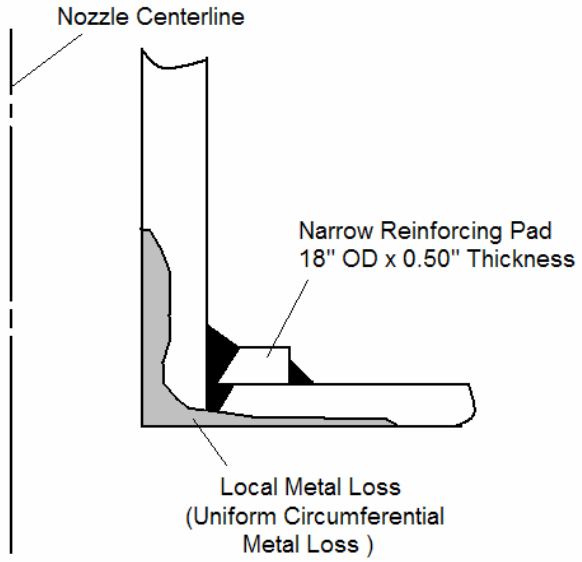


Figure EX3-1

Design Conditions	= 185 psig @ 650 F
Shell Inside Diameter	= 60 in.
Shell Thickness	= 0.60 in.
Shell Material	= SA 516 Gr. 70
Shell Weld Efficiency	= 1.0
Shell FCA	= 0.125
Nozzle Inside Diameter	= 12.0 in.
Nozzle Thickness	= 1.375 in.
Nozzle Material	= SA 105
Nozzle Weld Joint Efficiency	= 1.0
Nozzle FCA	= 0.125
Reinforcing Pad Material	= SA 516 Gr. 70
Reinforcing Pad OD	= 18" x 0.5" Thick
Nozzle and Pad Fillet Leg	= 0.375 in.

From the Inspection Data: Average Shell Thickness in Nozzle Zone = 0.50 in.
Average Nozzle Thickness in Nozzle Reinforcement Zone = 0.9 in.

Corrosion is uniform for each inspection plane.

In this case the FCA (Future Corrosion Allowance) is 0.125". The FCA is considered a cumulative loss of life as a function of time. The calculation below will remove the FCA from the actual wall thickness before the stress calculation is made, as this is a conservative approach for a relatively simple nozzle geometry. This approach may not be conservative for fixed tubesheet heat exchangers or other components where one member interacts with others that may or may

not be weakened by future corrosion causing a redistribution of load. Calculations may also be made to include the gradual effect of wall loss on the fatigue life. This approach is found in API 530.

The level 2 assessment performed in API 579 for this intersection showed that the nozzle was ***not*** acceptable for continued operation.

The origin for this nozzle is shown in the plot below:

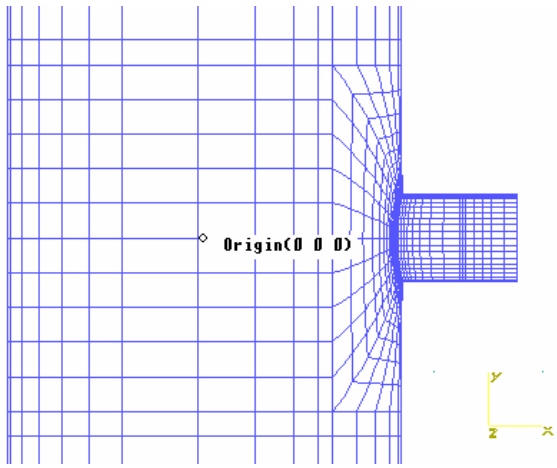


Figure EX3-2

The "x" offset will be about half the shell diameter = 30", and the radius of the flaw zone is larger than the reinforcing pad, so 14" will be used. The input to describe this flaw location in FE/Pipe is shown below. There will be three flaw zones specified since the metal loss area is so large:

1) Nozzle
2) Pad
3) Shell Surface

User's are strongly cautioned when evaluating local metal loss in the vicinity of reinforcing pads. High residual stresses may exist, inspection techniques are limited, fit-up quality is unknown, and attachment weld quality inspection may be restricted.

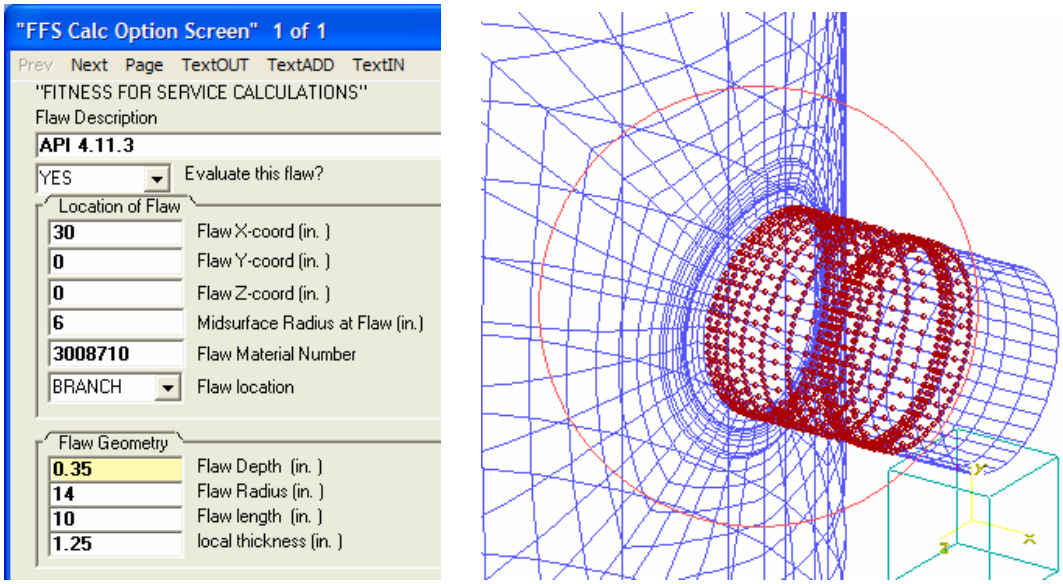


Figure EX3-3: Nozzle Flaw Zone Definitions

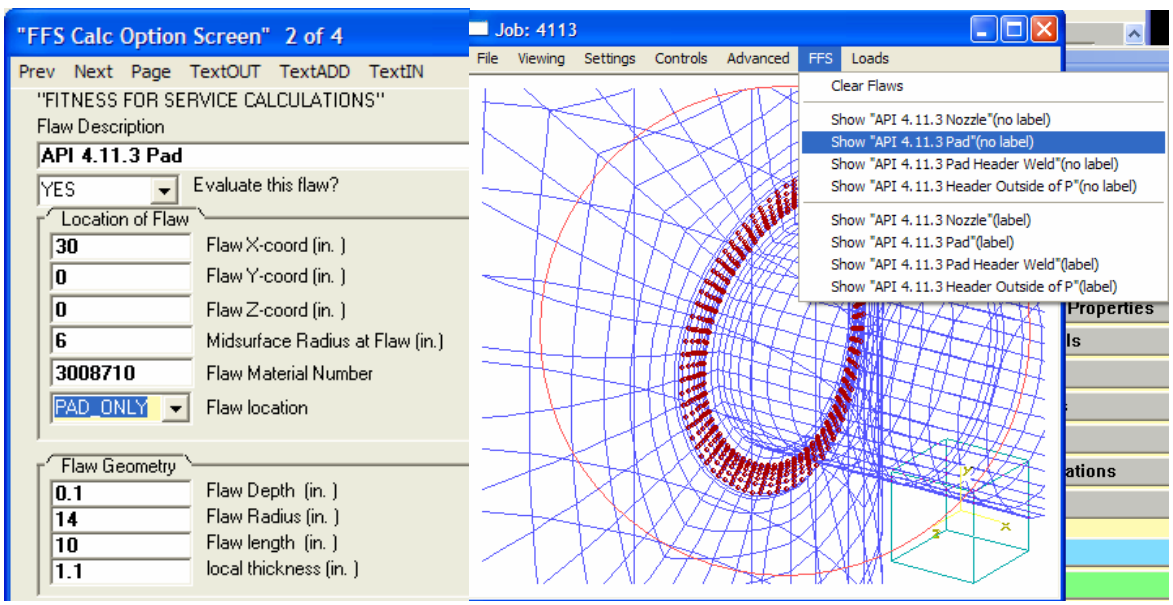


Figure EX3-4: Pad (Note how the user entered Flaw Description appears in the plot menu.)

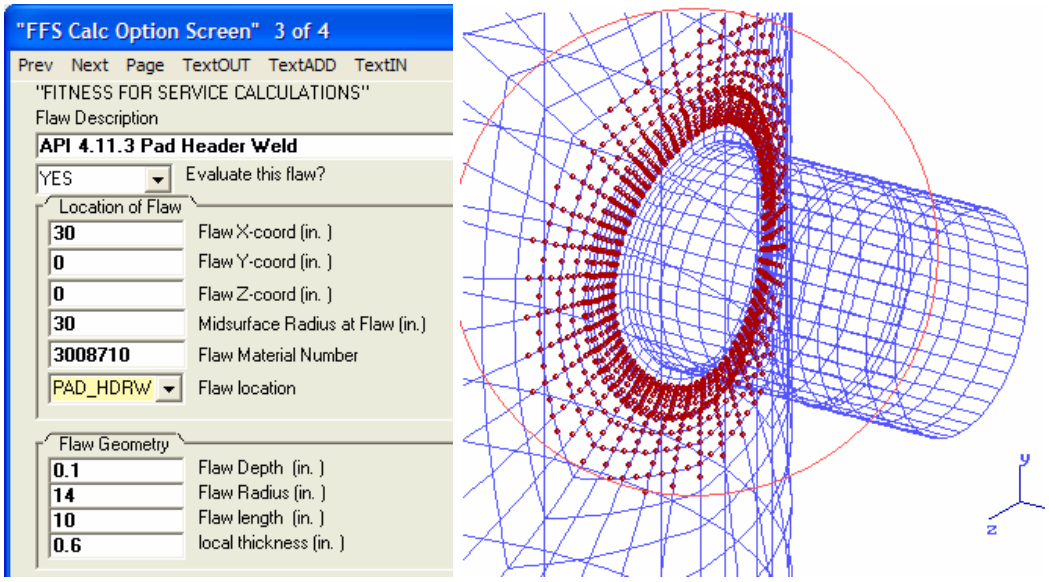


Figure EX3-5: Header in Pad Weld and Zone outside of Pad Weld

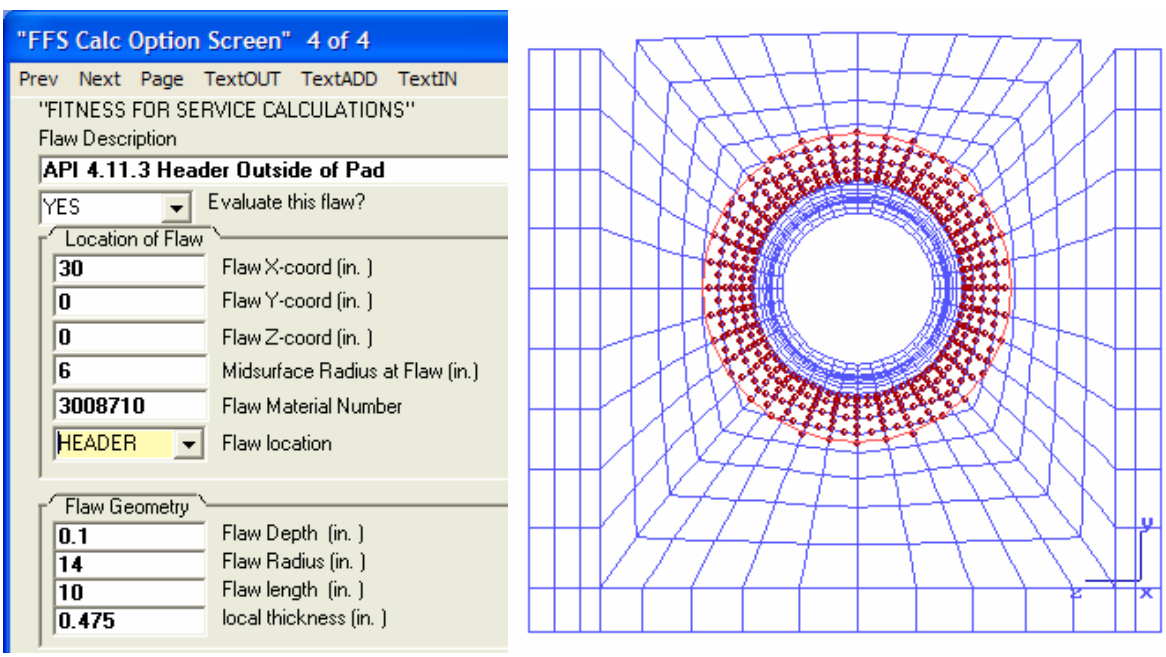


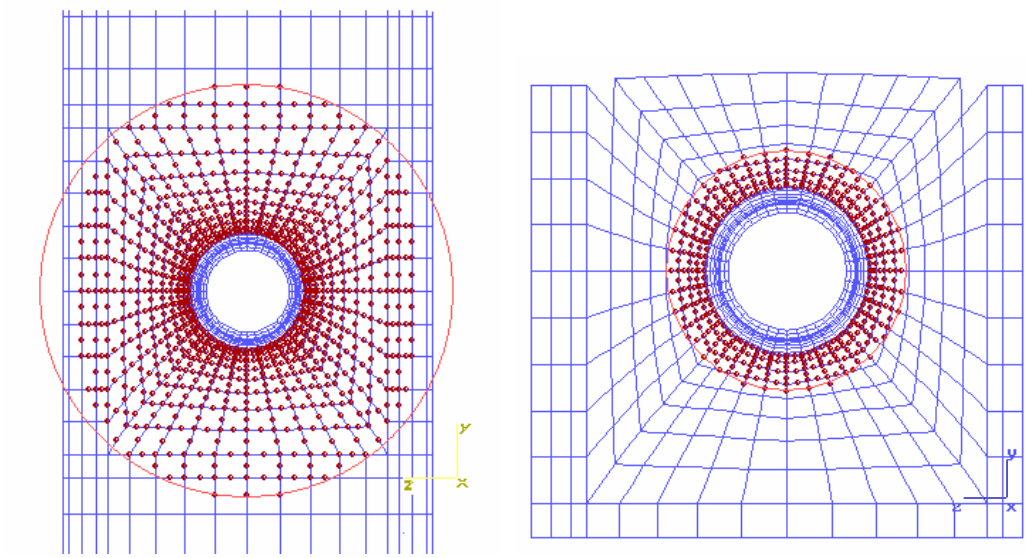
Figure EX3-6: Header – Outside of Pad Zone

Nodes in the Header outside of the pad zone will share areas, since the node region around the weld are within 3t of the weld itself. Individual nodes will exist in multiple flaws.

The worst case is evaluated. FE/Pipe evaluates the nodal stresses for each flaw zone, guaranteeing that the worst configuration is analyzed. (See the results below.) The user will see that flaws on FFS pages 2, 3 and 4 (above), overlap node areas and regions, and that a single region's result can appear under multiple flaw descriptions.

If the flaw radius: 

was larger, there would be more points in the header outside of the pad zone, since the number of nodes in this region is the subset of all nodes in the header and the flaw radius. A larger flaw radius is shown in the figure below for information:



Nodes In 34" Flaw Influence Sphere **Nodes in 14" Flaw Influence Sphere**

Figure EX3-7: Example Flaw Influence Sphere Sizes

There were four flaw locations that seemed to apply to this nozzle when the analysis was started. (The flaw locations are picked from the input FFS menu shown below.)

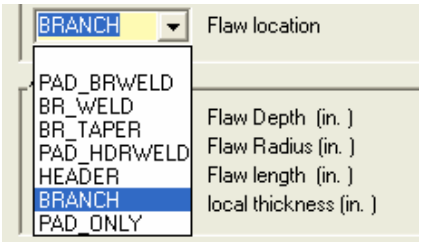


Figure EX3-8

The four selected for this local thinned area were:

1-BRANCH – This includes all the nodes in the branch that are in the locally thinned area. The most highly stressed nodes govern the conservative analysis used by FE/Pipe, and so only that section of the metal loss area that contains the most highly stressed nodes need to be included.

2-PAD_ONLY – This includes the nodes in the pad area. The default FE/Pipe model for these areas includes a locally thickened area since the highest stresses are usually in the nozzle or header shell adjacent to the pad when the high stresses are due to external loads. As can be seen by the stress plots the highest stress in the nominal thickness geometry is in the header/shell area adjacent to the pad.

3-HEADER AND PAD WELD – This includes the weld zone and 3t on either side of the weld zone, perpendicular to the weld. For this problem, the 3t area includes all of the pad and the area in the header that would otherwise have been in a header only zone. Outside the 3t zone, there are no nodes. Because of the size of this area, all nodes in the header and in the pad will be included in this calculation. The only difference between nodes evaluated in the HEADER AND PAD, in the HEADER alone, and in the PAD alone, is the "local thickness", (see the input forms above where each flaw area is shown). The "local thickness" is smallest for the HEADER, and greatest for the PAD only. An intermediate value was used for the HEADER AND PAD, section (0.6"). The local nominal thickness is the nominal thickness of the plate minus any FCA, or the design corroded thickness.

4-HEADER – This area includes all nodes in the header that are in the 14" radius flaw influence circle. The HEADER AND PAD WELD area includes 3t on either side of the weld, and this area includes nodes outside of that zone, and inside the 14" sphere. There may or may not be any nodes in this zone, if they are all in the HEADER AND PAD WELD zone, because it is so big.

The proximity of the flaw to a weld was not entered. (See the FFS input form below.)

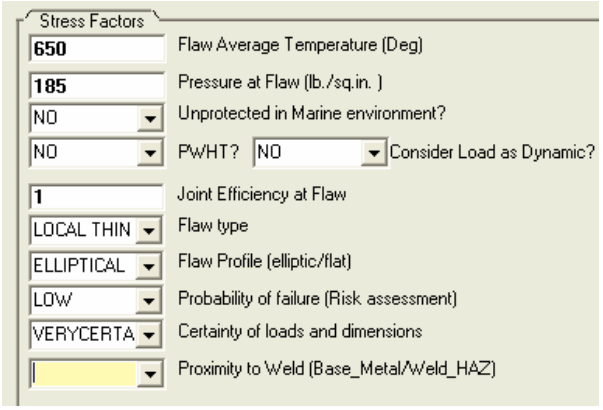


Figure EX3-9

Only the joint efficiency is used to evaluate local thin areas, and so the *"Proximity to Weld"* input is not needed. (The *"Proximity to Weld"* defaults to base metal.)

The FFS results output table of contents is shown below:

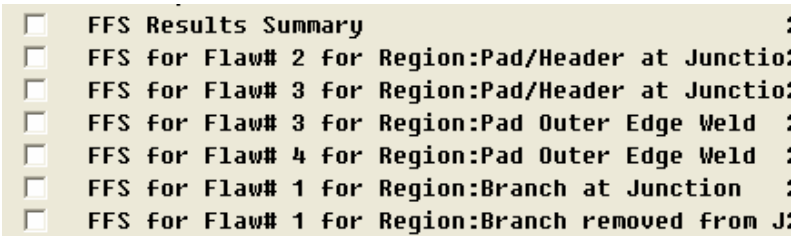


Figure EX3-10

As can be seen by studying the table of contents, nodes in the *"Pad/Header at Junction"* region area fell into the flaw defined by #2, and #3. Nodes in the region described by *"Pad Outer Edge Weld"* fell into the flaw defined by #3 and #4. The worst of the FFS allowable ratios will be reported, regardless of which region the node fell into. If the same node is in multiple flaw zones, it will be evaluated for each zone, and the most susceptible to failure reported.

The FFS summary shows that the thinned area is 8% over the API allowed flaw size at the pad outer edge weld, (as implemented in FE/Pipe).

```
FFS Results Summary


Flaw# 2 Region:Pad/Header at Junction Primary Metal Loss: 0.582
Flaw# 2 Region:Pad/Header at Junction PrimarSecondary Metal Loss: 0.000
Criteria SATISFIED


Flaw# 3 Region:Pad/Header at Junction Primary Metal Loss: 0.665
Flaw# 3 Region:Pad/Header at Junction PrimarSecondary Metal Loss: 0.000
Criteria SATISFIED


Flaw# 3 Region:Pad Outer Edge Weld Primary Metal Loss: 0.954
Flaw# 3 Region:Pad Outer Edge Weld Primary MSecondary Metal Loss: 0.000
Criteria SATISFIED


Flaw# 4 Region:Pad Outer Edge Weld Primary Metal Loss: 1.089
Flaw# 4 Region:Pad Outer Edge Weld Primary MSecondary Metal Loss: 0.000
Criteria NOT Satisfied


Flaw# 1 Region:Branch at Junction Primary Metal Loss: 0.675
Flaw# 1 Region:Branch at Junction Primary MeSecondary Metal Loss: 0.000
Criteria SATISFIED


Flaw# 1 Region:Branch removed from Junction Primary Metal Loss: 0.185
Flaw# 1 Region:Branch removed from Junction Secondary Metal Loss: 0.000
Criteria SATISFIED
```

This is in agreement with the Level 2 assessment performed in the API 579 document in 4.11.3.

For this model, where there is a large area of uniform metal loss (fully circumferential), it would not be unusual to run another calculation where the shell is thinned by 0.1 inch, and the nozzle thickness adjacent to the pad is thinned by 0.35 inches. Since the locally thinned area is in the area of the intersection, it will be reasonable to assume that the thinned area along the nozzle will extend at least 2 $(RT)^{1/2}$ = (2)(2.5) = 5.0 in along the nozzle length. The flaws will be turned off for this run. *The FE/Pipe implementation of API 579, uses local thinning rules and stresses based on the nominal material thicknesses. Adjustments are made to the stresses in the algorithm based on flaw length and depth.*

A thinned model for this reduced intersection is shown below:

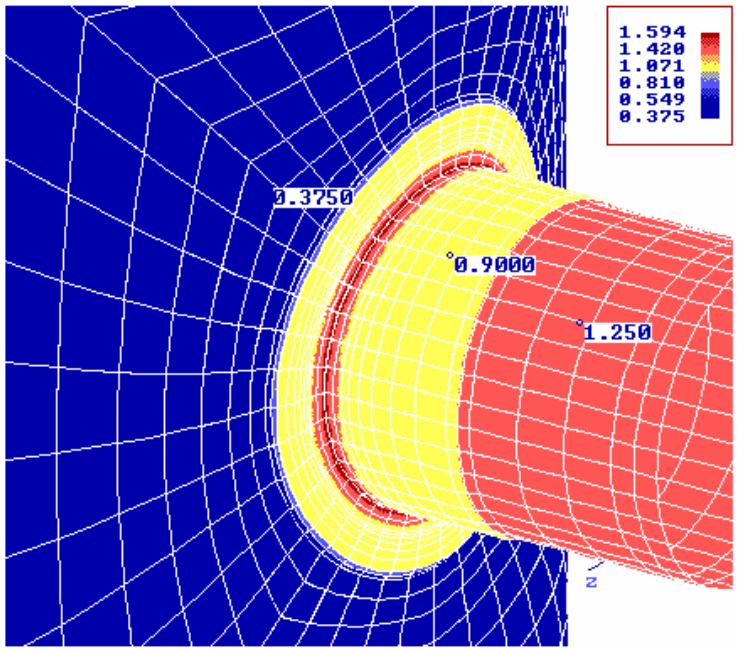
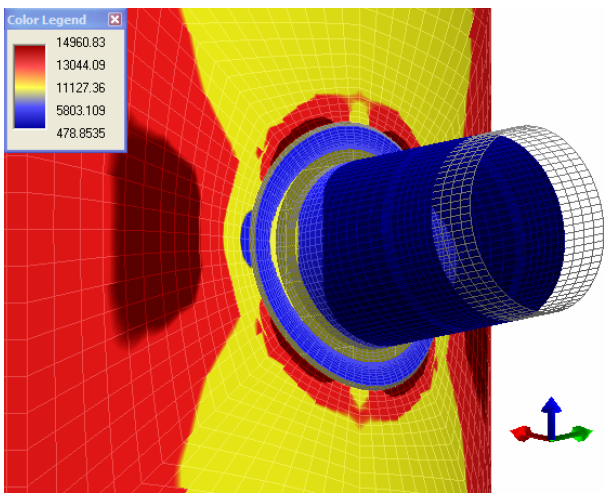


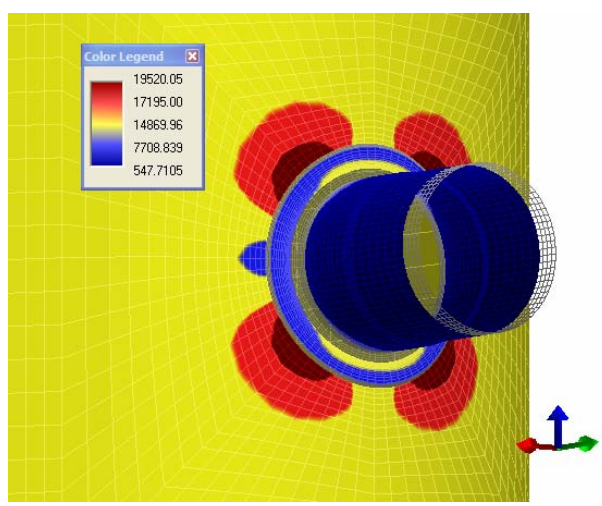
Figure EX3-11

The entire header wall thickness was reduced to the minimum measured value less the FCA, and the nozzle area within $2(RT)^{1/2}$ was reduced to the minimum measured values.

The results from this calculation are shown below. (PRG FFS calculations are not made on the reduced intersection, since the PRG FFS implementation utilizes the nominal stress at the flaw, and adjusts for flaw length and depth.)



Nominal Thickness Stress Calculation



Reduced Thickness Stress Calculation

Figure EX3-12

The stresses have certainly redistributed with the local thinning, but the membrane stresses only increased from 14,960 psi to 19,520 psi.

A portion of the ASME Code compliance report for the thin walled intersection is given below. The highest stress ratios exist for each stress category at the pad outer edge weld. The highest calculated stress is 69% of the 1.5Smh allowable.

```
ASME Overstressed Areas                                12:52:23 29 Jan 2007

             *** NO OVERSTRESSED NODES IN THIS MODEL ***

Highest Primary Stress Ratios                          12:52:23 29 Jan 2007

Pad Outer Edge Weld

       Pl      1.5(k)Smh     Primary Membrane Load Case 1
   19,520         28,200     Sect VIII Ref: AD-140, 4-112(i), 4-133,
      psi            psi                    Fig. 4-130.1, Table 4-120.1
                            Plot Reference:
            69%             1) Pl < 1.5(k)Smh (SUS,Membrane) Case 1


Highest Secondary Stress Ratios                        12:52:23 29 Jan 2007

Pad Outer Edge Weld

  Pl+Pb+Q       3(Smavg)     Primary+Secondary (Inner) Load Case 3
   21,225         63,150     Sect VIII Ref: 4-120(b)(4),4-134,4-136.6,
      psi            psi                    Fig. 4-130.1(Note 1)
                            Plot Reference:
            33%             3) Pl+Pb+Q < 3(Smavg) (OPE,Inside) Case 3


Highest Fatigue Stress Ratios                          12:52:23 29 Jan 2007

Pad Outer Edge Weld

Pl+Pb+Q+F             Sa     Primary+Secondary+Peak (Inner) Load Case 3
   14,327         41,997     Stress Concentration Factor = 1.350
      psi            psi     Strain Concentration Factor = 1.000
                            Cycles Allowed for this Stress = 347,546.
            34%             "B31" Fatigue Stress Allowable = 52625.0
                            Markl Fatigue Stress Allowable = 41701.0
                            WRC 474 Mean Cycles to Failure = 1,853,111.
                            WRC 474 99% Probability Cycles = 430,503.
                            WRC 474 95% Probability Cycles = 597,707.
                            BS5500 Allowed Cycles(Curve F) = 221,098.
                            Membrane-to-Bending Ratio = 3.964
                            Bending-to-PL+PB+Q Ratio = 0.201
                            Sect VIII Ref: 4-112(l)(2),Fig.4-130.1,4-135
                            Plot Reference:
                            5) Pl+Pb+Q+F < Sa (EXP,Inside) Case 3
```

At this time, it would not be unusual for the user to have to decide if an 8% overloaded condition per API 579 is justification for shutting down a process unit for repairs. After additional consideration (see below), the engineer might decide that situations warrant continued operation with the existing wall loss, and might outline an ongoing inspection and maintenance plan for the next major shutdown.

Typically, other considerations that should be considered when determining the suitability of a flawed area can include:

1. Is it safer to keep operating as is, or can the unit be shut down safely and repaired properly? (Repairing can often be more dangerous than operating as is.)

2. Is the rate of corrosion/erosion known, and easy to evaluate on a regular basis. Are process conditions changing that effect the ability to use historical data to predict future wall loss rates?
3. Are minimum thickness readings accurate? Is the inspection zone typically difficult to inspect, i.e. areas with intersection geometries, doubler plates, etc.?
4. What are the consequences involved in a failure? Has the failure mechanism been identified? Is there a possibility of full separation of a load bearing component? Could the pipe suffer a running failure and/or possibly a guillotine separation? Would failure of this line cause critical damage to people or other nearby equipment? What are the consequences of a leak? If there is a leak, can adequate safeguards be put in place to either continue running, or to safely shut down the operation?
5. Were all loads estimated and evaluated accurately? Are there other loads to consider that were not evaluated that would be significant?
6. What is the operating history of the line? Is the information that is available dependable? (The accuracy of information based on operator recollection can vary significantly.)
7. How long must the line operate before a permanent fix can be implemented? Were there ever periods where temperature or pressure excursions could have existed without being recorded?
8. What is the metallurgical condition of the material? Was there access for a visual inspection of the inside? Is there possibility of embrittlement, or other local material damage? Can actual material properties be compared to Code minimums used in design? Is the material suitably ductile, so that local overstrain on a percentage of the wall thickness be tolerated?
9. Has anything changed recently to affect the line, or is there anything scheduled in the near future that will change the operating or loading condition of the line?
10. Has a pipe stress analysis been performed to properly evaluate any external loads? Has the pipe stress model been evaluated for sensitivities that can include, settlement, support stiffness, intersection stiffness and installation accuracy. Large diameter, tight (stiff) piping systems, and systems subject to large thermal differences are particularly susceptible to modeling sensitivities.
11. Erosion, corrosion or cracks in one location often indicate that erosion, corrosion or cracks exist in another location also. Are there other places in the line where a similar metal loss could be more dangerous?

If the answers to these and other questions result in benign responses that are well understood, then the engineer may be justified in continuing operation without repair even though the API 579 analysis showed that the criteria is NOT satisfied. Many plant problems occur because of attempted repairs, welding or maintenance. The inevitable risks associated with field repairs must always be balanced against the real potential for failure.

Not all API analyses require multiple runs using other FE/Pipe models, but the engineer should always verify any particular approach when at all possible. In fact, this is one of the biggest uses for FE/Pipe – to quickly verify a simpler, or more complex calculation.

A limit load analysis could also be run on the nozzle if large loads, in addition to pressure, are applied to the nozzle. This type of analysis should only be used when there is adequate justification to leave a highly loaded nozzle in service, and when more simplified approaches do not show the nozzle to be acceptable.

Problem 9.11.1 Example Problem 1 API 579 – Using FE/Pipe

Design Conditions = 300 psi @ 650 F
Inside Diameter = 96 inches
Outside Diameter = 98.5 inches
Fabricated Thickness = 1.25 inches
Uniform Metal Loss = 0.10 inches
FCA = 0.125 "
Material = SA 516 Grade 70
Weld Joint Efficiency = 0.85
Original Equipment underwent PWHT.

R = (98.5 – 1.025) / 2 = 48.7 inches

Inspection Data (From API 579):

The flaw is located in the HAZ of a longitudinal weld seam on the inside of a cylindrical vessel. The flaw is parallel to the weld join. The depth of the flaw is 0.25 inches (readings vary). The flaw length is 1.1 inches. The flaw is 60 inches removed from the closest structural discontinuity.

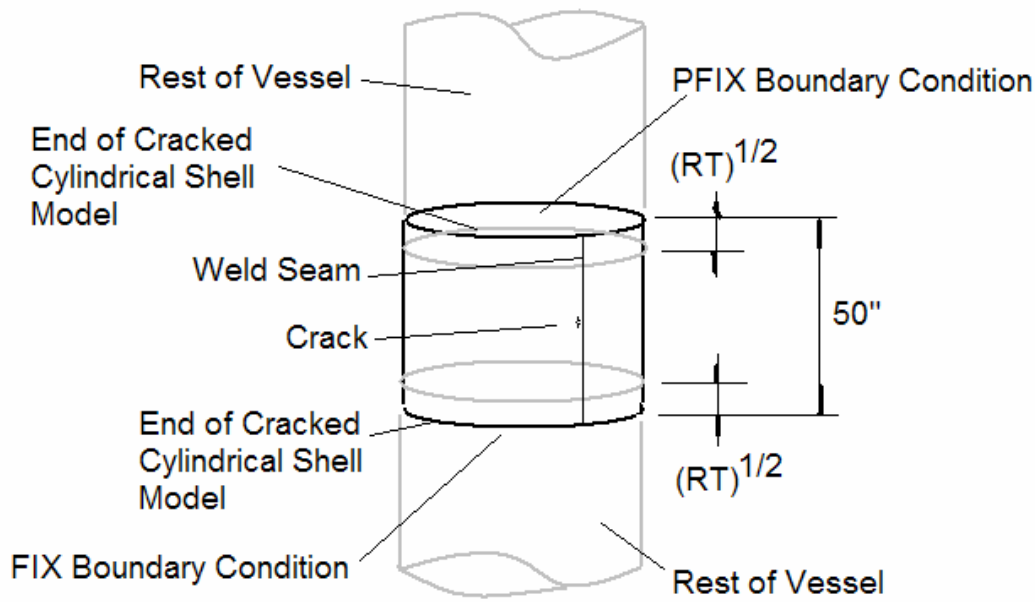


Figure EX4-1

The FE/Pipe assumption that the flaw has the worst possible propagation orientation relative to the maximum stress state is accurate for the longitudinal flaw in a cylindrical vessel.

The cylinder wall thickness will be: 1.25 – 0.125 – 0.1 = 1.025 inches.

A "close" proximity discontinuity is one that is within about $1.5(RT)^{1/2} = (1.5)[(48.7)(1.025)]^{1/2}$ = 10.6 inches. To make sure that the crack is not affected by either end's boundary condition on the shell, the total shell length will be 50 inches.

The FIX boundary condition on the bottom of the shell section restrains the bottom of the vessel in the axial and torsional translational directions, and in the circumferential rotational direction.

The PFIX boundary condition on the top of the vessel is a “Pressure” Fixity, and restrains the top of the vessel in the torsional translational direction and in the circumferential rotational direction. In any load case with pressure, a PD/4t axial stress is applied to the top of the vessel.

To decide where to locate the crack, the origin must be identified:

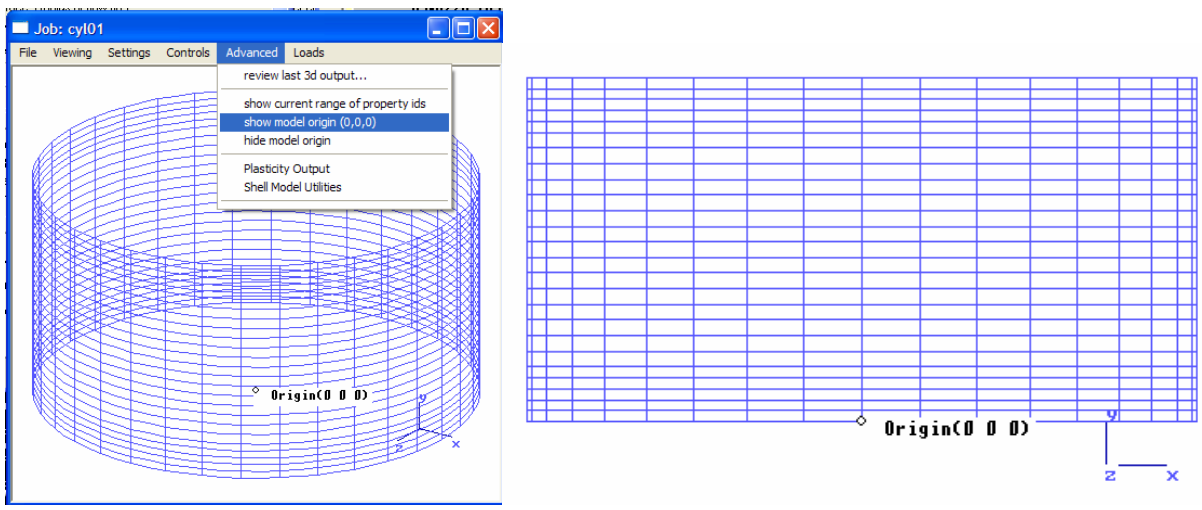


Figure EX4-2

The origin is on the bottom of the entered cylindrical shell, (Nozzles-Plates-Shells). To locate the crack on the outer surface, at the half-height of the model, as shown in the sketch above, the Y coordinate for the flaw should be 25 inches, and the X coordinate 48.7 inches. These coordinates will put the center of the flaw sphere of influence on the surface, at the middle of the vessel along the X axis. The total crack length is 1.1 inch. So the theoretical flaw radius should be 1.1/2 = 0.55 inches.

A plot of the sphere of influence and the nodes inside it are shown below.

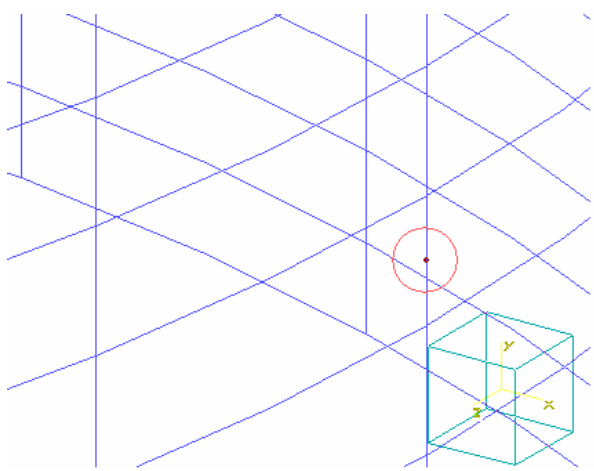


Figure EX4-3

The flaw sphere of influence is so small that only a single node is included in it. This is OK, since only a single stress value is required to find maximum membrane and bending stresses for use in the API evaluation. Since the cylinder is not exposed to discontinuity stresses, each node will have essentially the same stress value, and so a larger or smaller sphere in this instance would make no difference.

The input for this crack is shown below.

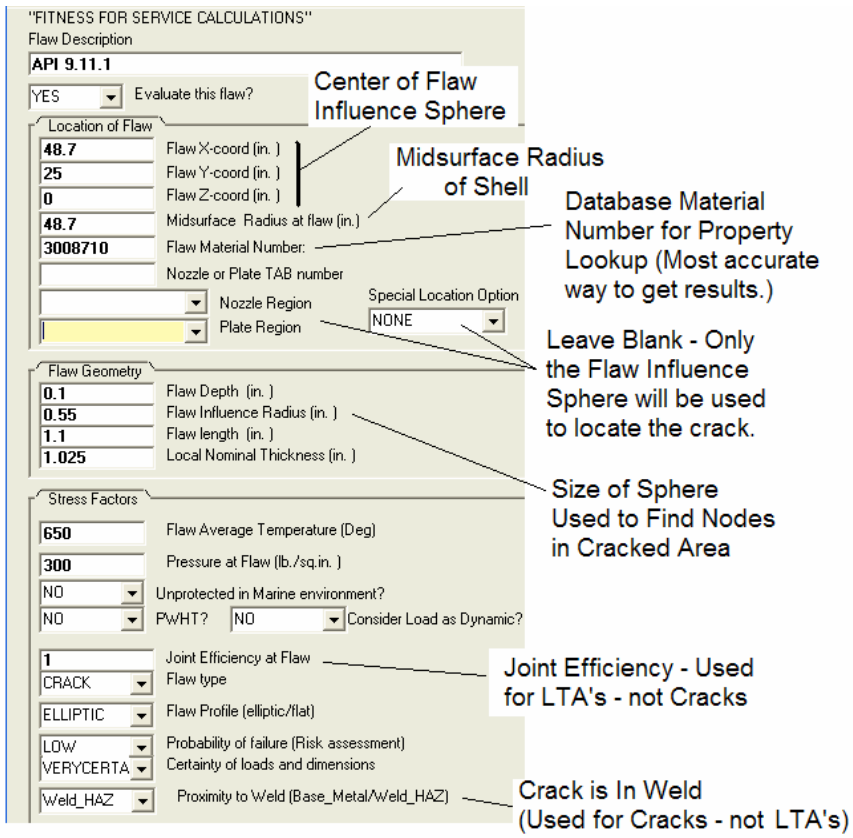


Figure EX4-4

Output for this crack evaluation is given below, and shows that the flaw is acceptable as echoed in API 579 5.11.1. Key items for evaluation are highlighted.

```
FFS Results Summary

Flaw# 1 Region:Cylindrical Shell Crack FAD Load Ratio: 0.304
Criteria SATISFIED


FFS for Flaw# 1 for Region:Cylindrical Shell

API 579 Fitness for Service Evaluation
-----------------------------------------------

Conservative assumptions were made when implementing the
fittness for service rules of API579 Sections 5.0 and 9.0.
It is the users responsibility to review and check the
results printed herein to verify that they apply and are
valid for the particular problem studied.

Descr:  API 9.11.1
```

```
Yield Stress at Room Temperature                =      38.000 ksi
Yield Stress at Operating Temperature           =      28.200 ksi
Flow Stress at Operating Temperature            =      38.200 ksi
Flow Stress at ROOM Temperature                 =      48.000 ksi
Modulus of Elasticity at Room Temperature       =   29400.000 ksi
Modulus of Elasticity at Operating Temperature  =   25999.998 ksi

Internal Pressure                               =     300.000 psi
Operating Temperature                           =     650.000 degF

Local Primary Membrane Stress in Area of Flaw   =      14.336 ksi
Local Primary Bending Stress in Area of Flaw    =       0.141 ksi
Local Secondary Membrane Stress in Area of Flaw =       0.000 ksi
Local Secondary Bending  Stress in Area of Flaw =       0.000 ksi

Initial Crack Depth                             =       0.100 in.
Initial Crack Half-Length                       =       0.550 in.
Component Wall Thickness at Flaw                =       1.025 in.
Component Inside Radius at Flaw Location        =      48.700 in.

Flaw is in base metal removed from welds.

Probability of Failure                          = 0.023000000

Coefficient of Variation                        =       0.100
(Primary Loads and Stresses have
 signifiant uncertainty due to random
 loading or modeling approximations.

Poissons Ratio used in this analysis            =       0.300


API 579 Section 9.0 Assessment of Crack-Like Flaws
--------------------------------------------------

Partial Safety Factor for Stress (Table 9.2)    =       1.750
Partial Safety Factor for KI (Table 9.2)        =       1.000
Partial Safety Factor for Crack Length          =       1.000
Folias Factor for Bulging                       =       1.005

Primary Load Ratio for FAD (LrP)                =       0.894
Secondary Load Ratio for FAD (LrSR)             =       0.000
Maximum Allowed FAD Load Ratio (Lr)             =       1.355

FFS for Flaw# 1 for Region:Cylindrical Shell

Primary Load Stress Intensity at Flaw (KIp)     =      15.450 ksi.sqr(in.)
Secondary Load Stress Intensity at Flaw (KIsr)  =       0.000 ksi.sqr(in.)
Factored Material Fracture Toughness (KIC)      =      71.352 ksi.sqr(in.)
Non-Factored Material Fracture Toughness (KIC)  =      71.352 ksi.sqr(in.)

Secondary Load Plasticity Interaction Factor    =       1.000
Primary+Secondary Toughness Ratio (Kr) for FAD  =       0.217
Allowed Fad Toughness Ratio (Kr[Allowed])       =       0.713

Per API 579 9.4.4.1 (a) FAD Acceptance Ratio    =       0.304
   (Component IS acceptable.)


Fatigue Analysis - From Input Flaw Size to Failure
--------------------------------------------------

The flaw fatigue analysis is used to determine the
number of allowed cycles to increase the initial flaw
to critical proportions. Partial Safety Factors per
9.4.3.2 (e) are NOT used.

Cycles to Grow Flaw to Critical Size            = 0.16433E+06

Final, Critical Flaw Depth (a)                  =       0.886 in.
Final, Critical Flow Total Length (2c)          =       4.245 in.


Initial Flaw Size Calculator
----------------------------


The initial flaw size calculator load cycles back through
the flaw growth life starting from a critical flaw size
and cycling progressively back to smaller and smaller
cracks until the total cycles have been exhausted.
```

```
The final flaw size is theoretically the largest flaw that
can propogate to a critical size in the given number of
cycles.

Total cycles counted back from max flaw size     =    7000.000

Smallest depth (a) to grow to failure            =       0.671 in.
Initial Flaw depth                               =       0.100 in.
Final Flow depth                                 =       0.886 in.


Leak Before Break (LBB) Analysis from Critical Flaw
---------------------------------------------------

The critical flaw length used (2c)               =       4.245 in.

The critical flaw size will leak without producing
an unrestrained, running crack.

The water flowrate through the final crack is    =       7.642 cu.in/sec
Leak-to-Break Ratio                                  =        0.654
```

The report shows that the specified crack is acceptable as is and does not require repair. A significant number of cycles are allowed (164,000+) before a critical size is reached, and the crack would likely leak without producing a running crack.

Problem 9.11.2 Example Problem 2 API 579 – Using FE/Pipe

A crack-like flaw in the long seam of a vessel. Is the crack acceptable for operation at 30F?

Design Conditions = 200 psi @ 750F
Inside Diameter = 120 in.
Outside Diameter = 122 in.
Fabricated Thickness = 1.0 in.
Corrosion Allowance = 0.0 in.
FCA = 0.0 in. (Future Corrosion Allowance)
Material = SA 516 70.
Weld Joint Efficiency = 0.85
PWHT = No.

Use a reference temperature of 30F, and a Probability of Failure of 10^{-3}, representing a shift of about 3 standard deviations from the mean of the failure line.

The flaw depth is 0.20 inches and its length is 3.2 inches. The distance between the flaw and the nearest structural discontinuity is 30 inches.

The actual operating temperature for the vessel is 30F, and can be evaluated for this temperature.

R = (122 – 1)/2 = 60.5 inches.
$(RT)^{1/2} = [(60.5)(1)]^{1/2} = 7.8$ inches.

The flaw is far enough removed from the closest discontinuity so that local stresses from the discontinuity are not expected. As for the previous example, a length of 50 inches will be used for this model. This will give (50-3.2)/(7.8) / 2 = 3, (three) times $(RT)^{1/2}$ on either side of the flaw before the boundary condition. It is also assumed that ovalization effects, which do not observe the $(RT)^{1/2}$ limits, are not present in the vessel.

Input for the flaw is shown below:

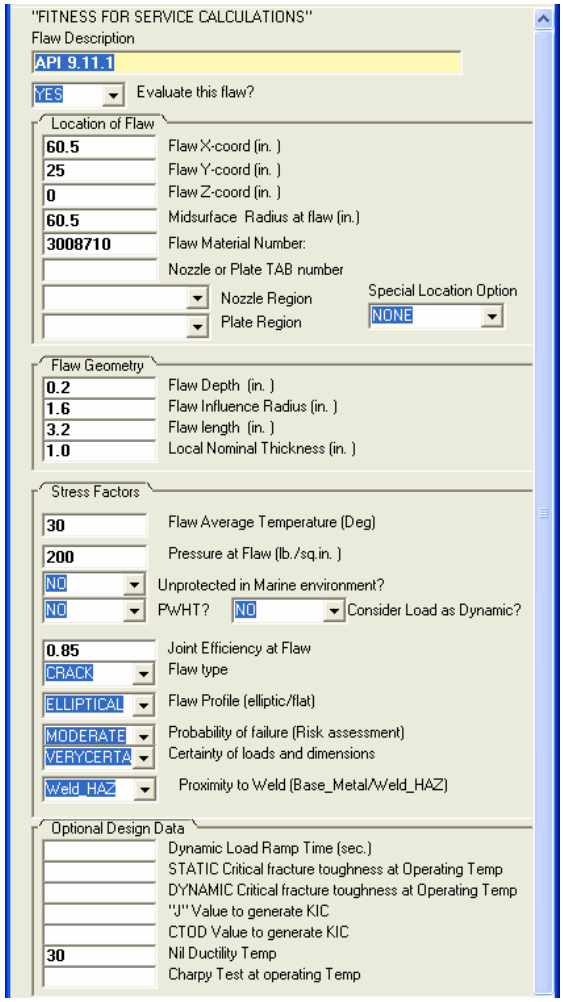


Figure EX5-1

The defined flaw sphere of influence and included model nodes are shown below.

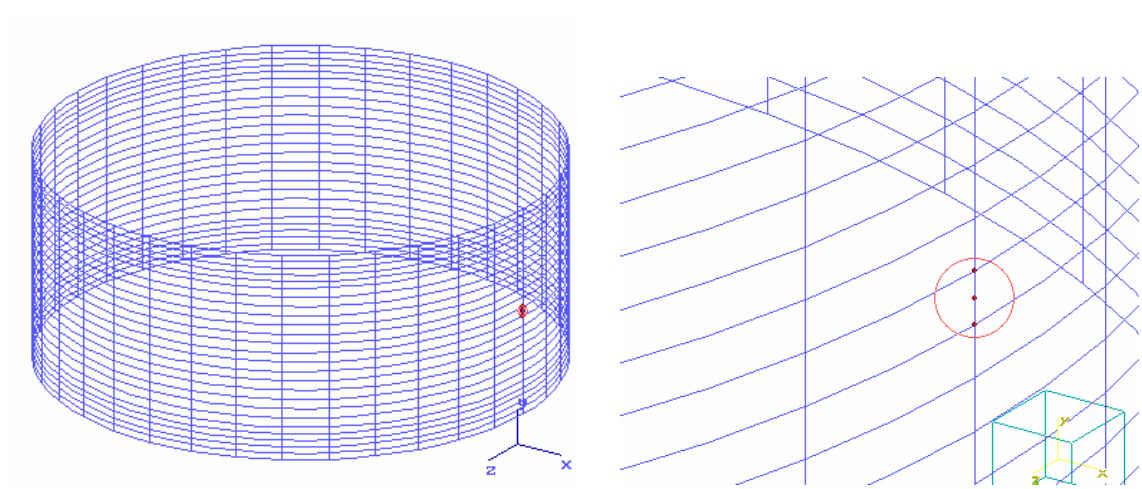


Figure EX5-2

There are three nodes included in the flaw zone. For FE/Pipe or NozzlePRO it is only important to include the node with the highest stress in the area where the flaw exists. Since this problem involves only a pressurized vessel, each node removed from boundary by more than $(RT)^{1/2}$ will have the same stress state and so only a single node is required. FE/Pipe uses the node in the sphere of influence with the highest stresses for the evaluation.

The FE/Pipe results summary for the 30degF condition requested is shown below, and seen to be satisfactory in agreement with the API evaluation. Critical lines are shown highlighted.

```
FFS Results Summary

Flaw# 1 Region:Cylindrical Shell Crack FAD Load Ratio: 0.830
Criteria SATISFIED

FFS for Flaw# 1 for Region:Cylindrical Shell

API 579 Fitness for Service Evaluation
-------------------------------------------------

Conservative assumptions were made when implementing the
fittness for service rules of API579 Sections 5.0 and 9.0.
It is the users responsibility to review and check the
results printed herein to verify that they apply and are
valid for the particular problem studied.

Descr:  API 9.11.2

Yield Stress at Room Temperature                 =      38.000 ksi
Flow Stress at Operating Temperature             =      48.000 ksi
Modulus of Elasticity at Room Temperature        =   29400.000 ksi
Modulus of Elasticity at Operating Temperature   =   29611.764 ksi

Internal Pressure                                =     200.000 psi
Operating Temperature                            =      30.000 degF

Local Primary Membrane Stress in Area of Flaw    =      12.161 ksi
Local Primary Bending Stress in Area of Flaw     =       0.094 ksi
Local Secondary Membrane Stress in Area of Flaw  =       0.000 ksi
Local Secondary Bending  Stress in Area of Flaw  =       0.000 ksi

Initial Crack Depth                              =       0.200 in.
Initial Crack Half-Length                        =       1.600 in.
Component Wall Thickness at Flaw                 =       1.000 in.
Component Inside Radius at Flaw Location         =      60.500 in.

Charpy Nil Ductility Temperature (RNDT)          =      30.000 deg F

Flaw is in an area that contains a weld or HAZ.

Longitudinal Weld Joint Efficiency               =       0.850
Probability of Failure                           = 0.001000000

Coefficient of Variation                         =       0.100
(Primary Loads and Stresses are
 computed and well known.)

Poissons Ratio used in this analysis             =       0.300


API 579 Section 9.0 Assessment of Crack-Like Flaws
-------------------------------------------------

Partial Safety Factor for Stress (Table 9.2)     =       1.500
Partial Safety Factor for KI (Table 9.2)         =       1.000
Partial Safety Factor for Crack Length           =       1.000
Folias Factor for Bulging                        =       1.028
```



```
Primary Load Ratio for FAD (LrP)                  =       0.494
Secondary Load Ratio for FAD (LrSR)               =       1.299
Maximum Allowed FAD Load Ratio (Lr)               =       1.263

FFS for Flaw# 1 for Region:Cylindrical Shell

Primary Load Stress Intensity at Flaw (KIp)       =      17.630 ksi.sqr(in.)
Secondary Load Stress Intensity at Flaw (KIsr)    =      46.118 ksi.sqr(in.)
Factored Material Fracture Toughness (KIC)        =      87.838 ksi.sqr(in.)
Non-Factored Material Fracture Toughness (KIC)    =      87.838 ksi.sqr(in.)

Secondary Load Plasticity Interaction Factor      =       1.135
Primary+Secondary Toughness Ratio (Kr) for FAD    =       0.797
Allowed Fad Toughness Ratio (Kr[Allowed])         =       0.960

Per API 579 9.4.4.1 (a) FAD Acceptance Ratio      =       0.830
    (Component IS acceptable.)
```

The assessment point for this flaw along with the developed Fracture Analysis Diagram (FAD) is shown below:

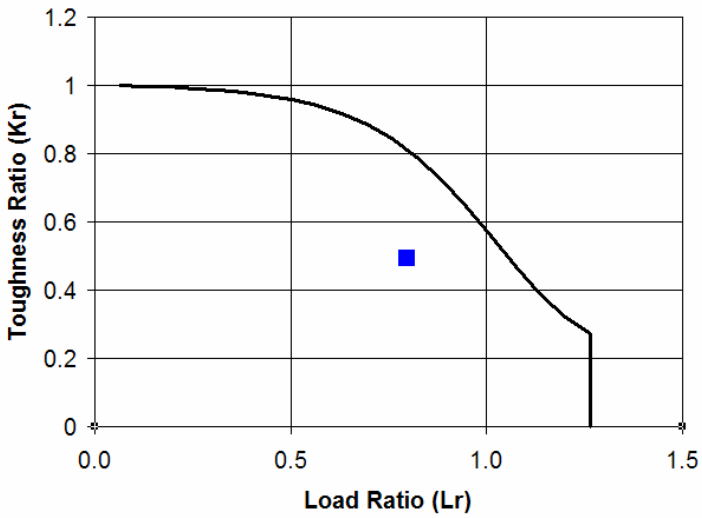


Figure EX5-3

The load ratio (Lr) can be thought of as the ductile failure ratio, and the toughness ratio can be thought of as the brittle failure ratio.

The assessment point is Lr,Kr (shown as a blue square for the example), that must be inside the FAD range shown.

Lr = Actual Load / Ductile Failure Load
Kr = Stress Intensity (K_I) / Allowed Stress Intensity (K_{IC})

Graphical output for the flaw evaluation is also provided. Nodes included in the flaw evaluation are drawn inside the header flaw red sphere of influence shown in the figure below on the right. (Plotted from the input.) The API 579 FAD Unity check for the flaw area nodes is shown in the middle view. The flaw plot reference is shown as plot #12 in the menu on the left. Any value shown in the plot greater than 1 occurs at a location where the defined flaw is larger than the flaw size permitted by the API 579 evaluation.

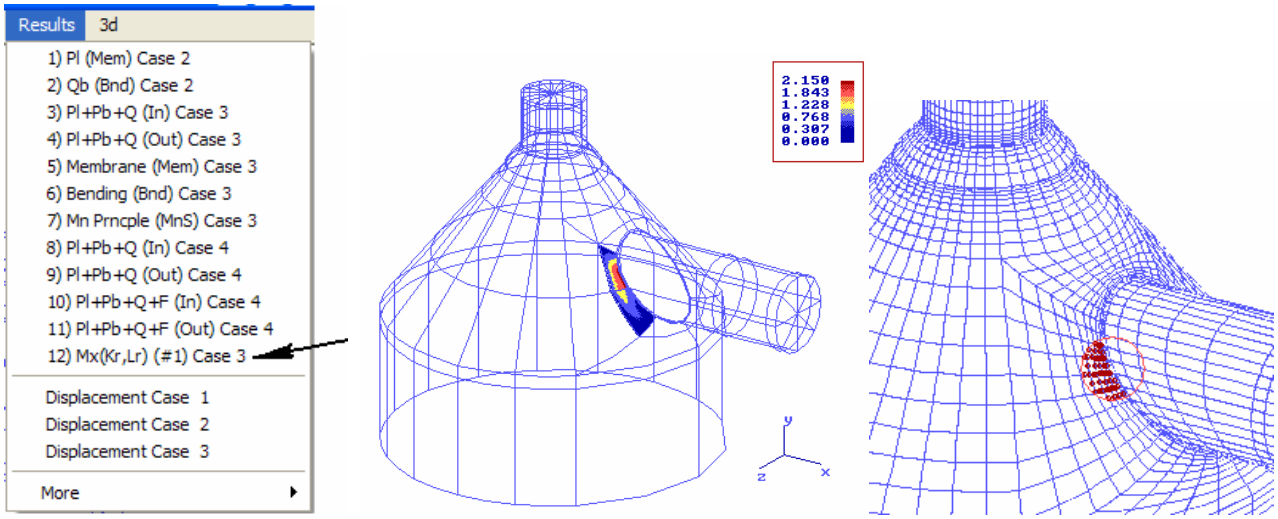

Results
3d
1) Pl (Mem) Case 2
2) Qb (Bnd) Case 2
3) Pl+Pb+Q (In) Case 3
4) Pl+Pb+Q (Out) Case 3
5) Membrane (Mem) Case 3
6) Bending (Bnd) Case 3
7) Mn Prncple (MnS) Case 3
8) Pl+Pb+Q (In) Case 4
9) Pl+Pb+Q (Out) Case 4
10) Pl+Pb+Q+F (In) Case 4
11) Pl+Pb+Q+F (Out) Case 4
12) Mx(Kr,Lr) (#1) Case 3
Displacement Case 1
Displacement Case 2
Displacement Case 3
More
2.150
1.843
1.228
0.768
0.307
0.000
y
z
x

Figure EX5-4

Section 3: ASME NH High Temperature Analysis

General Discussion

The rules of ASME VIII Division 2 are limited to temperatures below the creep range. In North America there is presently only one published code that addresses design in the creep range: ASME Section III Part NH. FE/Pipe offers automated conservative NH elastic analysis compliance reports in selected templates. The same processor is available in Mat/PRO for other templates and/or local analyses.

PRG's implementation of NH is generally conservative within its simplified approach. If the requirements of elastic analysis are not met, the designer has the option to use a more sophisticated approach. In other words this automated method can be used as pre-screening tools for more detailed evaluation.

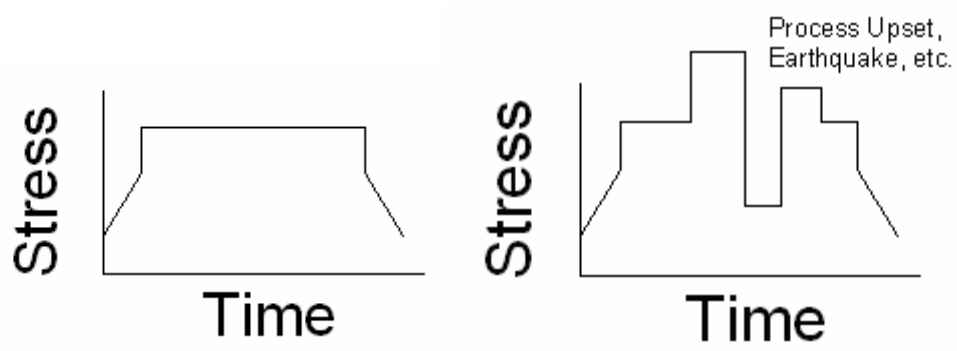
Assumptions inherent in PRG's automation of NH are listed below. For more details, contact support.

(a) The rules in Appendix T assume that the pressure design rules have been satisfied separately for the time dependant strength. Guidelines and approximations to the load controlled stress limits are provided.

(b) All cycles must be identical, or there must be a single predominant cycle.

(c) Only simple operating cycles are considered in the automated results.

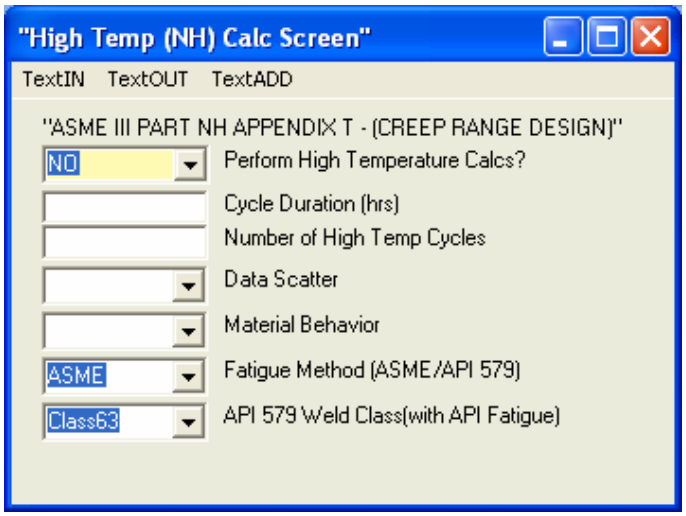
For example:



Simple Single Cycle Complex Single Cycle

FE/Pipe Program Input

The following standard input screen activates the NH elastic analysis rules, and includes the necessary additional inputs.



Perform High Temperature Calcs

This option is used to activate the creep and creep/fatigue interaction rules found in ASME Section III Part NH Appendix T. Categorization of stresses is conducted automatically, so input loads and boundary conditions the same way for a standard FE/Pipe analyses. The default entry is NO.

Cycle Duration (hrs)

If the process is cyclic, specify the duration of one operational cycle. If the system does not cycle, input the total life (design or actual). Variations in temperature throughout the cycle are not considered.

Number of High Temperature Cycles

Specifying a value greater than one will invoke the creep/fatigue interaction rules in ASME Section III part NH paragraph T-1411. The assumption is that all cycles are the same.

Data Scatter (Low/Median/High)

Sets the reference scatter band of the high temperature properties. Present data only includes "low" and "median". These are comparable to "minimum" and "average" rupture stress. "Median" represents the mean line fitted to test data. As such, "Median" often gives a fairly good prediction of failure with a detailed stress calculation. Use "Median" with caution. "Low" is specified for design in the rules of ASME Section III Part NH.

Material Behavior (Brittle/Ductile)

Use "Brittle" for cases where the metal has been rendered brittle by temperature and/or the process, such as carbide precipitation, sigma phase formation, and so on. (See ASME Section II part D Appendix6 for some discussion.).

Welds and HAZ are often considered "brittle", but it can be highly conservative if not pessimistic to subject the entire model to "brittle" requirements just for the sake of welded joints. A more realistic option is to use "ductile" for the overall model and use the NH calculator in Mat/PRO for local evaluations of welds.

Fatigue Method (ASME/API 579)

Specifies which fatigue curve is used in the T-1411 fatigue damage term when the "number of high temperature cycles" is greater than one. The ASME material curve and the API 579 property adjustments are performed automatically based on the materials inputs elsewhere in the template.

ASME	ASME Section III / ASME Section VIII Division 2 Appendix 5 fatigue curves
API 579	“As-welded” fatigue curves in API 579

API 579 Weld Class

This entry is only used when "number of high temperature cycles" is greater than one, and the "API579" is selected as "fatigue method". Each weld class has several applications defined in API 579. The simplest/most common are listed below. (See API 579 Appendix F for more detail).

Class 124	Unwelded base metal
Class 100	Inside corner of nozzle and flush ground full penetration butt welds
Class 80	Single sided butt weld with or without backing strip (pipe girth welds)
Class 63	Nozzle welds at toe of fillet
Class 50	Toe of fillet welded supports
Class 40	Fillet weld throat failure

Reporting

ASME III NH Evaluates Load Controlled (Primary) and Displacement Controlled (Secondary/Fatigue) Type loadings. The PRG implementation of NH utilizes an internal material database compiled from different sources to extent the NH approach, and so does not have exactly the Load Controlled allowables referenced in NH. For example, the PRG implementation produces the rupture stress at time and temperature for most materials in the full PRG ASME and B31.3 data base, but does not store:

1) The average stress to obtain a total strain of 1% (although this can be calculated), or
2) 80% of the minimum stress to cause tertiary creep,

and these values are used to determine the NH allowable stress St. As a result, St, (which is used for load controlled limits), is estimated using only 67% of the rupture stress at time and temperature, which is only one of the ways to develop the St limit.

The load controlled checks for percentages of yield and tensile can also be a little less conservatively estimated, than by the comparison to the Sm value alone. The PRG implementation uses the conservative approach and uses the Sm value alone, for load controlled checks using Sm.

Displacement controlled checks are independent of load controlled checks, and are comprised of a strain check in T-1300, and a fatigue, and a creep/fatigue interaction check in T-1400. The T-1300 check is a three level check. If level 1 is not satisfied, then level 2 can be checked. If level 2 is not satisified, then level 3 can be checked. If any one of the checks are satisfied then the Code rules are passed. In the calculation result shown below the T-1320 A1 check is passed, and so no further strain checks are required.

The T-1400 checks require the computation of creep strains at each cycle, and comparison to an adjusted strain-vs-cycle fatigue curve. These curves are produced for all materials internally in the NH processor. Results should be studied carefully for soundness and to confirm that they are reasonable.

High entered stresses may result in data points that are far along an asymptotic stress-strain curve, and can produce very small allowables. The user should note that high temperature properties drop off very rapidly as a function of temperature, and should plot Plasochronous curves, yield curves, tensile curves, and allowable curves as a function of temperature to determine the proximity to critical slope changes.

Two types of output reports are available.

```
☐  ASME III NH Summary
☐  ASME III NH Assessment for Region:Pad/Header at
```

ASME III NH Summary

A pass/fail summary for all of the locations defined in the model (same locations used in the standard ASME VIII code compliance reports). Sample report is shown below.

```
ASME III NH Assessment for Region:


NH Region:Pad/Header at Junction Criteria SATISFIED : 0.007
NH Region:Pad Outer Edge Weld Criteria SATISFIED : 0.007
NH Region:Header Outside Pad Area Criteria SATISFIED : 0.004
NH Region:Branch at Junction Criteria SATISFIED : 0.005
NH Region:Branch removed from Junction Criteria SATISFIED : 0.004
```

Detailed report for specified region. Key results are highlighted.

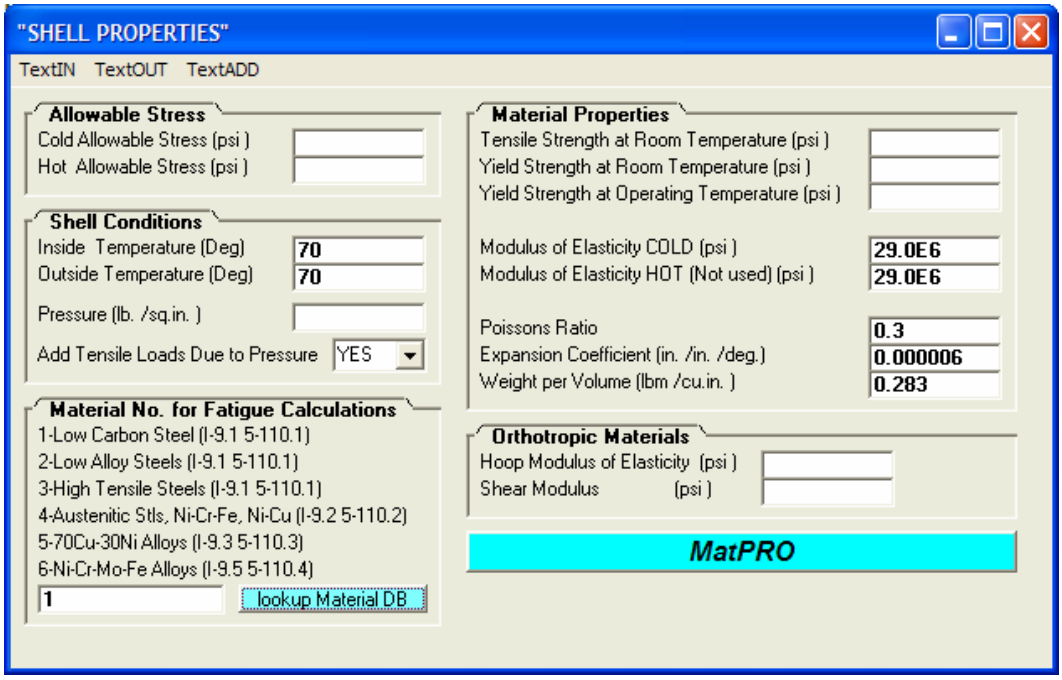

```
MatPRO 1.0 - PAULIN RESEARCH GROUP     HOUSTON, TX
==============================================================================================
Time Stamp : 2/6/2007  12:34:04 PM

Materials Database : "ASME II-D, Table 1A" (2006)


ASME III NH T-1300 Deformation and Strain Limits
ASME III NH T-1400 Creep-Fatigue Evaluation
-----------------------------------------------

The following implementation is a simplified approximation
of the method used in ASME III NH. Users should be sure that
the simplifications used are sufficiently representative of
the actual problem loadings and stresses.

Stainless Nozzle

Material = SA-213TP304H 18Cr-8Ni Smls. tube

Hours (hold time) at Elevated Temperature        =   2000.000
Elevated Temperature at most of Cycle            =    1050.000 degF      565.556 degC

Operating Elastic Stress Intensity Range         =      21.222 ksi      146.321 MPa
Constant Local Primary Membrane Stress (Pl)      =      14.222 ksi       98.057 MPa
Operating Inelastic Strain per Cycle (in/in)     =  0.0000000E+00

Location of Stress/Strain Evaluation             =   BASE METAL

Use Median of Creep Material Scatter Band
Brittle Measure (+0.3-Brittle  -0.3-Ductile)     =  -0.3000000

Number of Creep-Fatigue Interaction Cycles       =   40.00000
Fatigue Curve for Room Temperature Properties    = ASME Div 2
Stress Concentration Factor (SCF)                =   1.350000

Room Temperature Elastic Modulus                 =   27800.000 ksi   191674.250 MPa
Elevated Temperature Elastic Modulus             =   21950.000 ksi   151339.922 MPa
Room Temperature Yield Stress                    =      30.000 ksi      206.843 MPa
Elevated Temperature Yield Stress                =      15.100 ksi      104.111 MPa
Room Temperature Tensile Stress                  =      75.000 ksi      517.107 MPa
Elevated Temperature Tensile Stress              =      55.156 ksi      380.284 MPa

Average Modulus during Strain Cycling            =   24875.000 ksi   171507.094 MPa
Room Temperature Allowable Stress                =      20.000 ksi      137.895 MPa
Elevated Temperature Allowable Stress            =      10.100 ksi       69.637 MPa

Creep Rupture Properties at Time and Temperature:
Minimum Creep Rupture Strength                   =      28.740 ksi      198.155 MPa
Larson-Miller Parameter                          =   27.63456

General Computed Values and Load Controlled Stresses

Pl is less than pseudo-St
Pl (Local Primary Membrane Stress)               =      14.222 ksi       98.057 MPa
St (67% Rupture Stress at Time and Temp          =      19.256 ksi      132.764 MPa

Pl is GREATER than Sm - NOT Ok.
Pl (Local Primary Membrane Stress)               =      14.222 ksi       98.057 MPa
Sm (ASME Code Allowable Stress at Temp)          =      10.100 ksi       69.637 MPa

Stress to Produce 5.0% Strain at Time and Temp   =      27.195 ksi      187.503 MPa
Stress to Produce 2.0% Strain at Time and Temp   =      23.695 ksi      163.371 MPa
Stress to Produce 1.0% Strain at Time and Temp   =      21.195 ksi      146.134 MPa
Stress to Produce 0.5% Strain at Time and Temp   =      18.195 ksi      125.450 MPa

Strain due to Pl at Time and Temp                =  0.2430268     %
Strain due to Pl+Pb/1.25 at Time and Time        =  0.7583014     %

T-1320 Tests A1 and A2

T-1322 or T-1323 Have been satisfied.
Pl+Pb+Q(max) is less than Sy(avg)
Pl+Pb+Q(max)       =       21.222 ksi      146.321 MPa
Sy(avg)            =       22.550 ksi      155.477 MPa

No further strain limit checks are required.
```

```
T-1400 Creep-Fatigue Evaluation

T-1400 Peak Stress Comparison to High Temperature
Adjusted Allowable Cyclic Stress vs Cycle Curve

Calculated Cyclic Stress-Range is LESS THAN the
allowed fatigue stress range - OK.

Calculated Alternating Peak Stress =       14.325 ksi       98.766 MPa
Allowable  Alternating Peak Stress =      300.805 ksi     2073.974 MPa
Stress Based Percent of Allowable  =    4.762179     %

T-1432 Cyclic Strain Range Determination

Calculated Cyclic Strain-Range is LESS THAN the
Strain-Range allowed including creep-fatigue
interaction effects - OK.

Calculated Cyclic Strain Range    =   4.4109044E-03
Allowable  Cyclic Strain Range    =   2.4185289E-02
No. of Cycles Evaluated           =    40.00000
Cyclic Stress Range Allowed       =      601.609 ksi     4147.949 MPa

Creep Strain per Cycle            =   2.8085986E-03
Fatigue Strain per Cycle          =   1.6023058E-03
Multiaxial Plasticity Adjustment  =    1.000000
Plastic Poisson Ratio Adjustment  =    1.000000

Design Cycles are LESS THAN Allowed Cycles - OK

Percent Fatigue Life Used This Load Case =   4.0000000E-08 %
Conservative Fatigue Life Used This Case =    3.997611     %
Design Number of Cycles                  =    40.00000
Allowed Number of Cycles                 =   9.9999998E+10

T-1433 Creep Damage Evaluation

Summed Creep Life Fraction =   1.5375543E-03
Isochronous Integration Pt =            3

Allowed Creep Life from Membrane Stress (Pl,hrs)  =    1300702.
Creep Design Life (hrs)                           =    2000.000

Creep Life Fraction LESS THAN 1.0 - OK

T-1400 Eq.10 Using Fig T-1420-2

Conservative Creep-Fatigue Life Fraction LESS  THAN Allowable - OK
Conservative Creep-Fatigue Life Fraction =   4.1513663E-02
```

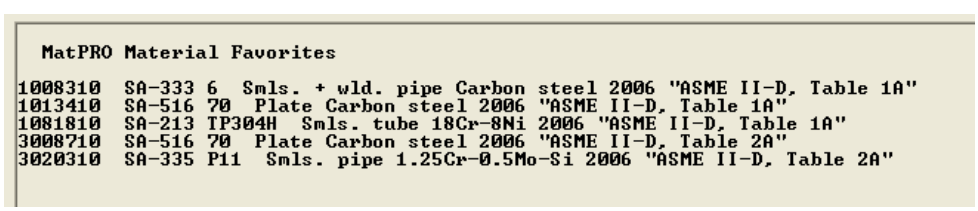
Section 4: FE/Pipe and Nozzle/PRO Link to Latest ASME and B31.3 Material Database

General Discussion

The FE/Pipe user can use the material ID number from MatPRO and have the material allowable stresses interpolated based on material and taken directly from the material database. See the screens below:



In the Material No. for Fatigue Calculations the user may enter either 1-thru-6 as before, or he may enter the MatPRO material id. The lookup Material DB material button can be pressed to get a list of all MatPRO material ID's for the user's favorite materials, an example of which is shown below:



The seven digit number appearing on the left is the MatPRO material ID. Note that the ID is Code year dependant. As code years change, the ID's will change, although the database will always support older code years and ID's.

The available data bases in this release include:

ASME Section II Part D years 2000/2001/2004/2005 and 2006, and the B31.3 2004 databse.

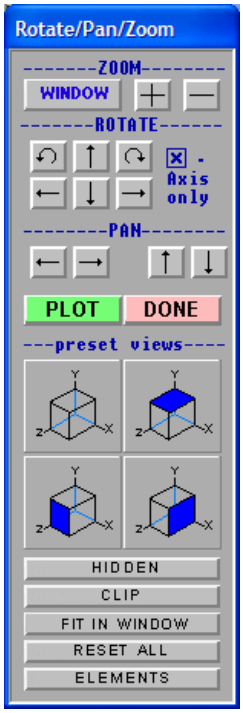
This feature is also available in NozzlePRO.

Section 5: FE/Pipe Version 4.5

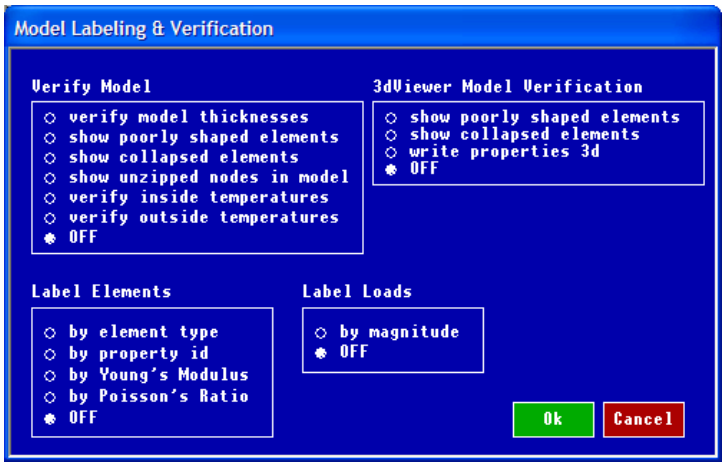
Model Generator User Interface Update

A variety of additional options has been added to the model generation interface – modgen:

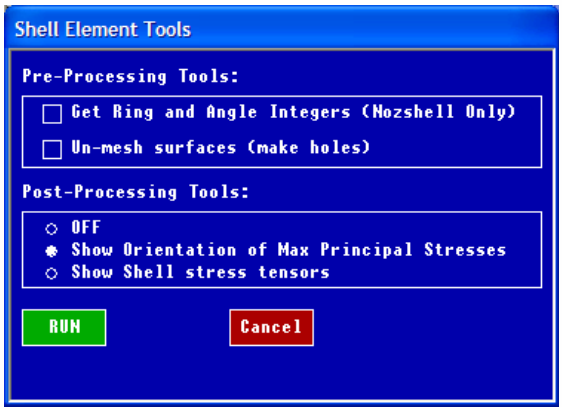
Rotate/Pan/Zoom – Panel Change:

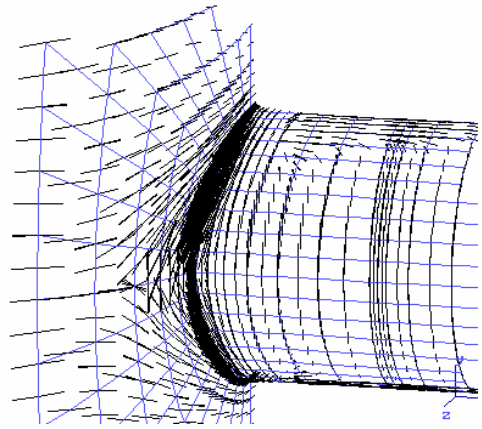


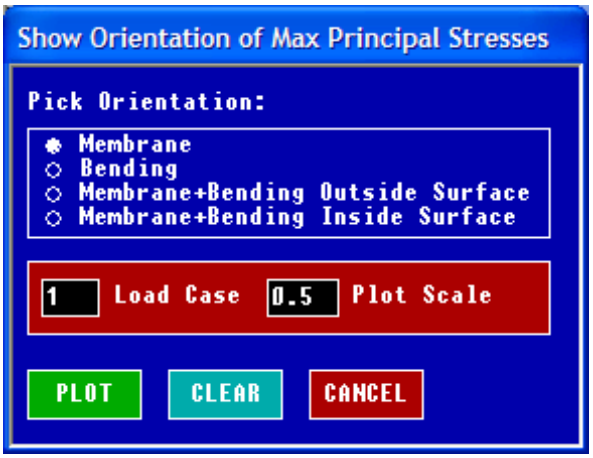
Model Properties Plotting – Collapsed elements can be displayed in the 3D viewer for ease of identification, and any material property – modulus, etc. can be displayed graphically for verification.



Stress Directions – Membrane, bending, or membrane+bending on the inside or outside surface maximum principle stresses can have their magnitude and orientation plotted. This is particularly helpful when evaluating cracked geometries in a complex stress state. The user can also color contour any local stress tensor component. Each of these features is accessed through the *"Shell Element Tools"* in the *"Advanced"* menu.







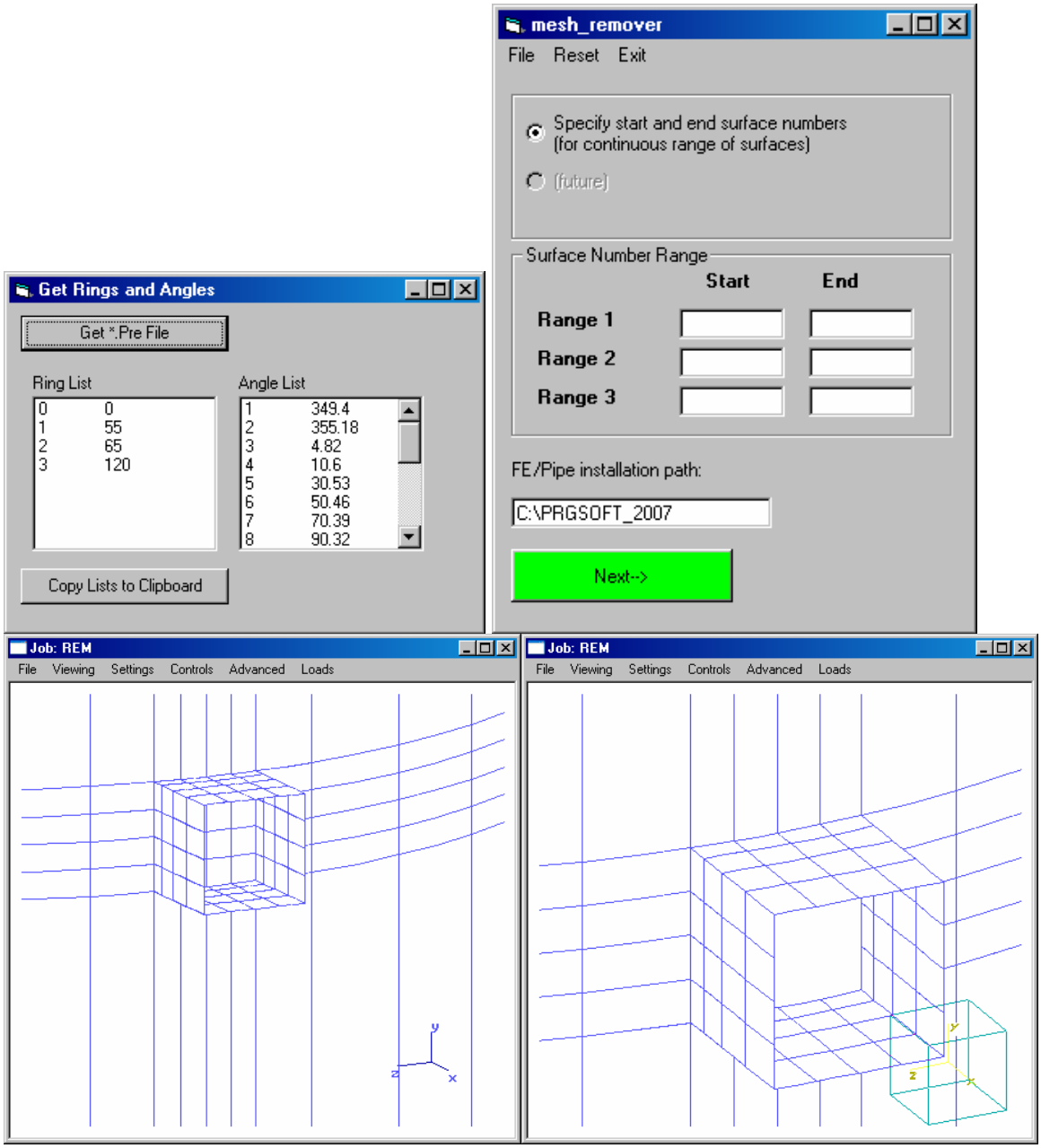
Two new model building utilities can be launched from the "shell element tools" in the "Advanced" men: "Get rings and angles" and "Mesh remover". Get rings and angles". The "Get rings and angles" utility finds the "integer" value of rings and angles that are called for in the "Global user surface controls" of the "general nozzles plate and shells template". Rings and Angles are displayed in the utility, and can be copied to MSWord or Excel in tab delimited format.

"Mesh remover" is a utility designed to create square nozzles using alt-meshed plates and breach openings. Using "Mesh Remover":

Model Checking or Run Analysis
Step 1 – build the basic model (without the "hole")
Step 2 – determine the surface ID's using "Stamps" from the "Settings" Menu (only whole surfaces can be un-meshed)

Step 3 – For model checking, use “plot model”. To generate stresses in the modified model, use “Prepare the model for analysis”.
Step 4 – Launch “Mesh remover” from “shell element tools”
Step 5 – Input the surface number ranges (in order)
Step 6 – Click “Next” and navigate to the <jobname>. Pre file.
Step 7 – Repeat steps 5 through 6 as required (eg. for multiple surface holes)

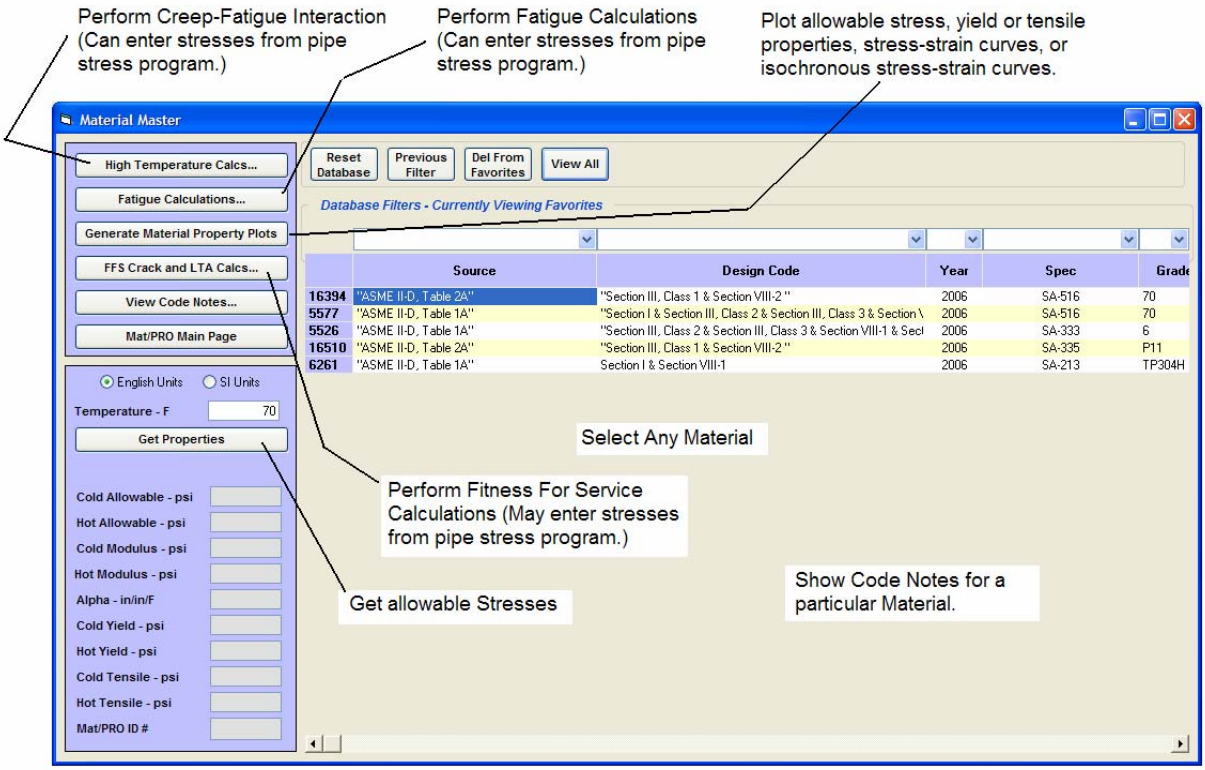
Analysis Only
Step 9 – Click “Analysis” in the PRGmaps window
Step 10 – Review results.



(Before Mesh Remover) (After Mesh Remover)

Section 6: Mat/PRO Version 2.0

MatPRO provides a variety of useful interfaces for plotting and data look up of ASME and B31.3 Code material properties, and extends those material data sets with data from API 579 and API 530, and methods from ASME Section III, Subsection NH (previously N47). MatPRO version 2.0 provides a "User's Favorites" interface that let's the user list favorites from any available data base, including ASME and/or B31.3. The favorites MatPRO interface is shown below:



Columns are user sizeable, and material order can be changed by dragging and dropping the material in the favorites list. (This way, the user can order the materials as he prefers to see them.)

B31.3 2004 Materials are available.

Section 7: Mesh/PRO Version 3.0

3.2 Shell Regions

Regions are groups of model surfaces that are uniquely defined for post-processing and/or joining to other models. A single mesh surface may exist in one or more regions. For instance, in many nozzle-shell junctions, surfaces surrounding the nozzle, located in the shell, may exist in regions such as “Header at Nozzle” and “Header”. Therefore, these elements surrounding the nozzle are assigned to two separate regions for post-processing purposes.

Regions can be defined to include entire surfaces or only portions of surfaces for post processing. “Join” type regions can be defined to allow the Mesh/PRO models to be included in database operation for joining to other FE/Pipe or Mesh/PRO models.

Regions have the following uses in Mesh/PRO:

1. Assign surfaces to regions for post processing purposes and identification in the output reports.
2. Assign surfaces to “no-stress” regions so that stress are not calculated for the nodes included in the specified region.
3. Assign surfaces to “join” regions for joining to other shell models or piping/beam models.

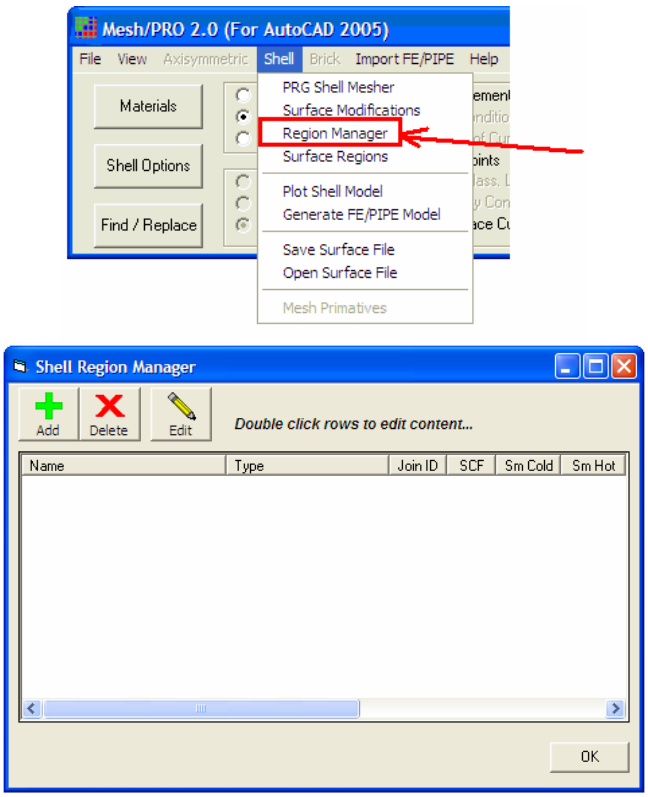
An example of Mesh/PRO’s shell joining and region operations can be found in the samples folder in the installation directory:

<installation>\Samples1\MeshPRO_Regions

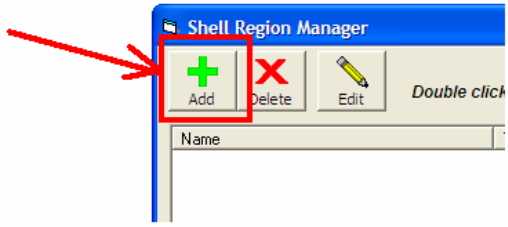
<u>Summary of Region Creation</u>

Once the surface has been added to the Mesh/PRO database, the following steps are necessary to add the surface to a particular region. Note that these steps assume that the user has already defined the surface mesh and added the surface mesh to the Mesh/PRO model using the Shell Mesher interface tools.

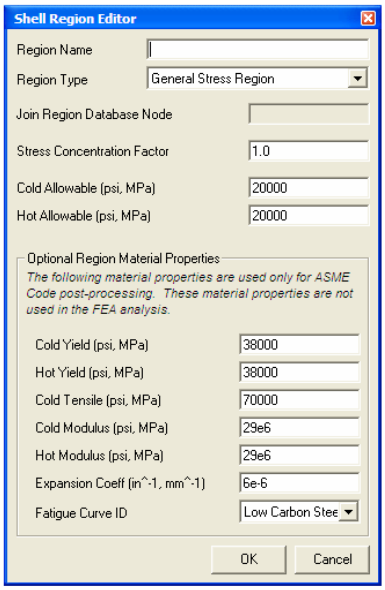
1. Open the REGION MANAGER screen. This can be done either thru the drop-down menus in the main Mesh/PRO screen or thru the Surface Regions screen.



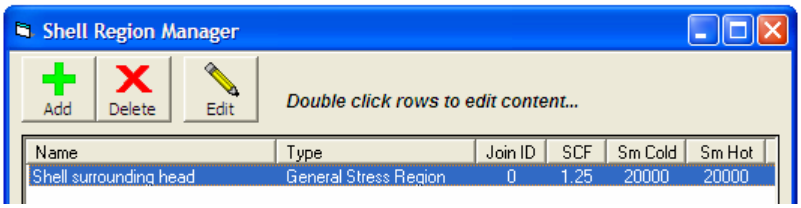
2. Once the REGION MANAGER screen appears, click the “ADD” button located in the tool bar. This will open the SURFACE REGION EDITOR screen where new surface regions are added to the model (or existing surface regions are edited).



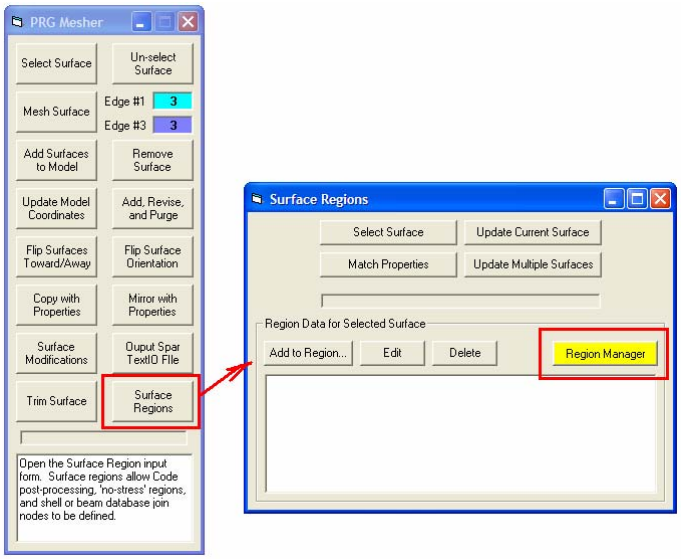
3. In the SURFACE REGION EDITOR screen, fill in all the required data for the new surface region which is to be added to the Mesh/PRO model. When finished, click OK. Note that most of the material data is optional.



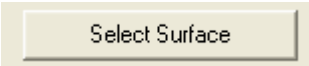
4. The newly created surface region should now appear in the region listing within the SHELL REGION MANAGER screen.



5. Now that the new surface has been created, open the SURFACE REGION screen. The SURFACE REGION screen is used to assign the nodes within a particular modeling surface to one or more surface regions (which are created as shown in Steps 1-4).

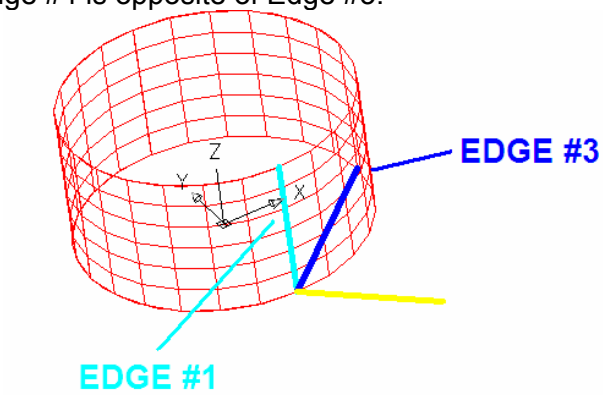


6. Select the surface which will be regionalized by choosing the Select Surface button. Click the Select Surface button, and then click on a surface in the AutoCAD screen which is currently included in the Mesh/PRO model. If the surface already has been included in existing surface regions, those will appear in the listing at the bottom of the SURFACE REGION screen.

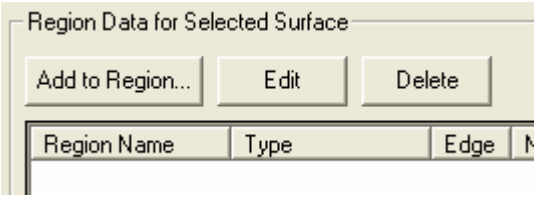


NOTE – the surface mesh must already have been added to the Mesh/PRO model using the Shell Mesher or an error message will be generated.

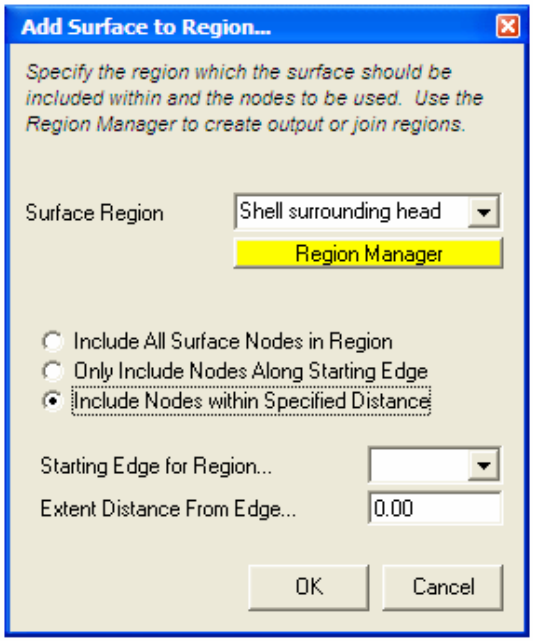
7. Once the surface has been selected, the orientation vector should appear. The CYAN line marks edge #1 and the BLUE line marks edge #3. Edge #2 is opposite of Edge #1 while Edge #4 is opposite of Edge #3.



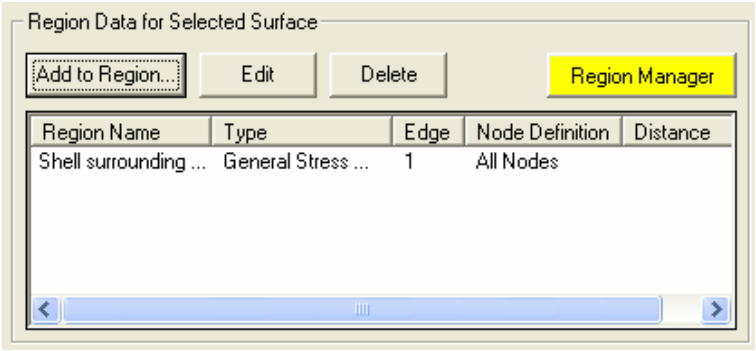
8. Next, click the Add to Region button. The Add to Region button allows the user to assign the selected surface to one or more surface regions.



9. When the Add Surface to Region screen appears, select the Surface Region which the selected surface shall be added to. In addition, specify the nodes which are to be included in the surface. Options for nodes to be included in the region are: all nodes, nodes only along one edge of the surface, or nodes starting on an edge extending a specified distance into the surface.



10. After completing all the required input in the Add Surface to Region screen, the newly created surface should appear in the Surface Regions listing. Use the Edit button to edit any of the defined surface region information or the Delete button to remove a surface from a particular region.



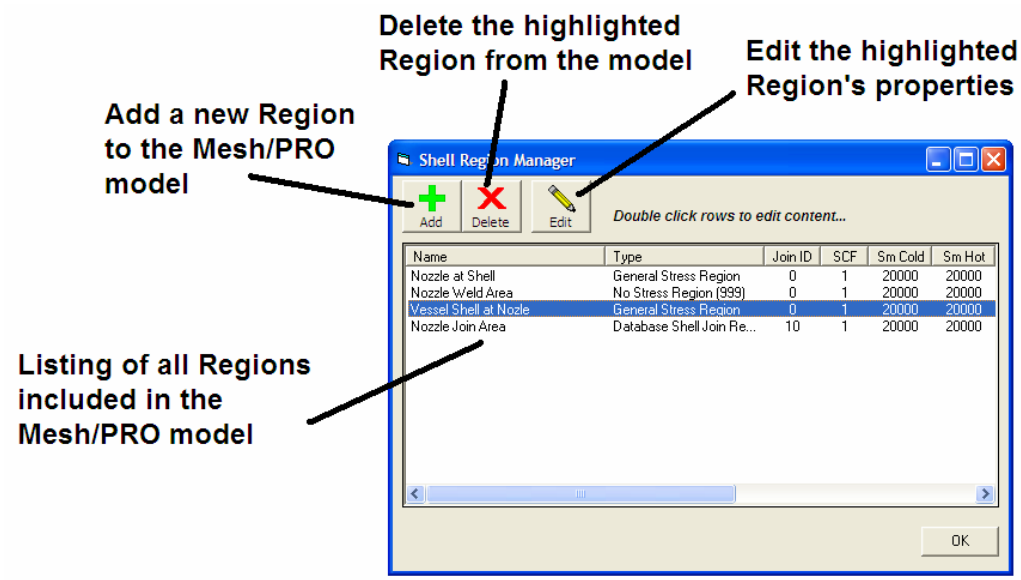
11. Repeat Steps 8-10 for each region in which the surface shall be included.

12. Click the Update Current Surface to un-select the current surface.

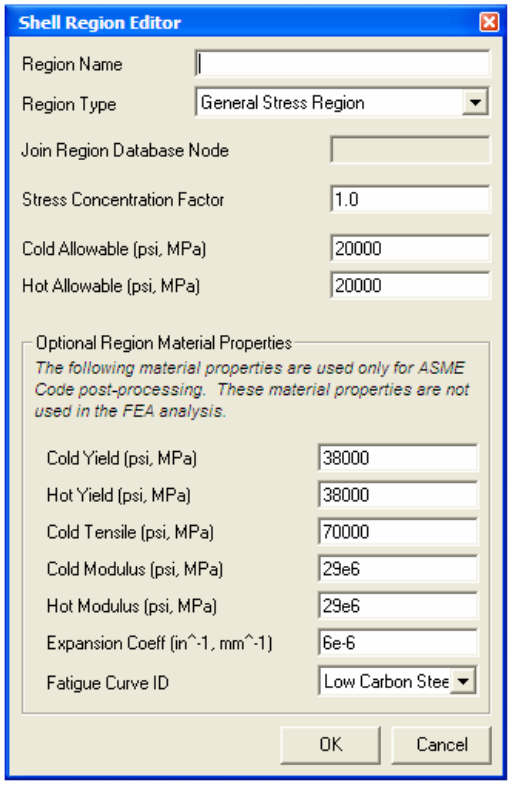
SHELL REGION MANAGER

The SHELL REGION MANAGER screen is used to add, modify, or delete entire Regions from the Mesh/PRO model. The SHELL REGION MANAGER screen is used to create regions which may then be used for modeling in Mesh/PRO. Once Surface Regions are defined, modeling surfaces are added to the Surface Regions using the Surface Regions input screen.

Add	Add a new Region to the list of surface regions for the current model. This button will open the REGION EDITOR screen (see description in next section).
Delete	Delete the region which is currently highlighted from the model.
Edit	Edit the region which is currently highlighted in the region listing. This button will open the REGION EDITOR screen (see description in next section).



SHELL REGION EDITOR



Region Name

Use the Region Name input box to specify a unique name which will be used to identify the region when reviewing the post-processed ASME Code stress results.

Region Type

Select the Region type to be used. The following options are available:

1. General Stress Region – this is used for assigning the surfaces to general post-processing regions for ASME results. The stress results in these regions will be compared against the allowable stresses and Code determinations made.

2. No Stress Region – this region type will “turn off” stress results for the any surfaces or parts of surfaces assign to a No Stress Region. Typically, the No Stress Region option is used for areas containing singularities such as shell-plate junctions or the weld regions at nozzle-shell junctions.

3. Database Shell Join Region – this option is used to join the ends of Mesh/PRO shell models to other shell models using the FE/Pipe database model operations. The surfaces assigned to any Database Shell Join Region must create a closed loop around the circumference of the nozzle or shell. If a closed loop is not created then the Mesh/PRO model will not be properly joined to the FE/Pipe database models.

4. Database Pipe/Beam Join Region – same as the Shell Join Region, except that these surface will be joined to a pipe or beam element in FE/Pipe.

Join Region Database Node
If the Region Type option is a JOIN type region (options 3 or 4), then specify a numeric integer value between 1 and 800. This join node should match the join node number of the model to which the Mesh/PRO model will be joined.

Cold Allowable
Specify the cold allowable stress for the region. This value is required and is used in the post-processing for ASME Code stress results.

Hot Allowable
Specify the hot allowable stress for the region. This value is required and is used in the post-processing for ASME Code stress results.

Cold Yield
Optional – if the cold yield stress is specified, then it will be used in one or more of the following ASME Code stress check procedures:

1. ASME Section VIII-2, 5-130 thru wall thermal gradient ratcheting check (or ASME III, NB 3200).
2. ASME Section VIII-2, 4-136.7 “Simplified Elastic Plastic Analysis” requires that the ratio of Sy to Su is less than 0.80.
3. Selection of appropriate fatigue curves for Wrought 70 Copper-30 Nickel alloys not exceeding 700°F (ASME Section VIII-2, Figure 5-110.3).

Hot Yield
Optional – if the hot yield stress is provided and a thru wall thermal gradient exists, then the hot yield will be used for thermal ratcheting checks given in ASME Section VIII-2, 5-130 (or ASME III, NB 3200).

Cold Tensile
Optional – if provided, the cold tensile stress will be used for one or more of the following ASME Code stress checks:

1. ASME Section VIII-2, 4-136.7 “Simplified Elastic Plastic Analysis” requires that the ratio of Sy to Su is less than 0.80.
2. Selection of appropriate fatigue curves for carbon steel, low alloy, Series 4xx stainless steels, and high alloy steels (ASME Section VIII-2, Figure 5-110.1).

Cold Modulus
Optional – the cold modulus entered in the REGION EDITOR screen does not influence the modulus of elasticity used in the finite element analysis. The value for the cold modulus to be used in the analysis is specified in the materials

screen. This entry will only be used for "join" type regions to define the modulus of elasticity of the join elements created between the two shell sections.

Hot Modulus
Not used at this time.

Expansion Coefficient
Optional – the thermal expansion coefficient entered in the REGION EDITOR screen does not influence the modulus of elasticity used in the finite element analysis. The value for the thermal expansion coefficient to be used in the analysis is specified in the materials screen. This entry will only be used for "join" type regions to define the thermal expansion coefficient of the join elements created between the two shell sections.

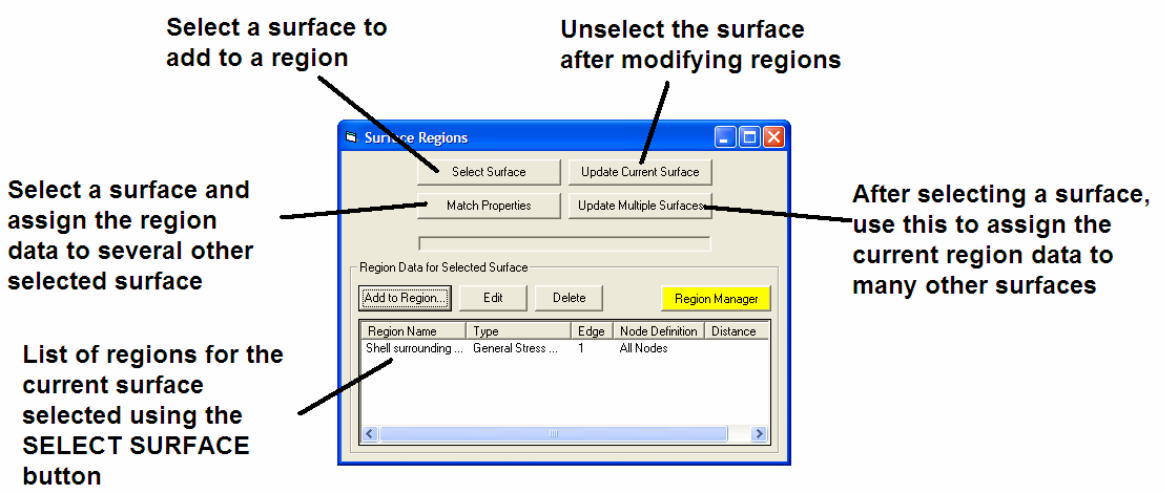
Fatigue Curve ID
Optional – specify the fatigue curve to be used in the ASME Code fatigue post-processing. The following options are available:

1. Low Carbon Steels (ASME VIII-2, Fig. 5-110.1)
2. Low Alloy Steels (ASME VIII-2, Fig. 5-110.1)
3. High Tensile Steels (ASME VIII-2, Fig. 5-110.1)
4. Austenitic Stainless Steels, Ni-Cr-Fe, Ni-Cu (ASME VIII-2, Fig. 5-110.2)
5. 70Cu-30Ni Alloys (ASME VIII-2, Fig. 5-110.3)
6. Ni-Cr-Mo-Fe Alloys (ASME VIII-2, Fig. 5-110.4)

SURFACE REGIONS
Use the Surface Region screen to select surfaces in the Mesh/PRO model and assign them to Surface Regions.

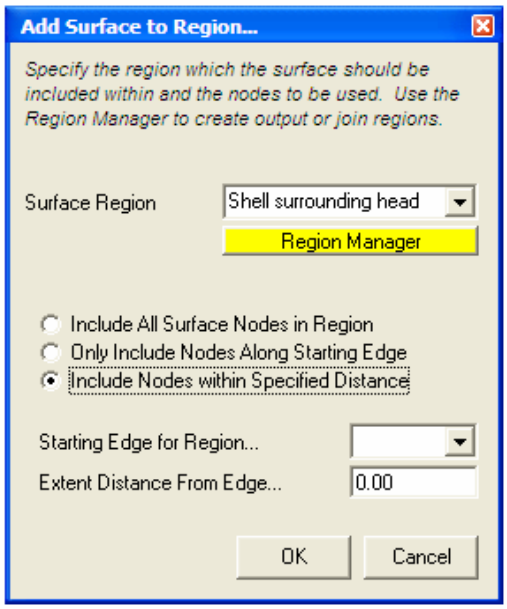
Select Surface	Select a surface to modify the region data for. Once the surface is selected, the surface may be added to existing regions, the region list can be edited, or the surface removed from specified region.
Update Current Surface	Use this button to un-select the current surface which was selected using the Select Surface button.
Match Properties	Similar to the Match Properties command in AutoCAD. Click this button, then select the parent surface, and then select the children surfaces which will inherit all the region data of the parent surfaces.
Update Multiple Surfaces	Use this button to select on or more surfaces which will be assigned the same region data as the current surface which is already selected.

Add to Region...	After a surface is selected using the Select Surface button, click the Add to Region button to add new surface regions to the existing list of regions for the current surface.
Edit	Use this button to edit existing region data for the current surface. Highlight a region entry in the surface region listing and then click the Edit button to modify the region's properties.
Delete	Highlight an existing region entry in the surface region listing then click the Delete button to remove the current surface from the highlighted region.
Region Manager	Use this button to access the Surface Region Manager screen. This screen is used to create surface regions for the Mesh/PRO model.



ADD SURFACE TO REGION

The Add Surface to Region screen is used to define or edit existing surface region entries for the current surface which is selected.



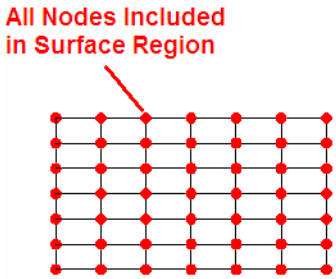
Surface Region

Select the surface region to which the currently selected surface should be added. Regions can be added to the Mesh/PRO model using the Region Manager button and screen.

Surface Region Node Options

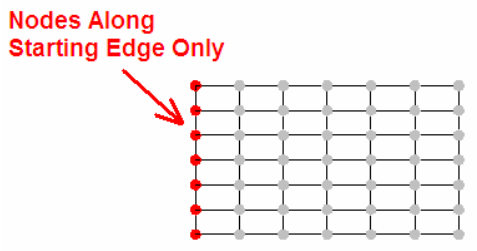
When a surface is added to a region, there are several options which control the nodes which are included within that region. The following options are available in Mesh/PRO:

1. Include All Surface Nodes in Region – All nodes in the selected surface will be included in the surface region.

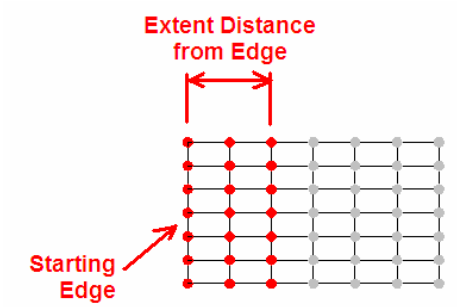


2. Only Include Nodes Along Starting Edge – Only the nodes along the starting edge for the surface region will be included in the region. The starting edge is selected from the Starting Edge for Region

combobox. Note that if the Surface Region specified is a “join” type region, then “Only Include Nodes Along Starting Edge” is the sole option available since join regions only occupy the edge of the surface.



3. Include Nodes Within Specified Distance – Only the nodes within the zone from the starting edge to the distance from the starting edge will be included in the surface region.

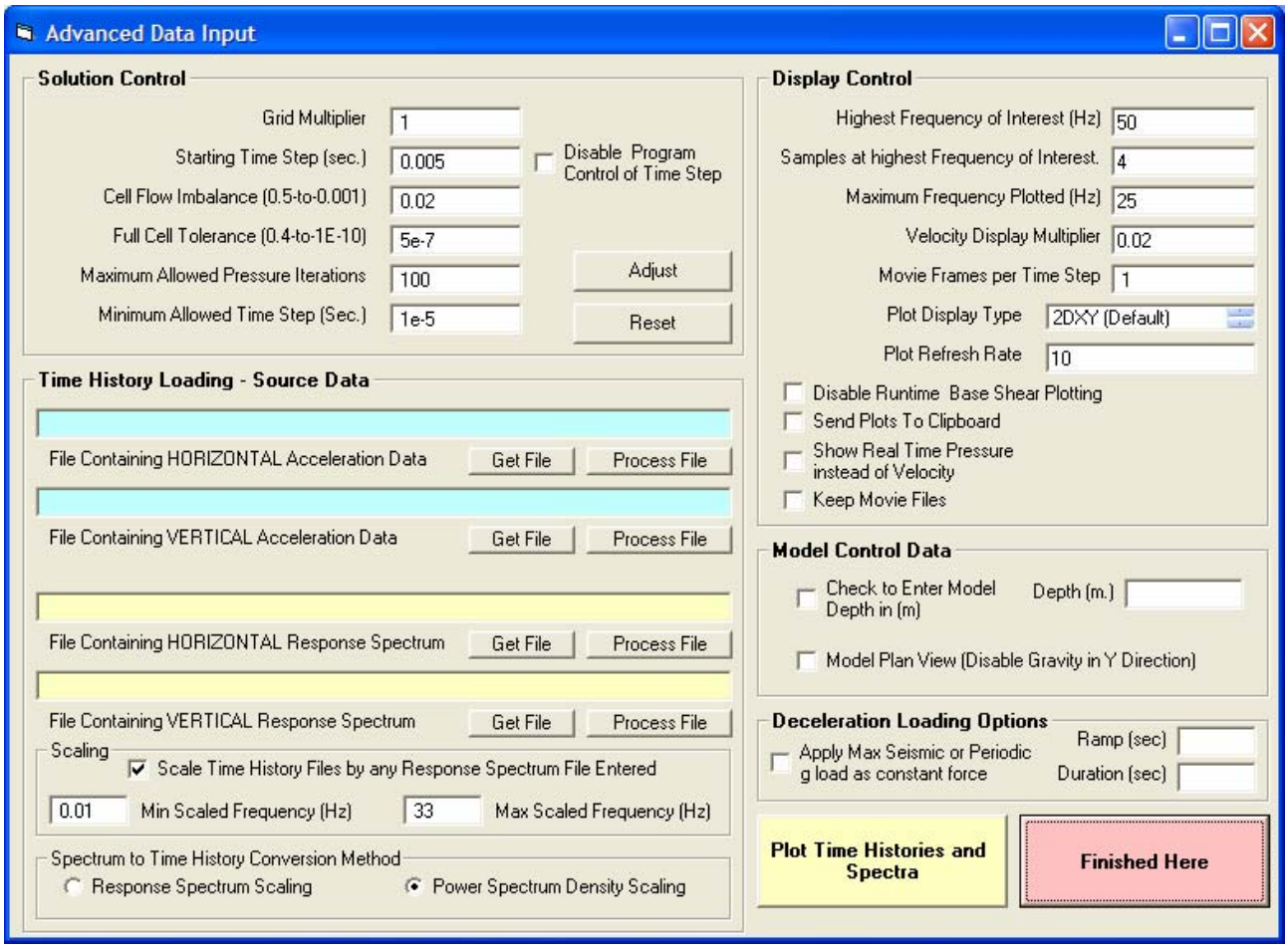


Extent Distance From Edge

Only available when the Node Option is “Include Nodes within Specified Distance”. Use this option to specify the width of the regionalized node zone, beginning at the starting edge, extending into the surface. Nodes within the zone will be included in the surface region, nodes outside the zone will be excluded from the region.

Section 8: Splash Version 3.0

Updated Advanced Options Control Screen



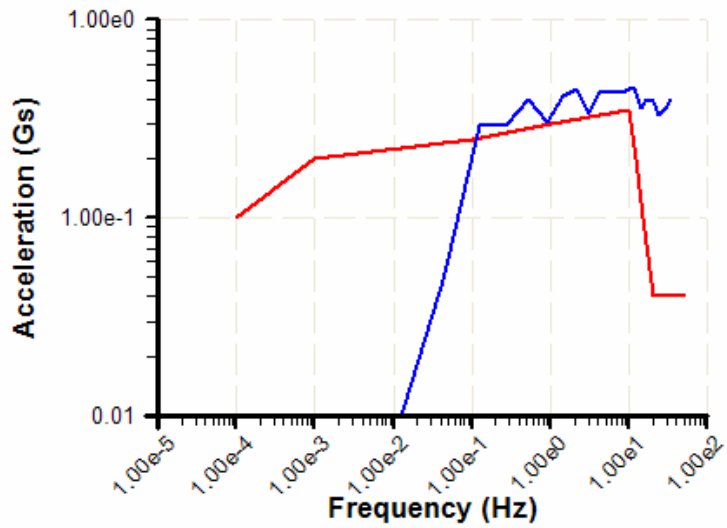
Items on the screen are grouped by function. Adjust and Reset buttons were added to the “Solution Control” section to with problems that tend to diverge.

There are several ways to use the Time History Source Data options:

1. Select a time history file from the PRG Time History Data base. The “Get File” button is pressed just below either the horizontal or vertical acceleration data file name text boxes. You should then navigate to the file of interest. The default file location for PRG software is found in <Installation Folder>\spectrum\Atime. Leave the response spectrum files blank and they will not be used or uncheck the “Scale Time History” checkbox to make sure that response spectra are not used to scale the time histories.

2. Select a base time history file. You can use the table below to find a site and/or earthquake that has similar characteristics to the maximum design earthquake. Enter the response spectrum that must be satisfied for the horizontal and/or vertical directions. The default location for response

spectra curves is in <Installation Folder>\spectrum\RFreq. Make sure the "Scale Time History" checkbox is checked. Generally, Power Spectrum Density Scaling produces the best result. This option is chosen by default. Once these two files are correctly selected, the run can be made. Note that there is a "Process File" button adjacent to each time history and response spectrum named textbox. The "Process File" button must be selected after each file is chosen for a particular loading direction (horizontal or vertical). Either the time history or the corresponding directional response spectrum "Process File" button can be pressed. A progress bar form will appear while the time histories are scaled and then. Once the progress bar disappears the data should be ready to use and you can select the "Plot Time Histories and Spectra" button. The scaled time histories or scaled responses can be plotted. An example spectrum plot is shown below.



The red line is the entered response spectrum and the blue line is the response spectrum of the synthesized time history to envelope the design response spectrum. Note that in the low frequency and high frequency ranges there cannot be correlation because the time history data set is of limited time extent and discretized only over the typical earthquake monitoring interval of 0.02 sec. If there is concern in the lower or higher frequency regimes the user can use the earthquake time scaling tools to either increase the total time and extend the low frequency content, or can reduce the time step which will extend the high frequency content of the scaled time history. In general neither of these limits restrict the accuracy of the solution so long as the selected time history has a sufficient duration to encompass needed low mode resonance.

As an example, a 200 ft. diameter storage tank has a first sloshing mode of vibration of approximately 0.12 Hz. A small table of first mode sloshing values when the height of the liquid is 1.5R is given below.

Diameter	Frequency	Period	(5) Period
200 ft.	0.12 Hz.	8.3 sec.	41.6 sec.
100 ft.	0.17 Hz.	5.88 sec.	29.4 sec.
50 ft.	0.24 Hz.	4.17 sec.	20.8 sec.
10 ft.	0.545 Hz.	1.83 sec.	9.2 sec
5 ft.	0.77 Hz	1.42 sec.	7.14 sec
2 ft.	1.218 Hz.	0.82 sec	4.11 sec.

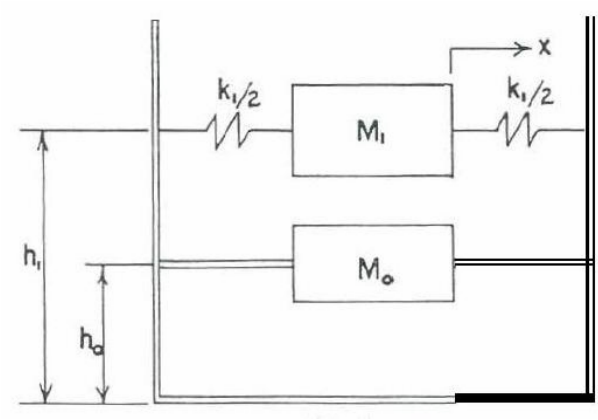
As the liquid level drops the first sloshing mode frequency drops as a function of the hyperbolic tangent to zero at a liquid height of zero. For tanks that are not fully filled, the mass is smaller, but the frequency is lower and may correspond to a structural mode.

This implies that the design procedure should have two characteristics:

1) Determine the worst case sloshing loads, and make sure that both statically and dynamically, the structure is sound for these loads. This will typically occur at the point where the tank or vessel is filled, or very close to being filled.

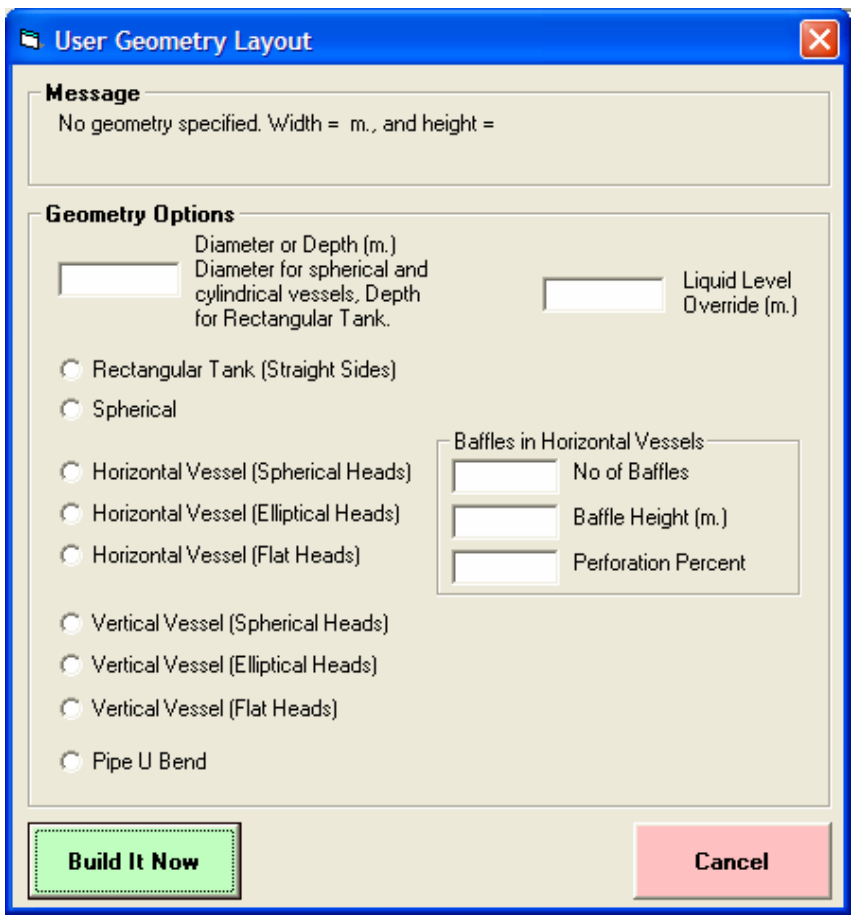
2) Determine the critical structural mode and frequency. If the critical structural mode is horizontal and at a frequency less than the tank frequency at a height of 1.5R, then the dynamic sloshing load at a height corresponding to that frequency should be calculated and used in an evaluation of the structure.

The usual concern with sloshing is that a dynamic response greater than a static response can occur. This is true depending on the frequency content of the excitation and of the container supporting structure. A simplified sloshing conceptual model is shown in the figure below. The mass M0 is the typical “static” mass that moves along with the vessel. The mass M1 is the “dynamic” mass that moves horizontally with the type of low frequencies shown in the table above. If this sloshing frequency corresponds to a structural frequency, then amplification can occur and the total load will be proportional to the mass times the dynamic amplification factor for repeated loads, which is typically five to ten for pipe and vessel type large body movements.



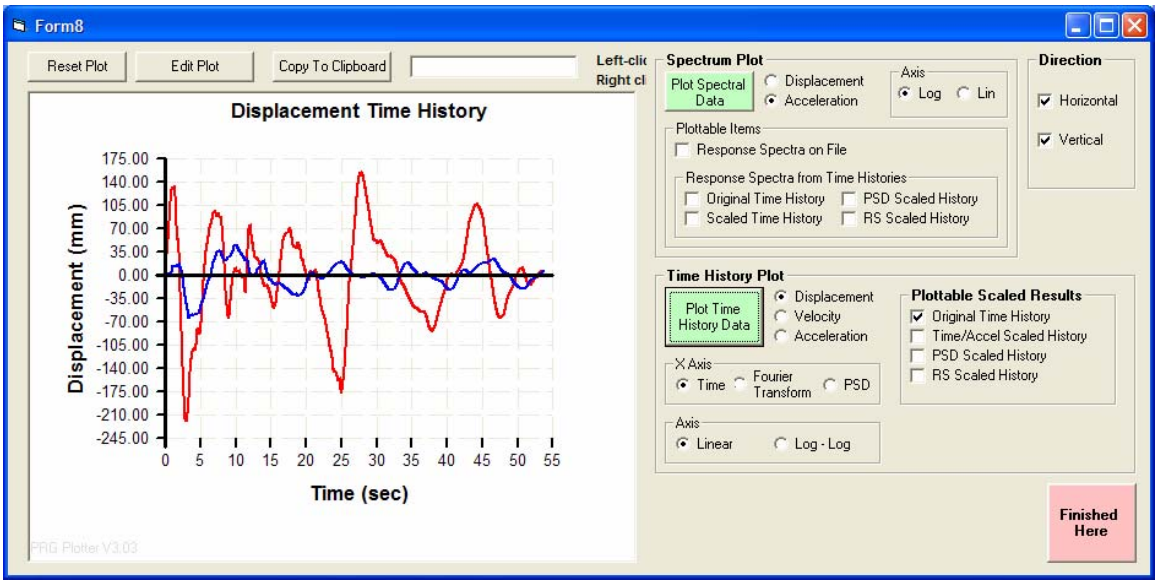
The sloshing interaction of the masses and the surrounding structure are included in DynaVessel, as this was one of the main problems it was designed to solve.

Updated User Geometry Layout Screen

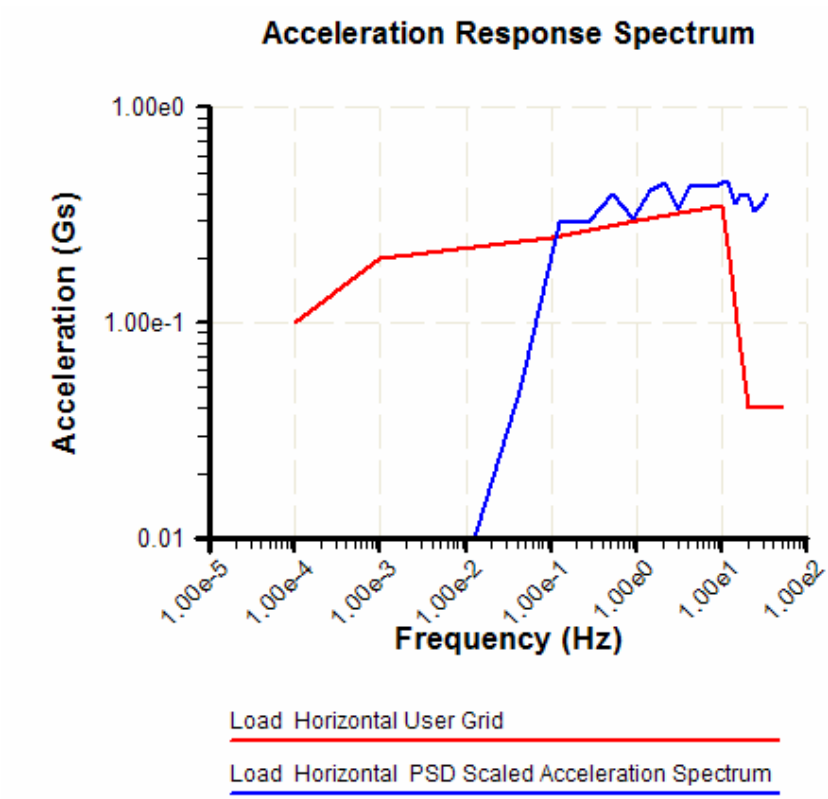


The User Geometry Layout Screen has been modified to include vertical and horizontal vessels with flat heads. The screen format has been changed, and the automatic input of baffles has also been added. You must now specify the diameter of the vessel and you can provide a liquid level override. You should be sure that the vessel is properly enumerated in the *"Message"* frame at the top of the Geometry Layout form.

2D Time History and Response Spectrum Plot Form

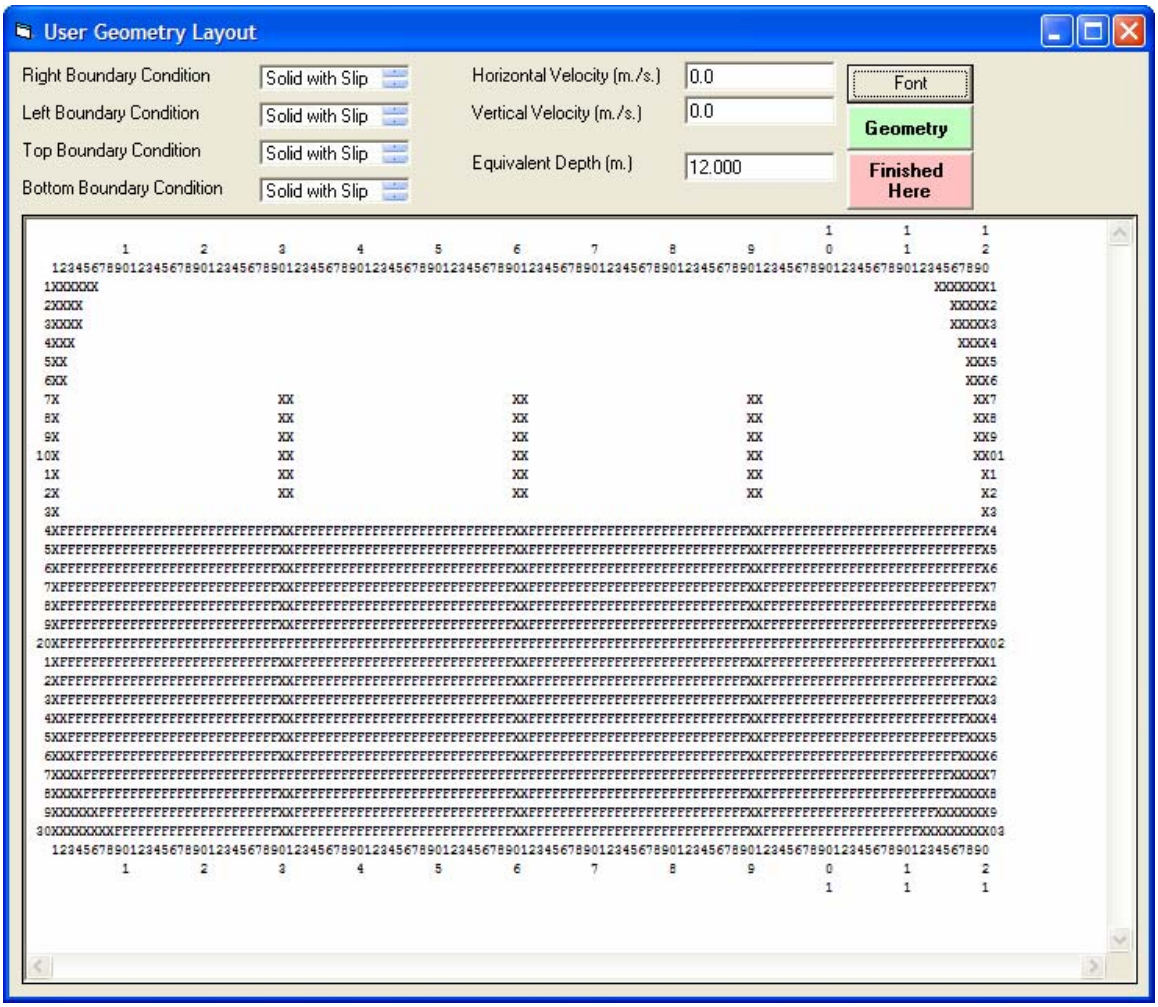


The spectrum plot options allow you to see how the response spectra from the synthesized time histories envelopes the design response spectra of interest. Examples are given below.



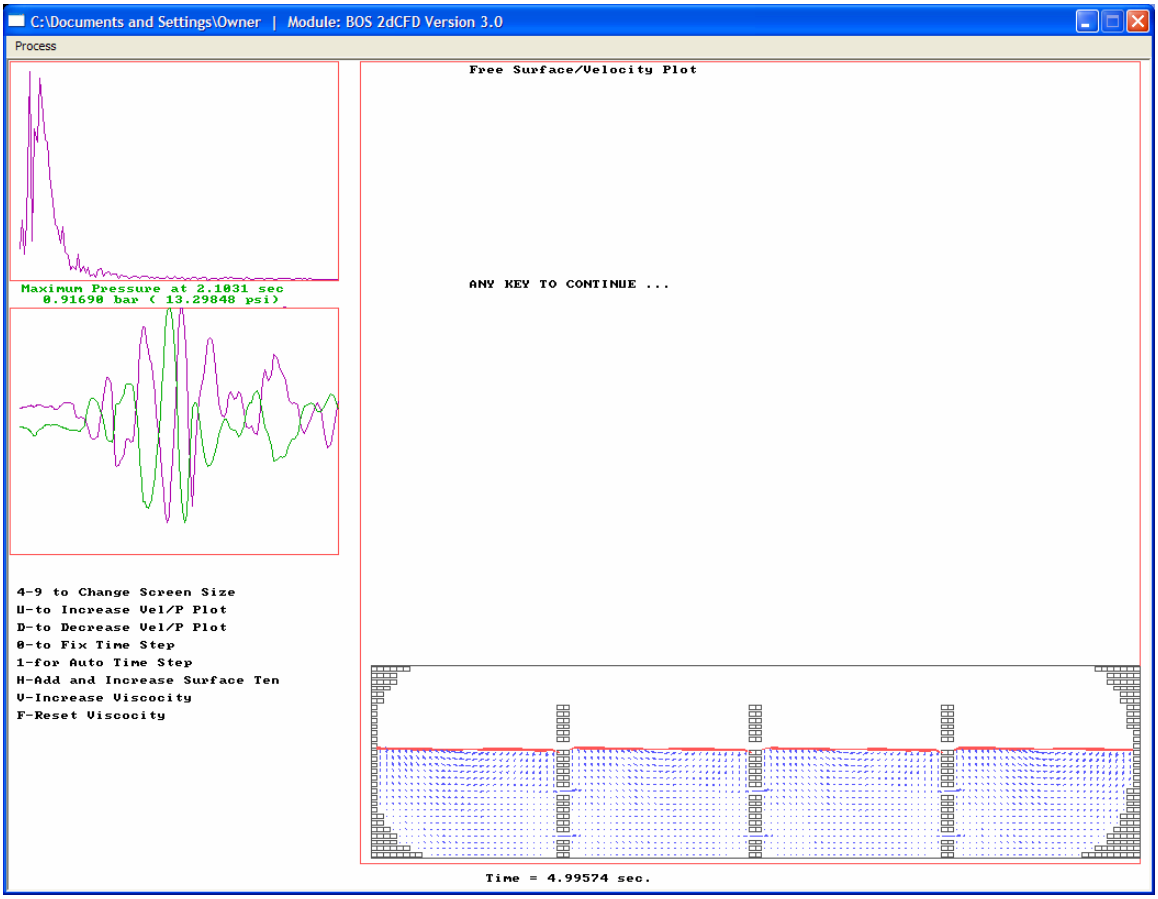
Scaled time histories can also be compared to the original time history, and both horizontal and vertical components can be plotted separately or together.

Splash User Grid Model with Automatic Perforations



20% perforated baffles option available in the new Geometry Layout screen is illustrated in the horizontal vessel schematic above. Baffles or impediments that exist internal to the flow area should be at least two cells wide or high. This is so that acceleration terms from extrapolated grids can go completely to zero when a fluid particle comes in contact with a wall.

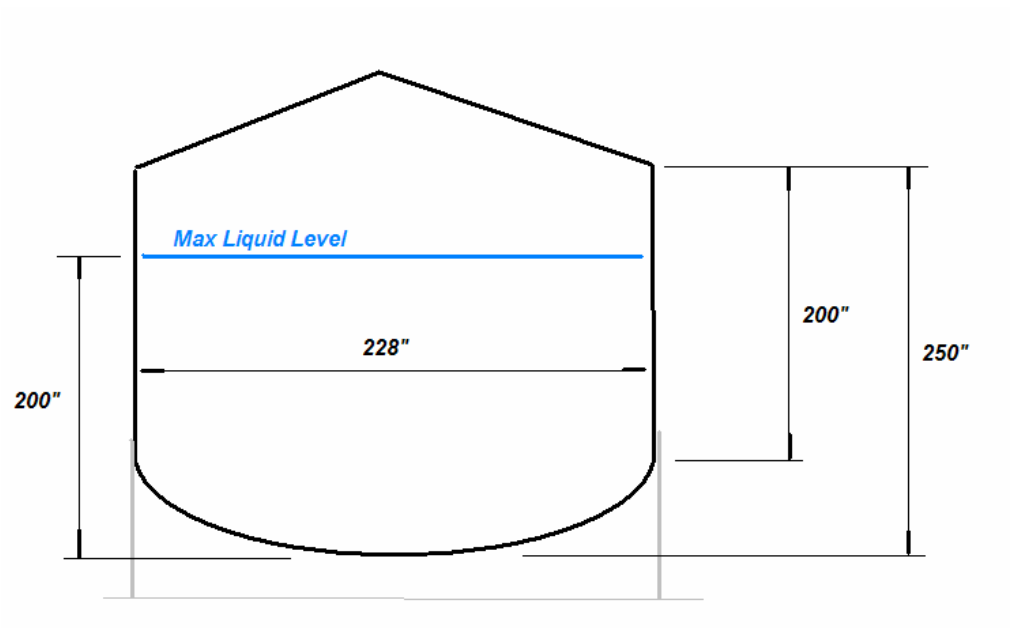
Splash Resizable Interactive CFD Solution Screen



The interactive solution screen can be resized to facilitate viewing the solution.

Example Splash User Defined Response Spectrum Loading

A vertical and horizontal response spectrum are to be applied to the following vessel:

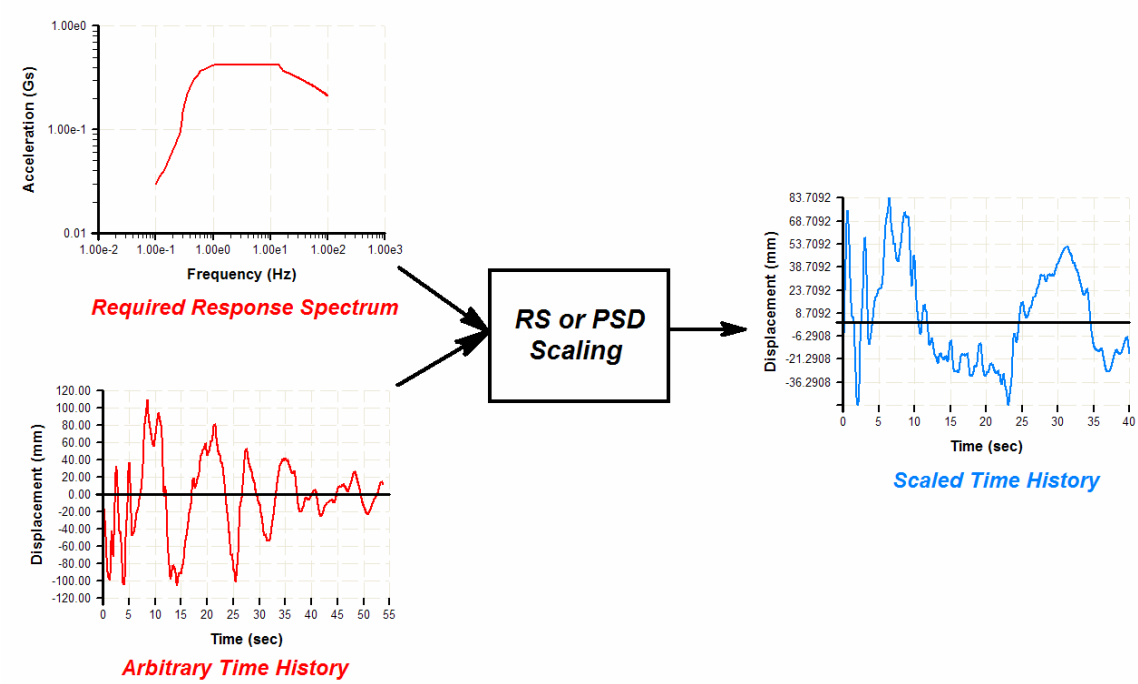


VOF free surface solutions operate in the time domain, while response spectra are specified in the frequency domain. To get from the frequency domain to the time domain a suitable time history waveform must be prepared. Splash adjusts time history waveforms to comply with an input response spectrum automatically.

The given time history waveform is transformed into the complex frequency domain using a Fourier transform. The Fourier transform is scaled using either of two methods – a Power Spectrum Density (PSD) method or a Response Spectrum (RS) method. Once the scaling is completed in the frequency domain, an inverse Fourier transform is performed to obtain the suitably scaled time history whose response spectrum envelopes the design response spectrum.

More comprehensive spectrum scaling methods are available in DynaVessel, and may be used there to produce time history data files that can be read into Splash if needed. (DynaVessel frequency scaling can be performed on a “stretched” time, on a shortened time step, or on a scaled history basis. The DynaVessel user may also provide scale factors that are frequency dependant (an equalizer). These advanced methods are not usually needed for the seismic excitation of liquid filled vessels however.

The process is outlined in the diagram below:



The response spectra to be used as a critiera for the vessel excitation is provided in an Excel spreadsheet as shown below.

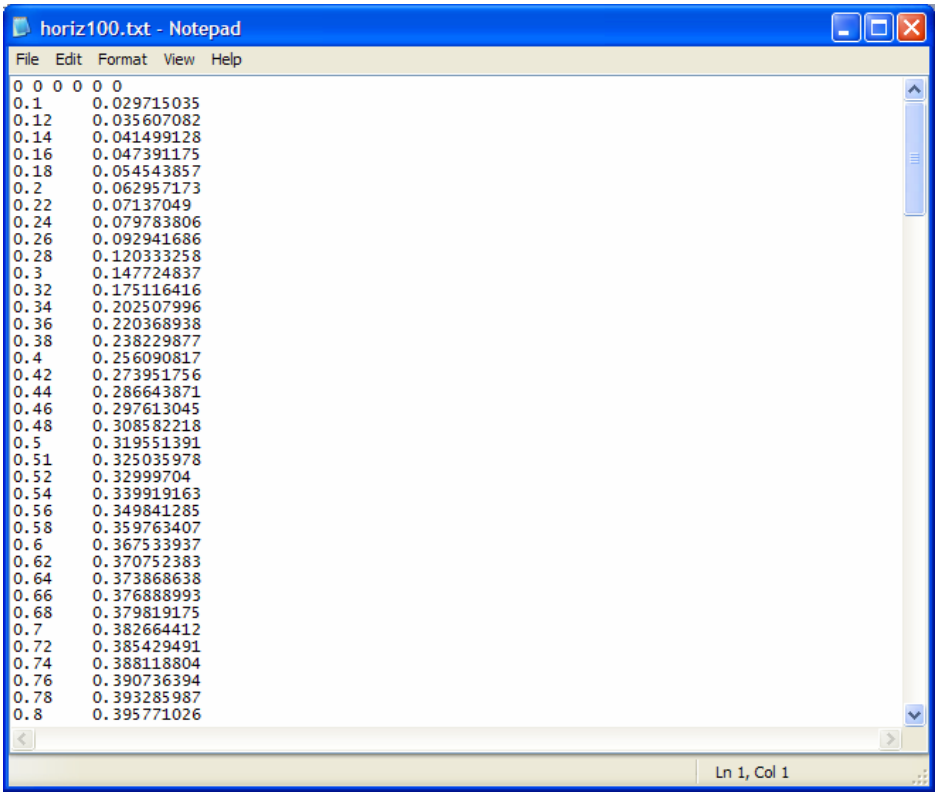
tal_IRS 6-7-06.xls

K	L	M	N	O	P	Q	R	S
Transverse Direction								
3%		4%		5%		2%		
Freq, HZ	SA, g's	Freq, HZ	SA, g's	Freq, HZ	SA, g's	Freq, HZ	SA, g's	Freq, HZ
0.1	0.031731894	0.1	0.030652094	0.1	0.029715035	0.1	0.033091842	0.1
0.12	0.038104553	0.12	0.036781152	0.12	0.035607082	0.12	0.03809003	0.12
0.14	0.044477211	0.14	0.042910211	0.14	0.041499128	0.14	0.043088218	0.14
0.16	0.05084987	0.16	0.049039269	0.16	0.047391175	0.16	0.048086406	0.16
0.18	0.058916799	0.18	0.056630397	0.18	0.054543857	0.18	0.055333265	0.18
0.2	0.068677998	0.2	0.065683596	0.2	0.062957173	0.2	0.064828794	0.2
0.22	0.078439197	0.22	0.074736795	0.22	0.07137049	0.22	0.074324323	0.22
0.24	0.088200397	0.24	0.083789993	0.24	0.079783806	0.24	0.083819852	0.24
0.26	0.103929334	0.26	0.098145543	0.26	0.092941686	0.26	0.098634467	0.26
0.28	0.13756149	0.28	0.128408152	0.28	0.120333258	0.28	0.129406344	0.28
0.3	0.171193655	0.3	0.158670767	0.3	0.147724837	0.3	0.160178229	0.3
0.32	0.204825819	0.32	0.188933383	0.32	0.175116416	0.32	0.190950114	0.32
0.34	0.238457984	0.34	0.219195999	0.34	0.202507996	0.34	0.221721999	0.34
0.36	0.2663505	0.36	0.237382828	0.36	0.220368938	0.36	0.258182939	0.36
0.38	0.292734557	0.38	0.255569655	0.38	0.238229877	0.38	0.294643881	0.38
0.4	0.317764976	0.4	0.273756481	0.4	0.256090817	0.4	0.331104823	0.4
0.42	0.341573902	0.42	0.291943307	0.42	0.273951756	0.42	0.367565765	0.42
0.44	0.364275027	0.44	0.310341006	0.44	0.286643871	0.44	0.398486055	0.44
0.46	0.385966873	0.46	0.328808996	0.46	0.297613045	0.46	0.427559462	0.46
0.48	0.406735377	0.48	0.347276986	0.48	0.308582218	0.48	0.456632869	0.48
0.5	0.426655946	0.5	0.365744976	0.5	0.319551391	0.5	0.485706275	0.5
0.51	0.436319354	0.51	0.374978971	0.51	0.325035978	0.51	0.500242979	0.51
0.52	0.445795112	0.52	0.37874251	0.52	0.32999704	0.52	0.512655688	0.52
0.54	0.449079354	0.54	0.386269582	0.54	0.339919163	0.54	0.537481103	0.54
0.56	0.452363585	0.56	0.393796648	0.56	0.349841285	0.56	0.562306517	0.56
0.58	0.455647816	0.58	0.401323714	0.58	0.359763407	0.58	0.587131929	0.58
0.6	0.458940412	0.6	0.405934244	0.6	0.367533937	0.6	0.604261743	0.6
0.62	0.462233008	0.62	0.408572538	0.62	0.370752383	0.62	0.609322262	0.62
0.64	0.465525604	0.64	0.411210831	0.64	0.373868638	0.64	0.614222103	0.64
0.66	0.4688182	0.66	0.413849124	0.66	0.376888993	0.66	0.618971156	0.66
0.68	0.472110796	0.68	0.416487418	0.68	0.379819175	0.68	0.623578424	0.68
0.7	0.475403392	0.7	0.419125711	0.7	0.382664412	0.7	0.628052129	0.7

The 5% frequency and acceleration row is highlighted and “copied” to the clipboard.

Open “Notepad” or some other word processing program and “paste” in the two columns of numbers.

Fill in the six values at the top of the file that correspond to the control values. When the acceleration values are given in g's and the damping percent is 5%, the control values should all be zero. For the data highlighted in the spreadsheet above, the top of the notepad document with the control record is shown below:




```
0 0 0 0 0 0
0.1     0.029715035
0.12    0.035607082
0.14    0.041499128
0.16    0.047391175
0.18    0.054543857
0.2     0.062957173
0.22    0.07137049
0.24    0.079783806
0.26    0.092941686
0.28    0.120333258
0.3     0.147724837
0.32    0.175116416
0.34    0.202507996
0.36    0.220368938
0.38    0.238229877
0.4     0.256090817
0.42    0.273951756
0.44    0.286643871
0.46    0.297613045
0.48    0.308582218
0.5     0.319551391
0.51    0.325035978
0.52    0.32999704
0.54    0.339919163
0.56    0.349841285
0.58    0.359763407
0.6     0.367533937
0.62    0.370752383
0.64    0.373868638
0.66    0.376888993
0.68    0.379819175
0.7     0.382664412
0.72    0.385429491
0.74    0.388118804
0.76    0.390736394
0.78    0.393285987
0.8     0.395771026
```

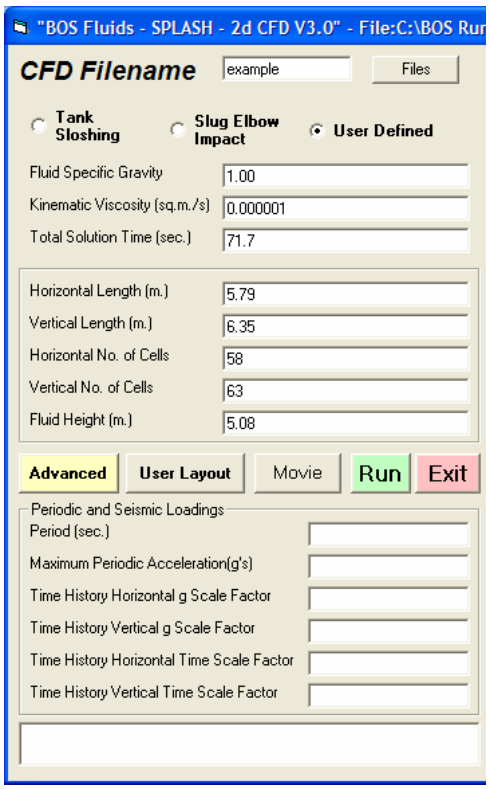
The file can be saved and the spectrum read into Splash or DynaVessel. The default location for spectral data files is <installation>\spectrum\Rfreq. The default location for time histories is <installation>\spectrum\Atime, although either file type can be stored in any convenient location. Each file type must be an ascii text file and end with the extension .txt.

The critical inventory (liquid) level for vertical vessels can be estimated as approximately 1.6R. The convective portion of the sloshing force will be a maximum from this level to the maximum fill height. The sloshing frequency will also be approximately the same from a height of 1.6R until the maximum height, and so it is conservative to use the maximum liquid level above 1.6R to find the greatest combination of impulsive and convective loadings. For the vessel detail shown above: 1.6R = 182" . This is measured from the bottom of the head, and so any liquid level at or above this value will result in the maximum dynamic loading. The maximum liquid level from the bottom head is 200" and so this value will be used in the calculation.

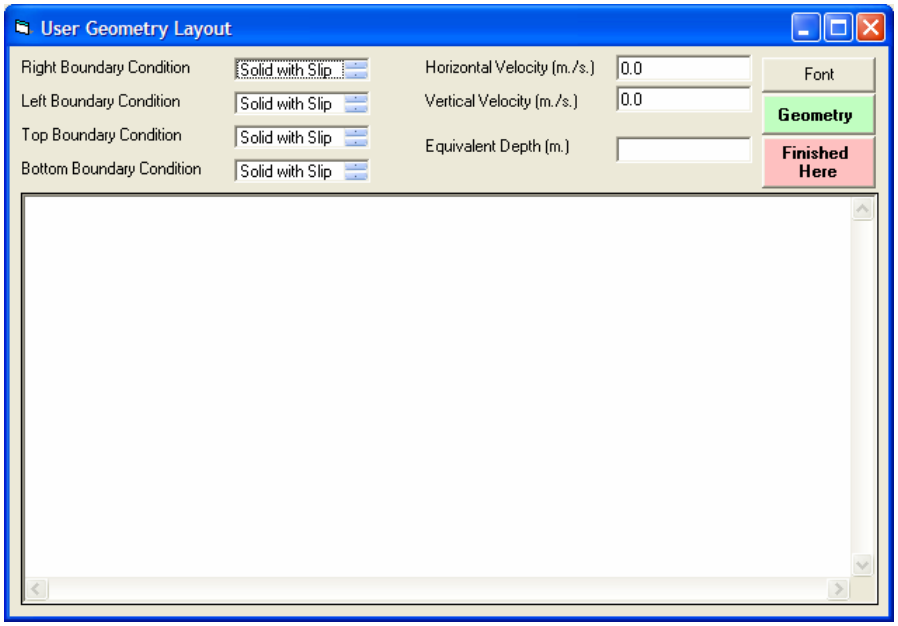
The top head will not be modeled as it will be assumed that the 50" freeboard is sufficient. The total model space height will be 250" and the model space width will be 228". There will be an elliptical head at the bottom of the vessel. Converting these to units for Splash:

(250) / (12) / (3.28) = 6.35 m. … vertical model size
(228) / (12) / (3.28) = 5.79 m. … horizontal model size
(200) / (12) / (3.28) = 5.08 m. … liquid level

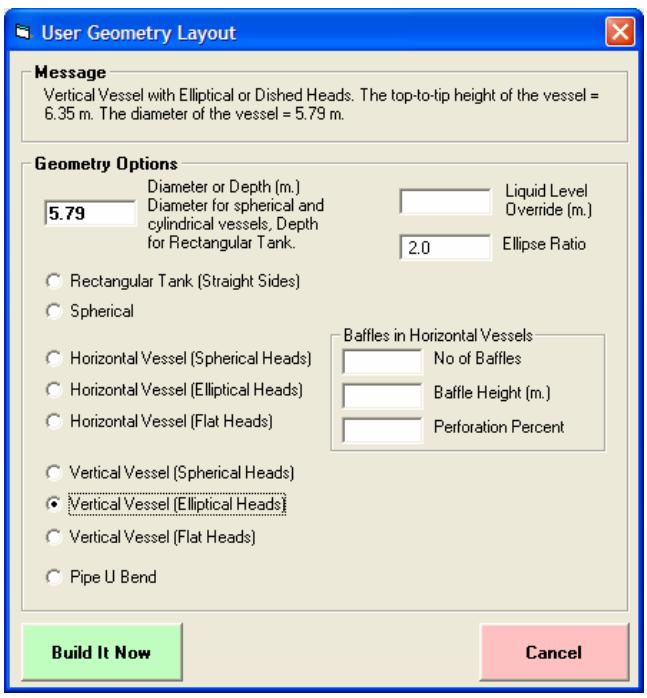
It is desired to have the total number of model cells less than 5000, and so the vertical model cell size will be 127/2 and the horizontal model size will be 116/2. This input to splash is shown below.



The User Layout button is used to finalize the model details:



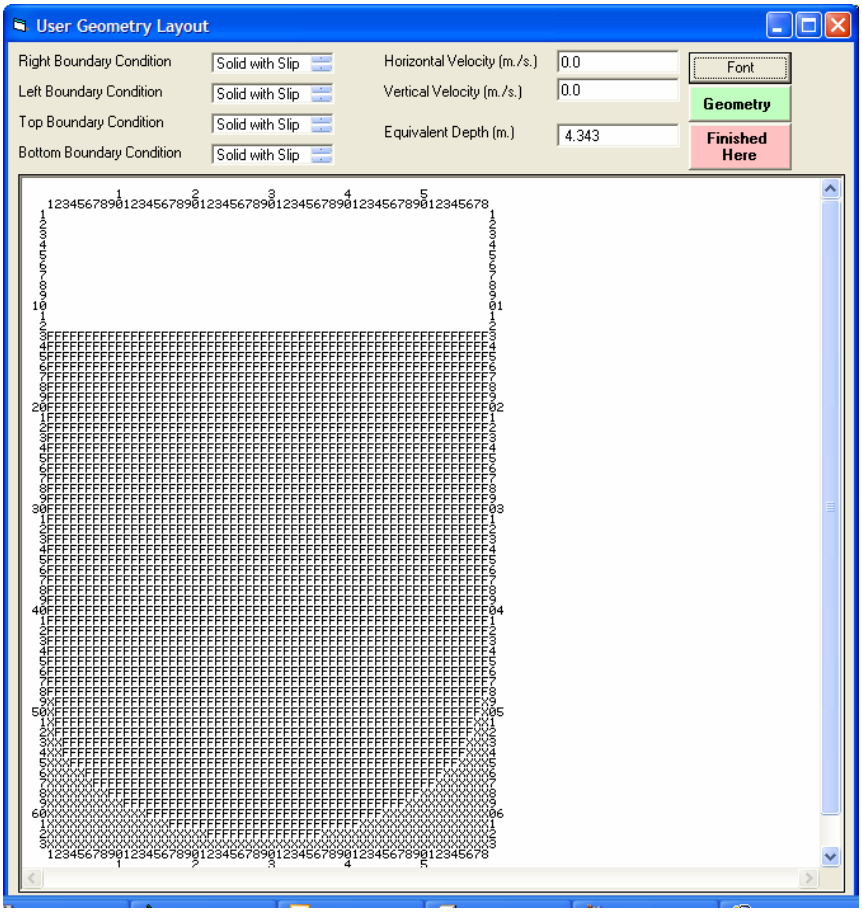
The “Geometry” button is used to describe the specific model to be constructed:



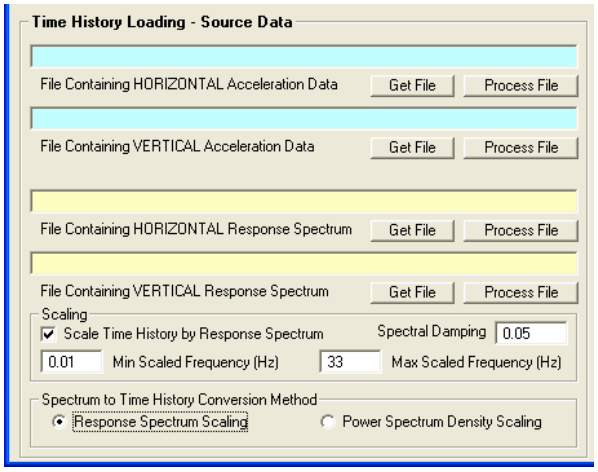
Click on the green “Build it Now” button:

The automatic model construction contains both heads. The model height space could be increased to accommodate the extra head on the top, or the user can remove the head using the ascii editing tools. The resulting image below represents the liquid level and freeboard to be analyzed. Note that even though each cell size is approximately as wide as it is long, the character space used for editing individual cells is established by the font family and will likely not correspond to the input. Graphic images used during

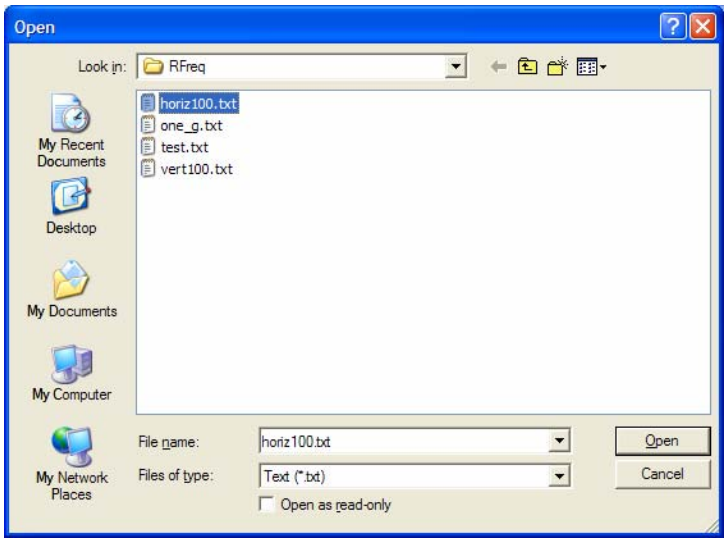
solution are scaled, and should give a more accurate representation of the vessel being analyzed. Note also that an equivalent model depth perpendicular to the page is computed so that accurate values of base shear and overturning moment are produced for the given geometry.



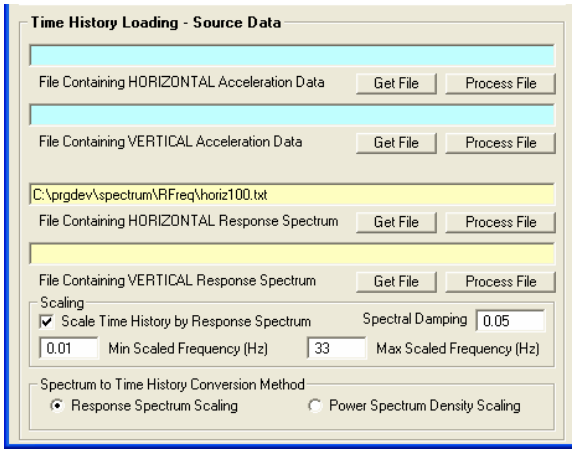
Once the model is constructed an appropriate time history must be synthesized to envelope the response spectra to be satisfied. (The user is referred to the DynaVessel user's guide for a more in-depth discussion of frequency-time history scaling.) The time history synthesis process will be demonstrated in detail for the horizontal component of the loading. An identical procedure is repeated for the vertical component of the loading. Time history scaling is initiated from the "Advanced Form:"



Use the “Get File” button to locate the horizontal response spectrum file saved in the step above. This is demonstrated below:

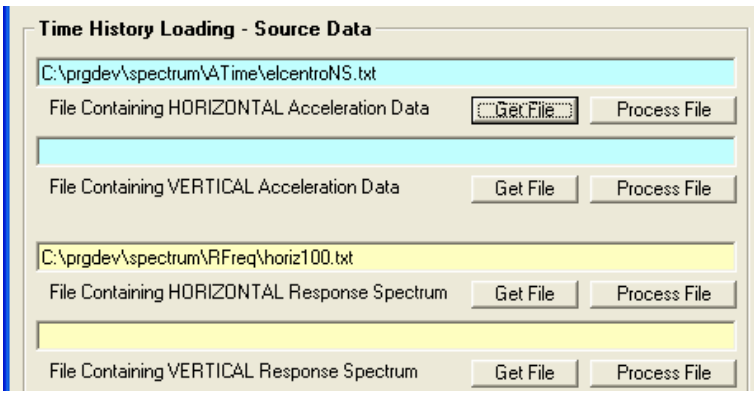


When the file is “opened” it’s name will appear in the Data Input text cell:



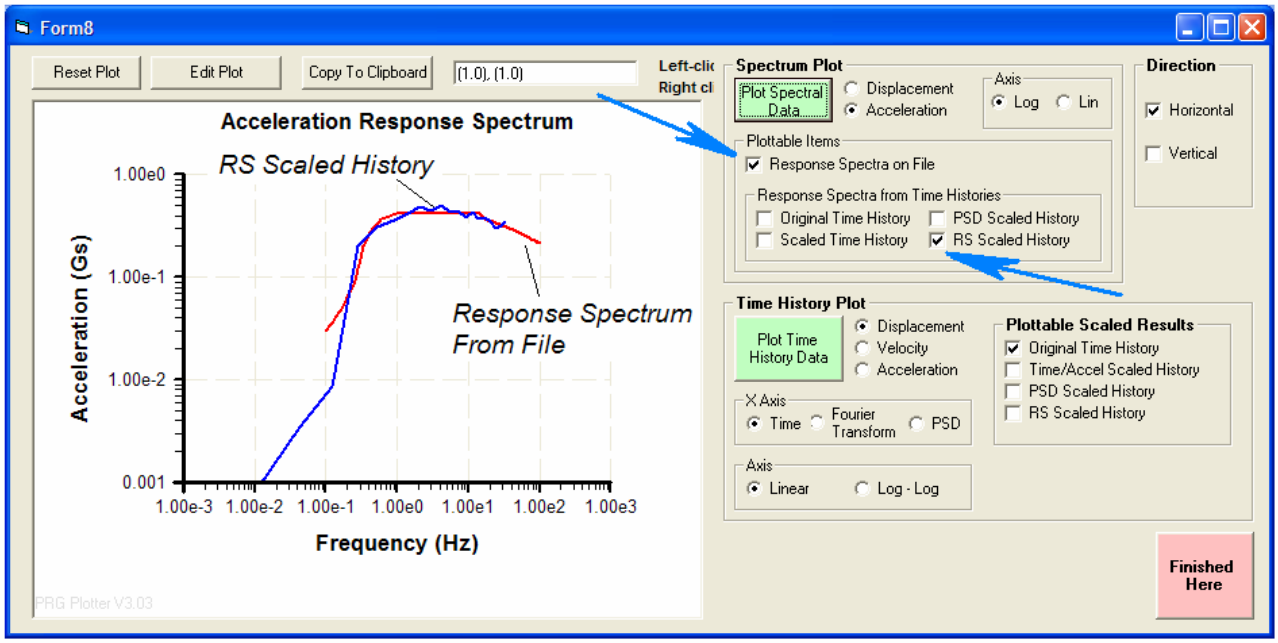
An arbitrary earthquake acceleration time history must now be selected. The EL Centrol NS component spectrum is a reasonable time history to select for this purpose. The earthquake acceleration time history used is not exactly arbitrary. It must contain reasonable frequency content over the range of interest. EL Centro spectra are good for this reason though.

Use the Get File button for the Horizontal Acceleration Data to select the EL Centro North-South time history and the Advanced form should appear:



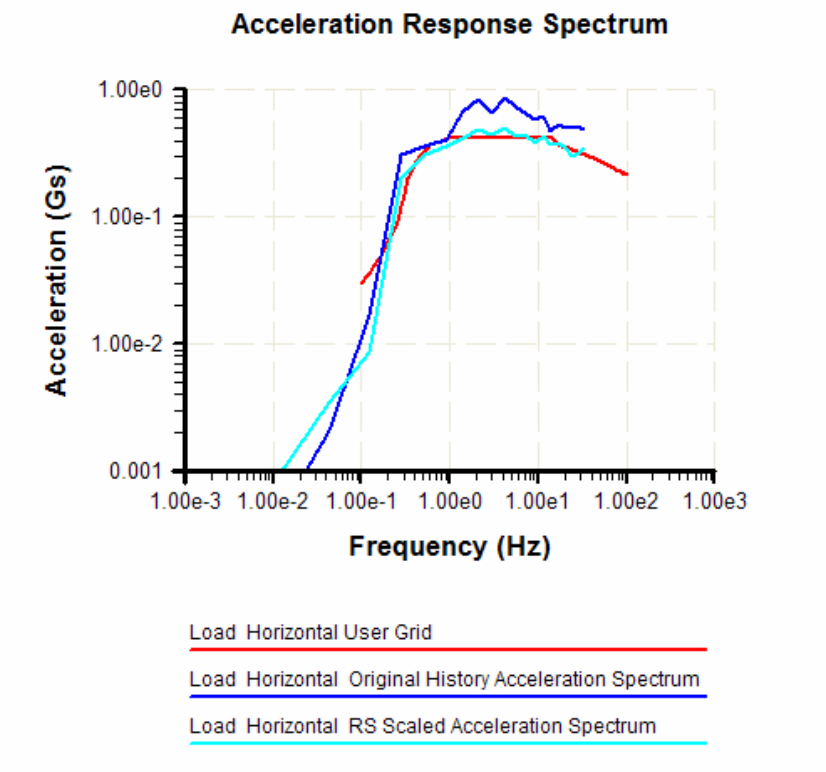
Once the time history and the spectrum are selected the “Process File” button to the right of either of the “Get File” buttons should be clicked.

A progressbar will appear on the screen and then disappear when the scaling has completed without error. To check the results click on the yellow “Plot Time Histories and Spectra” button. When the following screen appears, check the “Response Spectra on File” checkbox and the “RS Scaled History” checkbox as shown below and hit the green “Plot Spectral Data” button. Note how the scaled response spectrum in blue, follows the design response spectrum read from the user’s data file in red.

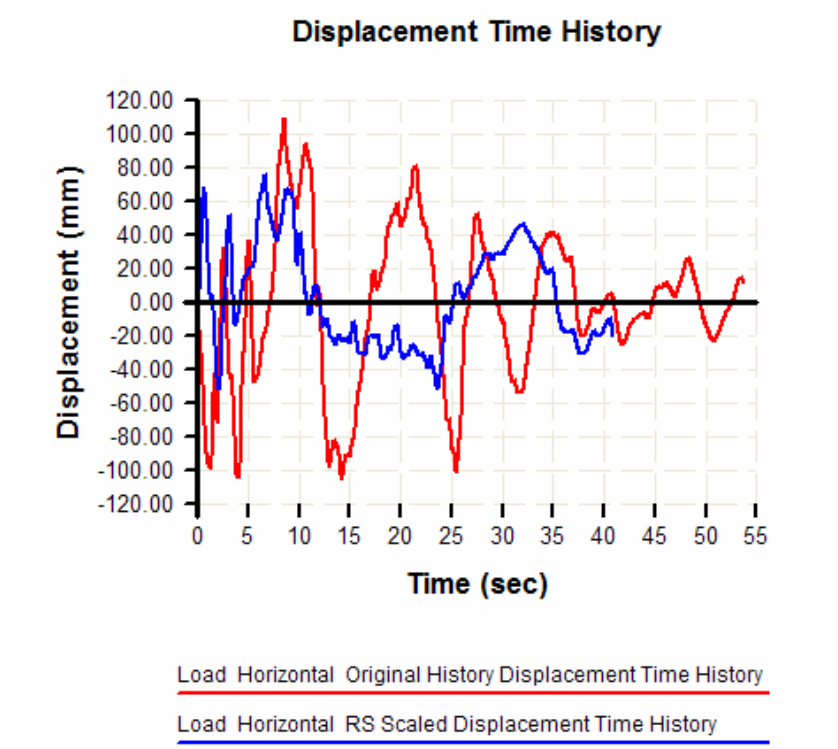


There are a variety of items that can be checked from this form.

The response spectrum from the original NS EL Centrol time history can also be added to the plot:

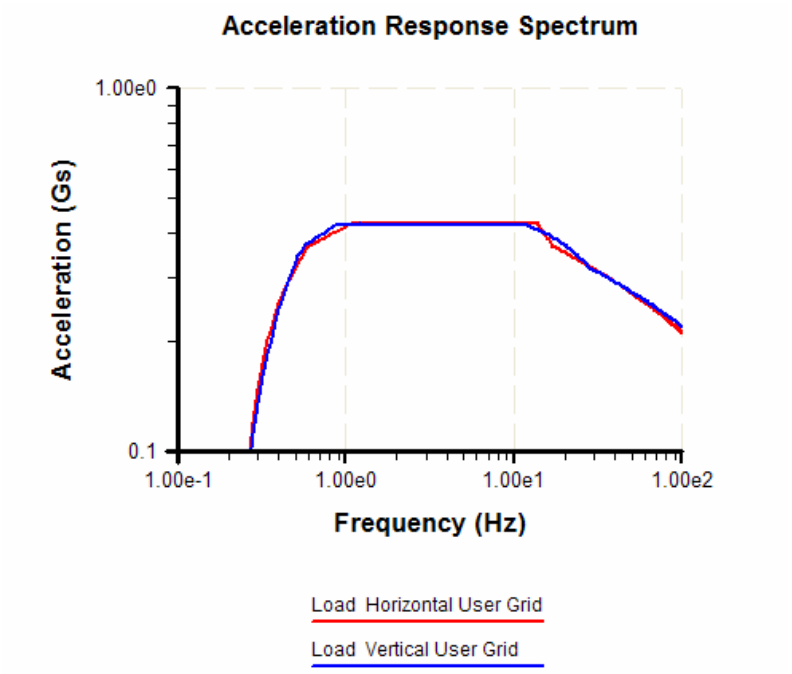


The synthesized time histories can also be displayed:

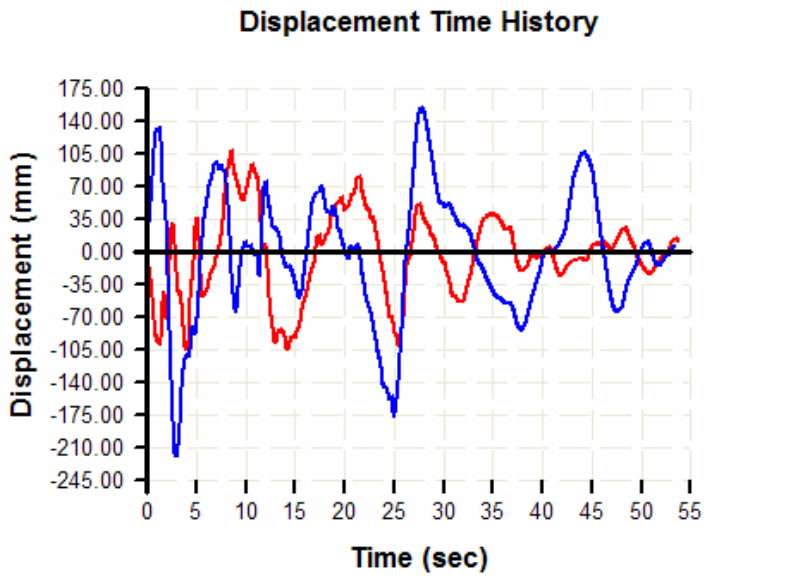


The original El Centro NS component is shown in red, and the synthesized time history to comply with the client's data is shown in blue.

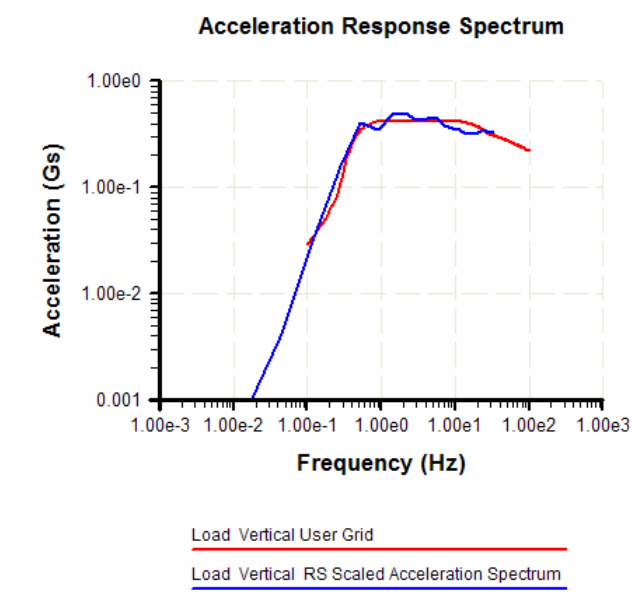
The vertical response spectrum should be processed in a similar manner. The vertical and horizontal spectra provided for the example are very close as can be seen by plotting them together.



Since the response spectra are used as scales for time histories, the time histories must be statistically independent. For this reason the NS Elcentro time history should NOT be used for both the vertical and horizontal components. The EW Elcentro time history may be similar to the NS component, but will be statistically independent. The NS and EW EL Centro time histories are shown below.

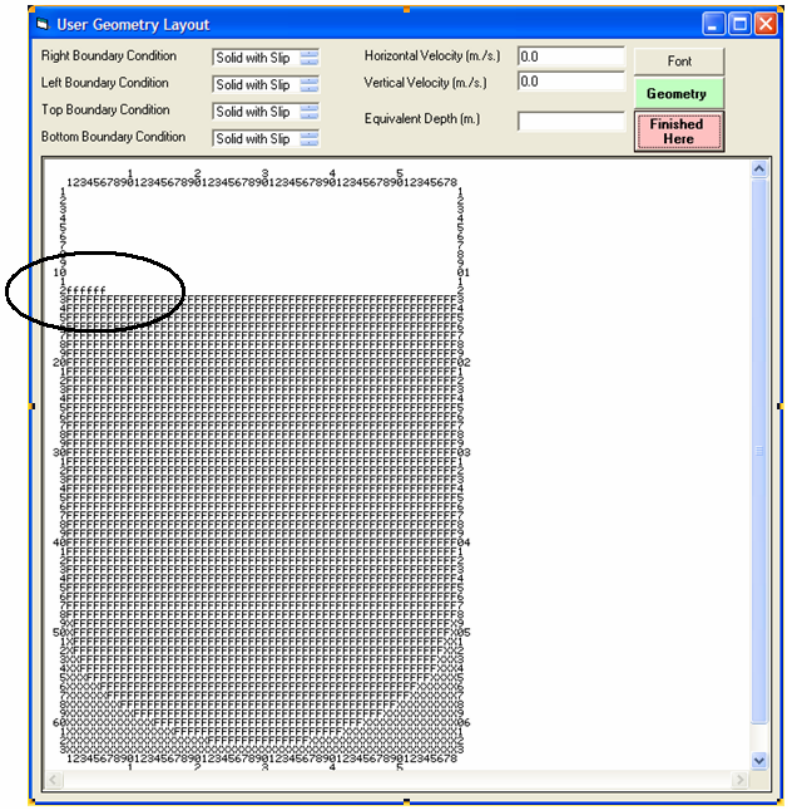


The EW component scaled to the design vertical response spectrum yields the following response spectrum correlation.



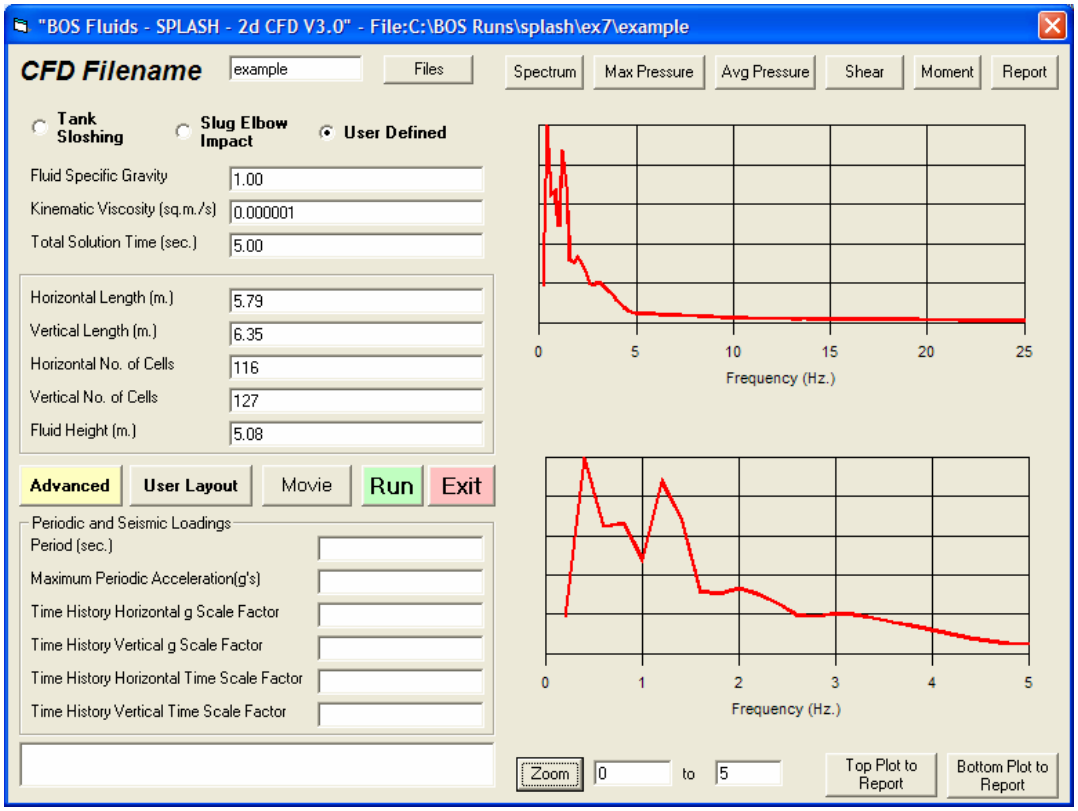
Through inspection of the waveform shapes, statistically independent, but very suitable time histories are now developed for the Splash analysis.

As a first analysis it is desired to know principle sloshing acoustic natural frequency. A slight elevation change in the liquid level is provided at the edge of the tank as shown in the figure below.

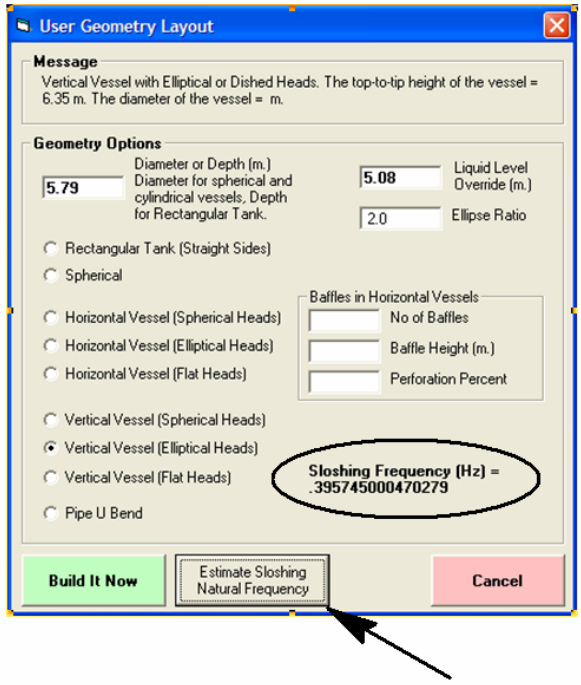


This model with the elevation change is run without any loading (note how the time history scale factors are blank or zero). The elevation fluid discontinuity will produce a small perturbation that will stimulate the sloshing natural frequency of the vessel.

The default time result is shown below:

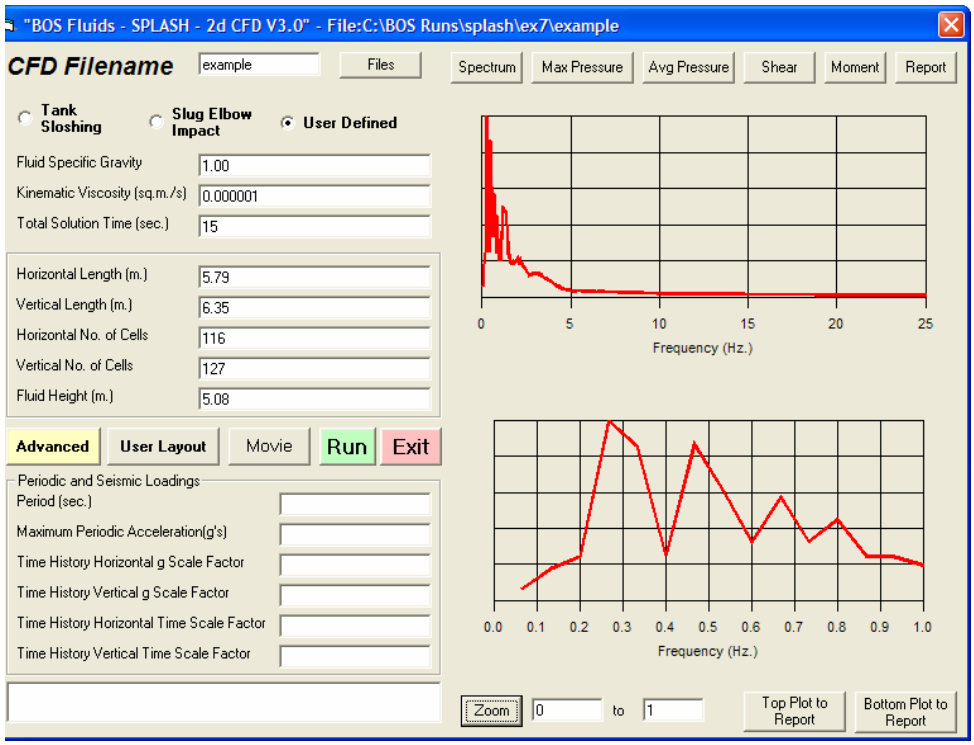


From this and the zoomed frequency it appears that the first mode is at approximately 0.5 Hz. and the second mode is at about 1.25 hz. Splash defaults to a run time of 5 seconds. To get better low frequency response the duration of the run must be extended. A 5 second run time could roughly identify a sloshing mode with a period of 5/2 = 2.5 sec, or a frequency of 1 / 2.5 = 0.4 Hz. The sloshing natural frequency estimator on the User Geometry Screen can be used to get an estimate of the frequency for comparison:

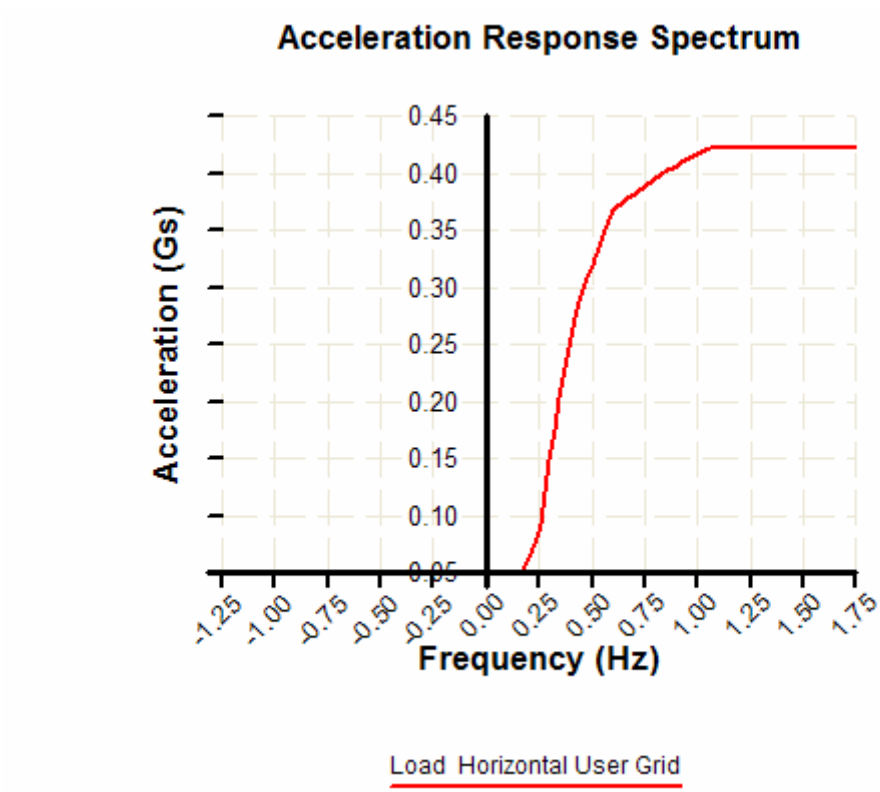


As can be seen, the comparison is good. Increasing the time should not show any additional low modes, but should improve the clarity of the modes already calculated. Increasing the solution time to 15 seconds should permit frequency estimates to periods of 7.5 s. = (1/7.5) = 0.133 Hz.

The result from this calculation is shown below. From the extended time solution it can be seen that the first mode is at around 0.3 Hz and that there is very little sloshing response outside of 3 Hz.



This is significant when the response spectrum is inspected. A section of the response spectrum plotted using a linear scale is shown below.



As can be seen, the sloshing frequency range of interest occurs at a point in the response curve where the maximum amplitude is changing significantly.

The synthesized time history waveform used for the excitation in this model, (and shown above) have excitation for at least 50 s.

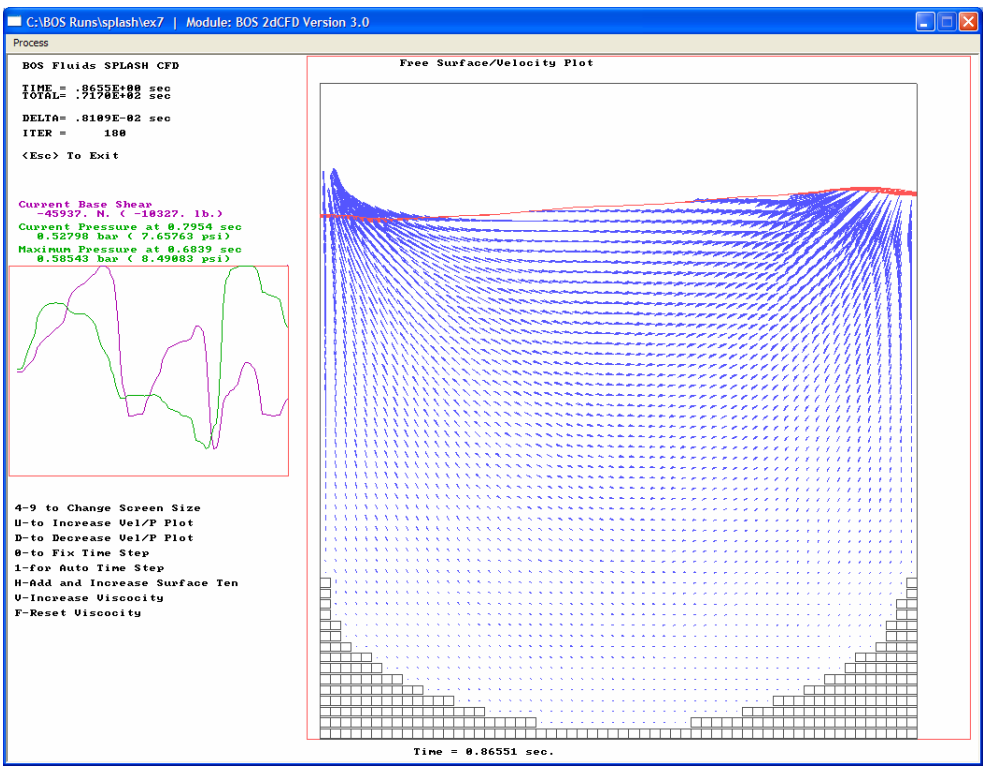
The total time used for the spectrum solution will include 55 s. for the time waveform, and then five periods after the excitation has ended. The fundamental sloshing period is:

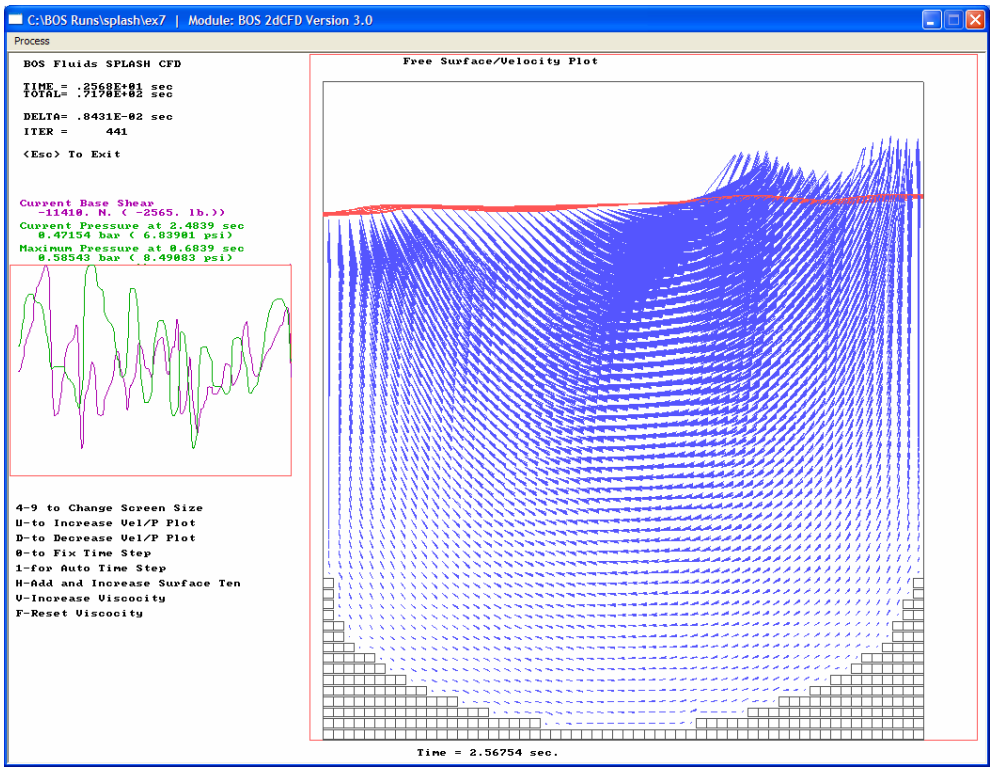
$$P = 1/0.3 = 3.3 \text{ s.}$$

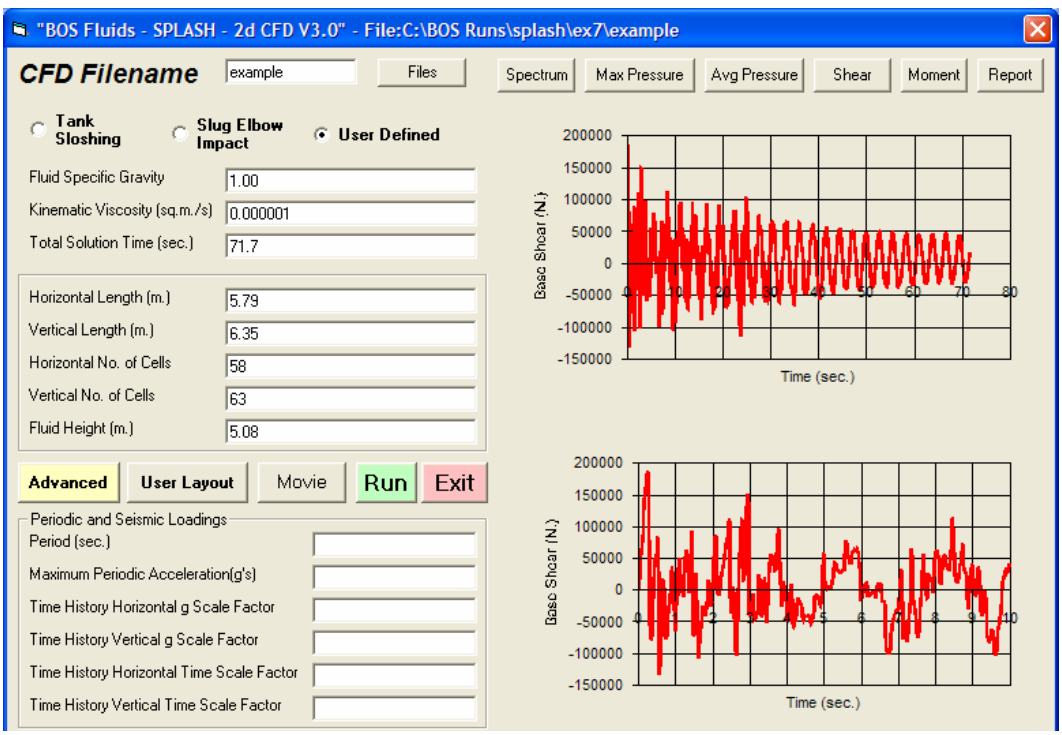
The total time of solution will be:

$$55 + (5)(3.3) = 71.7$$

Several plots during the earthquake simulation are shown below:



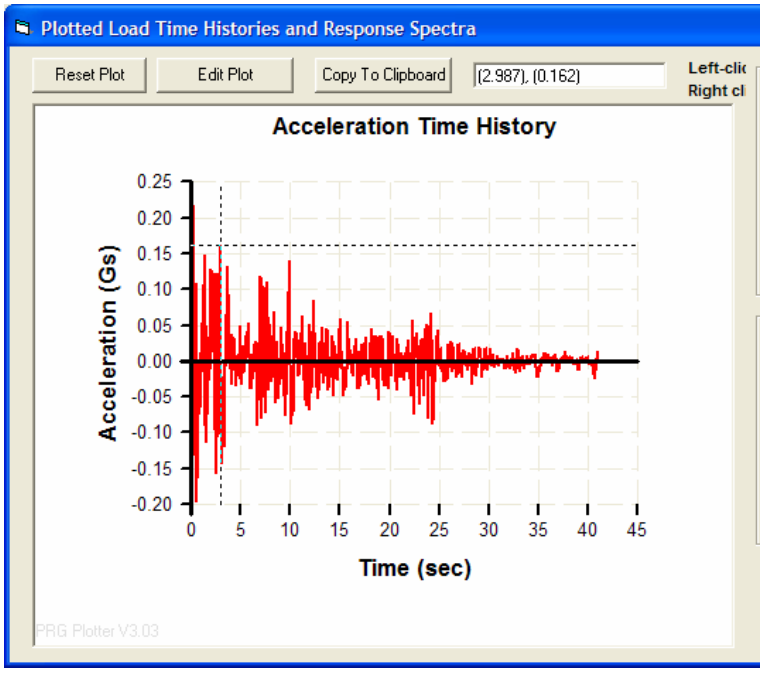




The maximum base shear is shown above and can be read from the “Splash” report as:

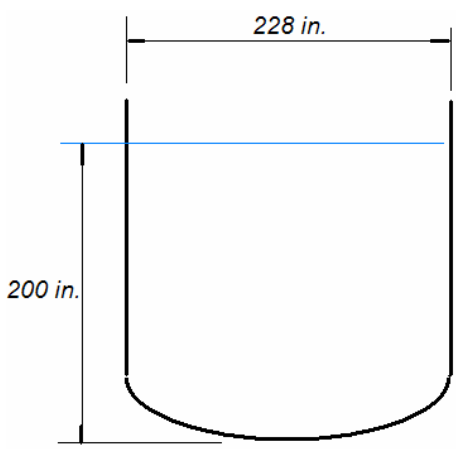
Max Shear Force (N.) 185662.513

The maximum time history from the acceleration can be read from the 2D plot:



and is 0.16 g.

The maximum weight of liquid in the container:



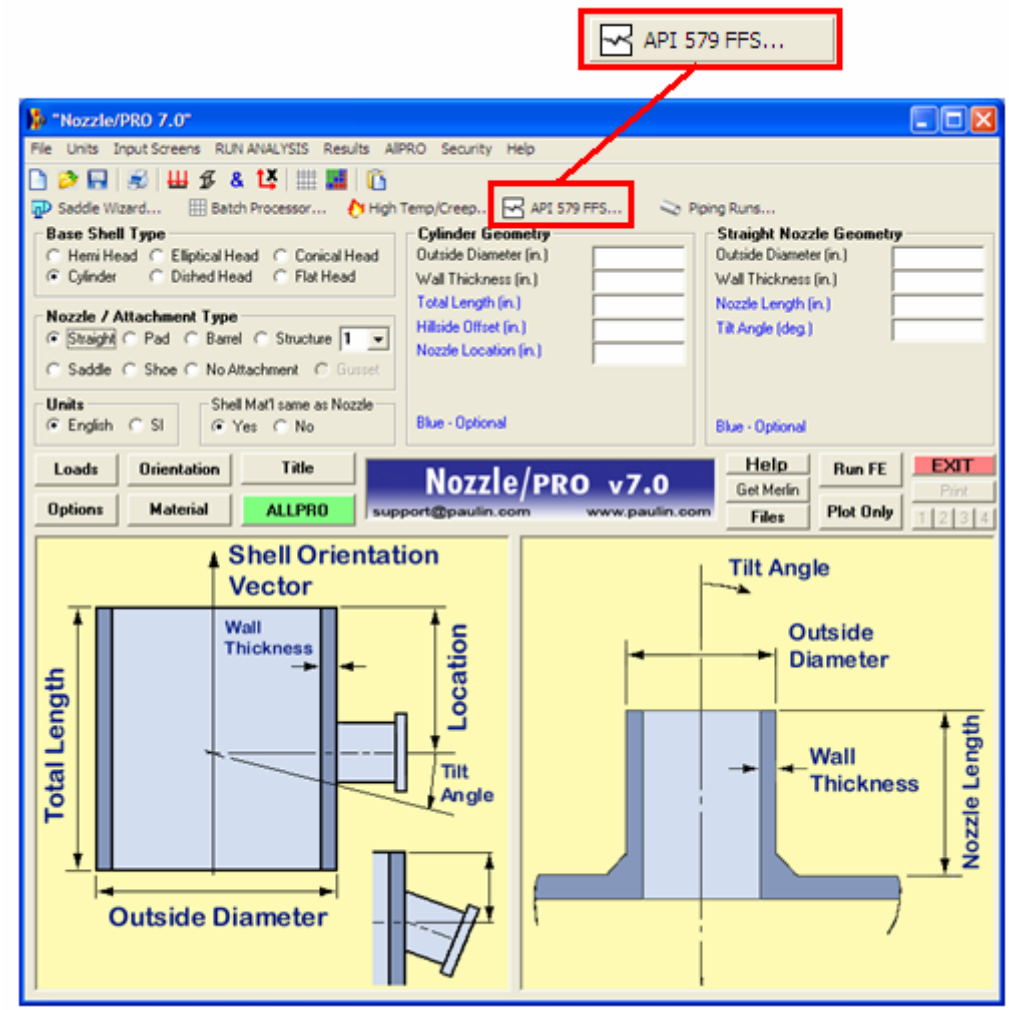
can be found as: 1,186,900 N. This weight, multiplied by the maximum time history acceleration gives the maximum load that would be applied if the liquid was a solid mass (impulsive load component only.)

Impulsive Load = (0.16)(1,186,900) = 189,906 N.

There is no reduction or amplification of the liquid load due to the magnitude of the response spectra in the sloshing frequency range. When being subject to the design response spectrum given, the liquid in the vessel could be modeled as a rigid body without any over-or-under conservatism.

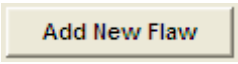
Section 9: Nozzle/PRO Fitness for Service

Local thin areas and crack like flaws may be evaluated for most Nozzle/PRO geometries using the Nozzle/PRO fitness for service input form. Fitness for service evaluations are conducted using API 579 methodologies for Level 2 & Level 3 checks. Up to ten flaws may be defined for each model. To access the fitness for service options, click the "API 579 FFS" icon as shown below:



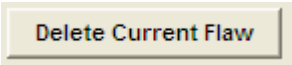
Nozzle/PRO Fitness for Service

Add New Flaw



Additional flaws to be evaluated may be defined using the "Add New Flaw" button. Each time the "Add New Flaw" button is clicked, a new flaw will be added to the input form. Up to ten flaws may be defined.

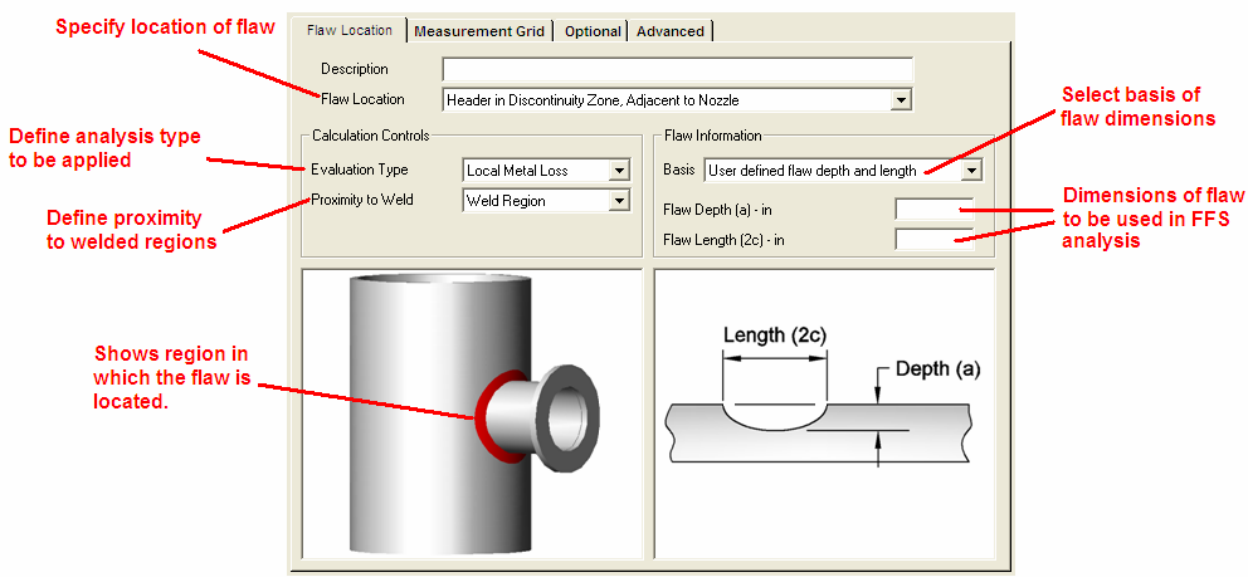
Delete Current Flaw



User defined flaws may be deleted from the input form using the "Delete Current Flaw" button. To delete a flaw, first select the desired flaw from the flaw tabs and then click the "Delete Current Flaw" button.

Flaw Location Input Sheet

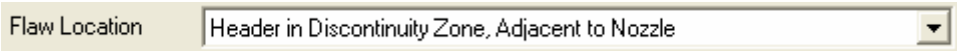
The Flaw Location input sheet provides input to define the location, type, and geometry of the flaw.



Description

The user may provide a descriptive name for each flaw.

Flaw Location



The flaw location input is used to define the general location of the flaw on the model. Options are available for each region within the parent and attachment. The user should select the general region or area in which the flaw is located. Nozzle/PRO will use the maximum stress within the specified region and

evaluate the flaw. The region in which the flaw will be located is highlighted in red within the images in the the lower left panel of the Fitness for Service screen.

Evaluation Type

Evaluation Type | Local Metal Loss

The Evaluation Type input is used to specify the type of fitness for service evaluation desired. Available analysis types include local metal loss and crack like flaw evaluations. The user may also evaluate the defined flaw as both a local metal loss and crack like flaw to determine the worst case scenario.

Proximity to Weld

Proximity to Weld | Weld Region

The Proximity to Weld option provides the ability to specify the location of the flaw in relation to welds. The default selection is "Weld Region" and should provide a conservative evaluation. If the proximity to a weld is unknown, the user should consider using the "Weld Region" option.

This flaw locator is not used for local thin areas, but is used for crack-like flaw evaluation. The effect of welds in local thin areas is included in the evaluation by the specification of the weld joint efficiency. For joint efficiencies of 1, the fact that the local thin area is in a weld has no effect.

Basis

Basis

Each flaw must have a "Basis" defined so that the dimensions of the flaw may be determined. Here, the "Basis" input defines what procedure or input will be used to establish the dimensions of the flaw. Several options are available including:

1. **Assume a default flaw size** – *Nozzle/PRO will assume that the flaw has a depth of 0.25 times the thickness of the material and a length equal to six times the thickness.*
2. **User defined flaw depth and length** – *the user must define the depth and length of the flaw which will then be used in the fitness for service evaluation.*
3. **Maximum measured flaw depth/length** – *this option should be used when a thickness measurement survey is available. In this case, the user will input the thickness survey data within the "Measurement Grid" sheet. Nozzle/PRO will determine the maximum depth and length based on the critical flaw depth and lengths using API 579 procedures. The maximum depth and length will be used irrespective of whether they are defined for the circumferential or longitudinal directions.*

Flaw Depth

Flaw Depth (a) - in | 0

Defines the maximum depth of the flaw. Only used when the flaw basis is "User Defined Flaw Depth and Length" Note that this input will be automatically generated when the flaw basis is the "Maximum measured flaw depth/length".

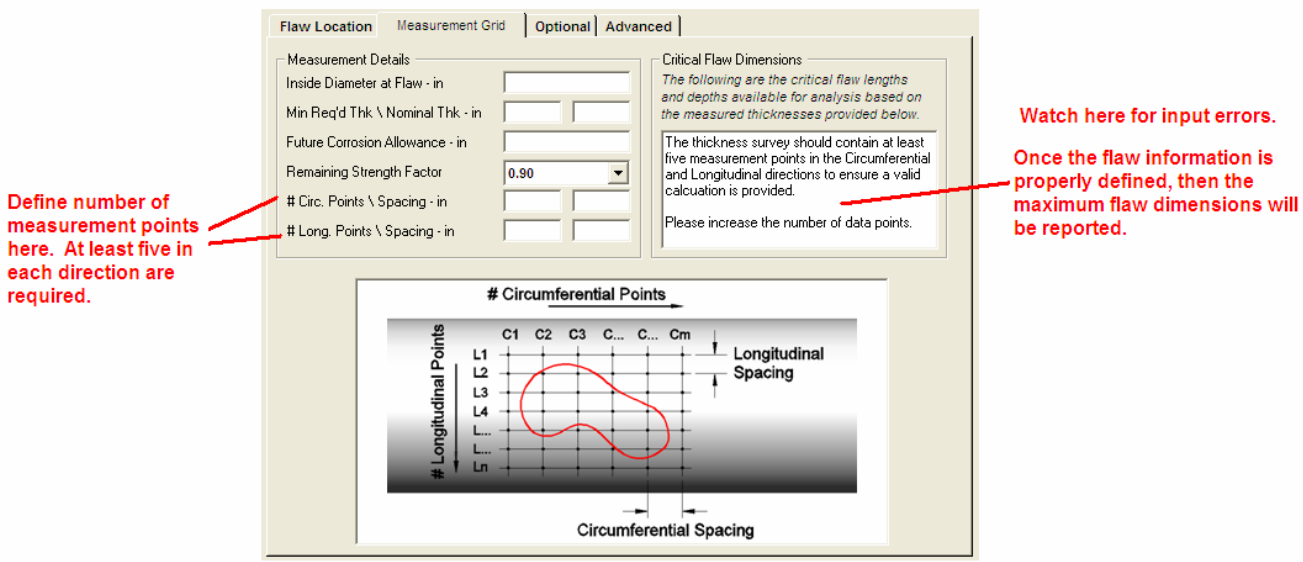
Flaw Length

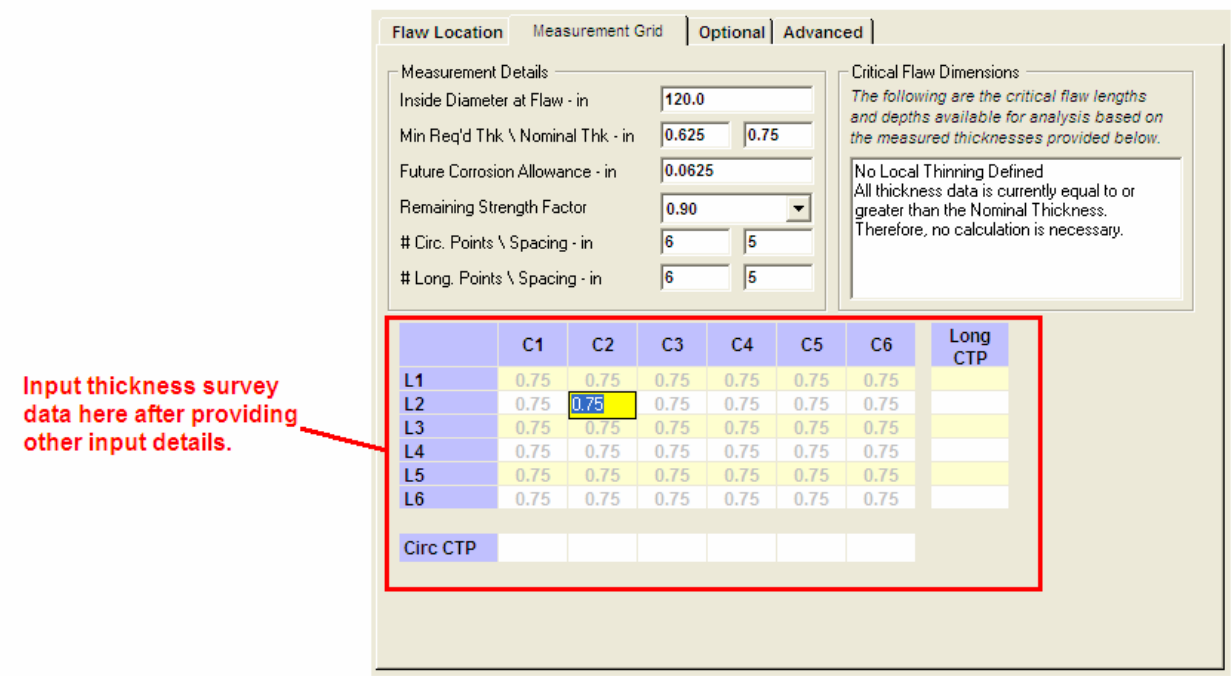
Flaw Length (2c) - in | 0

Defines the maximum length of the flaw. Only used when the flaw basis is "User Defined Flaw Depth and Length". Note that this input will be automatically generated when the flaw basis is the "Maximum measured flaw depth/length".

Measurement Grid

The Measurement Grid input sheet is used to define the characteristics of local thin areas using an array of thickness measurements which encompass the flaw. Note that the Measurement Grid input sheet is only used when the flaw basis has been specified as "Maximum measured flaw depth/length" (see Flaw Location input discussion for more details).





Inside Diameter at Flaw

Inside Diameter at Flaw - in

The inside diameter of the shell at the location of the flaw. The inside diameter is used to calculate the length over which thickness averaging will be conducted.

Min Req'd Thk \ Nominal Thk

Min Req'd Thk \ Nominal Thk - in

The minimum required thickness (left input box) and the nominal thickness (right input box) at the flawed location.

Future Corrosion Allowance

Future Corrosion Allowance - in

Defines the future corrosion allowance for the flawed location. The future corrosion allowance is used in conjunction with the minimum required thickness and thickness survey data to determine maximum flaw dimensions. The future corrosion allowance should be based on operating experience, inspection data, and corrosion rate estimates. The future corrosion allowance is applicable to the expected future operating period.

Remaining Strength Factor

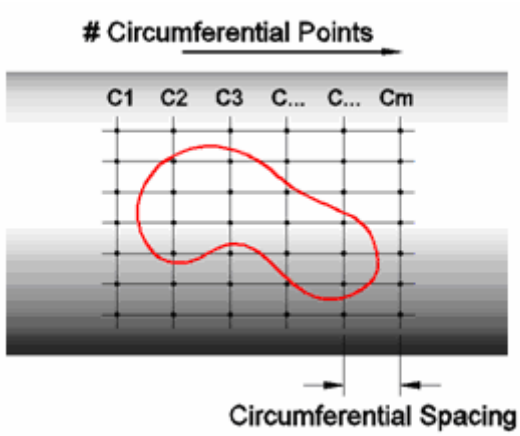
Remaining Strength Factor 0.90

The Remaining Strength Factor is the ratio of the strength of the damaged component to the undamaged component. Here, "strength" relates to the resistance against a limit type load to cause collapse or catastrophic failure. *API 579 recommends an RSF factor of 0.90 for equipment in process services*. Essentially, the remaining strength factor relates the strength of the damaged component to that of the undamaged component. Therefore, lower RSF values indicate that the analyst is willing to permit the equipment to operate a much lower strength than originally intended. Higher values are more stringent in that the flaws must be of a less significant nature and thus produce strengths closer to the undamaged state.

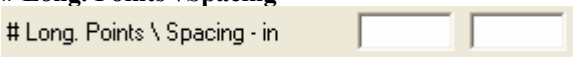
 # Circ. Points \ Spacing

Circ. Points \ Spacing - in

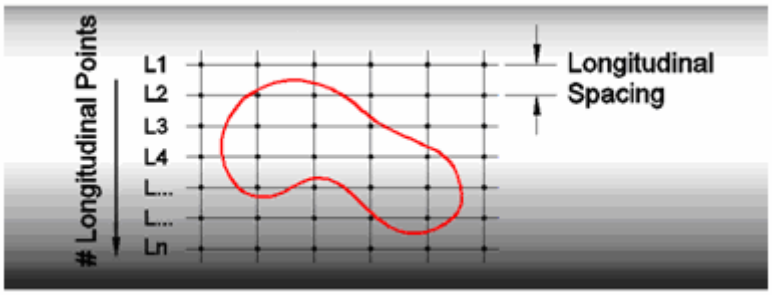
Defines the number of circumferential measurement planes along the longitudinal direction of the shell (left input box). The spacing distance along the longitudinal axis, between the circumferential planes, is defined in the right input box. See image below for details. As recommended by API 579, a minimum of five measurement points must be provided in both the longitudinal and circumferential directions. Users should refer to API 579 for the proper spacing of the measurement points.



 # Long. Points \ Spacing



Defines the number of longitudinal measurement planes in the circumferential direction of the shell (left input box). The spacing distance along the circumferential direction, between the longitudinal planes, is defined in the right input box. See image below for details. As recommended by API 579, a minimum of five measurement points must be provided in both the longitudinal and circumferential directions. Users should refer to API 579 for the proper spacing of the measurement points.



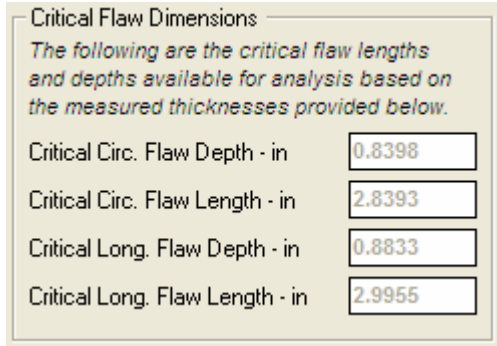
Thickness Measurement Input Spreadsheet

After the measurement details described above have been provided, the thickness measurement spreadsheet should appear. The thickness measurement spreadsheet provides input spaces for all the circumferential and longitudinal thickness measurements. The user should enter the data from their thickness survey in the columns and rows indicated by C1, C2, etc and L1, L2, etc. The column titled "Long CTP" and row titled "Circ CTP" summarize the critical thickness profiles in the longitudinal and circumferential directions, respectively. The minimum measured thickness along the critical thickness profiles is highlight in red font. User defined values which coincide with the nominal thickness are shown in light gray font while values less than the nominal thickness are shown in black font.

	C1	C2	C3	C4	C5	C6	Circ CTP
L1	1.25	1.25	1.25	1.25	1.25	1.25	1.25
L2	1.25	1	1	.9	1.25	1.25	0.9
L3	1.25	1	.9	.95	1.25	1.25	0.9
L4	1.25	.95	.85	.9	1.25	1.25	0.85
L5	1.25	1.25	1.25	1.25	1.25	1.25	1.25
L6	1.25	1.25	1.25	1.25	1.25	1.25	1.25
Long CTP	1.25	0.95	0.85	0.9	1.25	1.25	

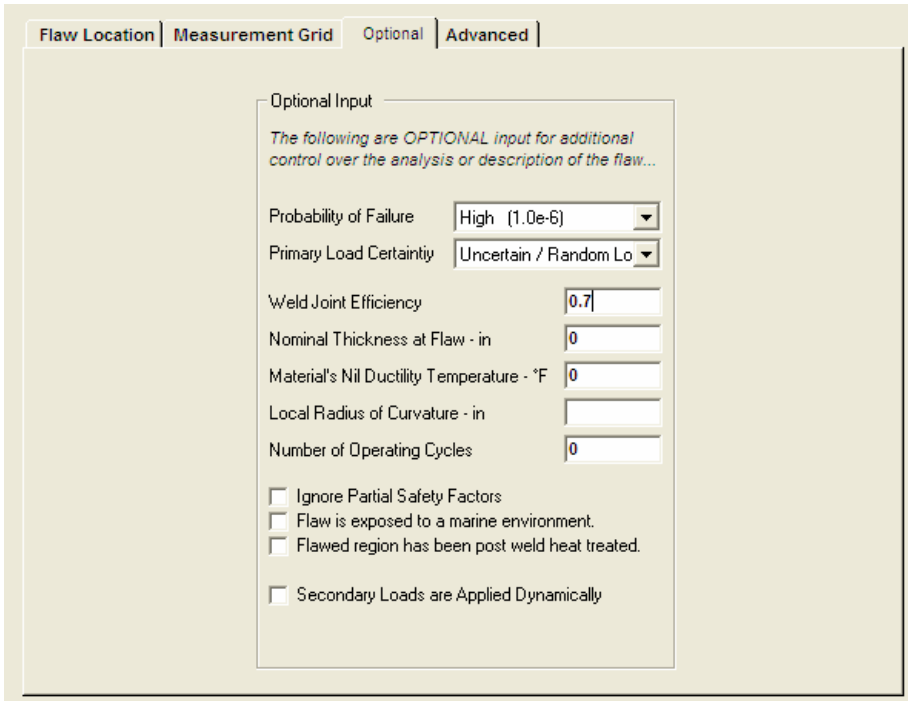
Critical Flaw Dimensions

Once all values have been specified in the thickness measurement spreadsheet, the critical flaw dimensions calculated per API 579 will be available. Maximum flaw dimensions are provided for both the circumferential and longitudinal directions as shown below. These are output results for the API 579 flaw size calculations and may not be modified by the user. Note that these output results are only visible once the user has completed all of the input fields in the "Measurement Details" frame and the thickness measurement spreadsheet is visible.

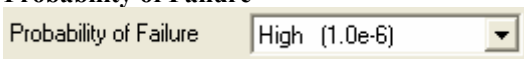


Optional Input

The optional input page provides additional control over the fitness for service evaluation. A description of these input fields is provided below.



Probability of Failure



Defines the probability of failure for the user defined flaw. This input is only used for the analysis of crack like flaws. The probability of failure is used in conjunction with the "primary load certainty" input to determine the partial safety factors applied to the primary and secondary stresses (and other user defined flaw variables). **Higher probability of failure values will result in greater partial safety factors, effectively increasing the design margins in the fitness for service evaluation and providing for a "safer" design.**

- High is the most conservative assumption (lowest risk of failure).
- Medium is slightly less conservative (moderate risk of failure).
- Low represents the lowest margin against the mean failure curve (higher risk of failure)

The default value of HIGH equates to a margin of 4 standard deviations below the mean failure curve. Medium represents approximately three standard deviations below the mean failure curve. Low represents approximately two standard deviations below the mean failure curve.

Primary Load Certainty

Probability of Failure | High (1.0e-6)

The primary load certainty relates to the coefficient of variation (COV) related to the uncertainty in the primary stress distribution. This input is only used for crack like flaw evaluations. Three options are available for the primary load certainty input:

1. **Well Known** – *primary loads and stresses at the flawed zone are computed or measured and are well known. This option corresponds to a COV = 0.10.*
2. **Reasonably Known** – *primary loads and stresses in the flawed zone are computed or measured and are reasonably well known. Here, the uncertainty is due to the possible variation in the loading or calculation methods.*
3. **Uncertain / Random Loadings** – *calculated or measured primary loads and stresses are significantly uncertain. The uncertainty is a result of the unknown or random nature of the applied loading or estimates made in the calculation of the primary stresses.*

Weld Joint Efficiency

Weld Joint Efficiency | 0.7

Defines the joint efficiency of the welded joint. When a crack like flaw is located at a welded region, the primary stresses will be divided by the weld joint efficiency. The weld joint efficiency typically ranges between 0.65 and 1.0.

Nominal Thickness at Flaw

Nominal Thickness at Flaw - in | 0

Typically, the nominal thickness at the flaw is determined by the thickness at the flaw as defined in the finite element model. However, the user may override the finite element model's thickness such that the fitness for service calculations use the options "Nominal Thickness at Flaw" value. Note that this thickness will not modify the local thickness in the finite element model, it is only used in the fitness for service post processing calculations.

Material's Nil Ductility Temperature

Material's Nil Ductility Temperature - °F | 0

The nil ductility temperature is only used for crack like flaw evaluations. Defines the nil ductility temperature for the material of construction in which the flaw is located. Used as the "reference temperature", and defined as the temperature corresponding to a Charpy impact value of 15 ft-lb for carbon steels and 20 ft-lb for Cr-Mo steels. The Nil Ductility Temperature is not used for stainless steels.

Note that if the nil ductility is actually zero degrees, then a value near zero but not exactly zero should be specified (for instance 0.1). A value of zero will not initialize the input and results in a default nil ductility value for the materials.

Local Radius of Curvature

Local Radius of Curvature - in

Defines the local radius of curvature of the vessel or pipe at the location of the user defined flaw. If no input is provided, then Nozzle/PRO will use the user defined value for the parent or attachment given in the main Nozzle/PRO screen.

Number of Operating Cycles

Number of Operating Cycles 0

Defines the estimated number of operating cycles for which the flaw will be exposed to during the anticipated future service life. The number of operating cycles is not the total cycles for the equipment. Instead, it is only the number of future cycles to which the flaw is exposed.

Ignore Partial Safety Factors

☐ Ignore Partial Safety Factors

The partial safety factors are used to provide additional margin against failure in light of uncertainties in the loadings and calculations. On occasion, the analyst is more concerned with reducing the safety margin and estimating a more realistic margin against failure. In such cases, the user may evaluate the flaw without the use of the partial safety factors. Note that this input is only used for the evaluation of crack like flaws.

Flaw is Exposed to a Marine Environment

☐ Flaw is exposed to a marine environment.

Used for crack growth rate calculations only; does not affect the local thin area computations. Increases the crack growth rate for both stainless and carbon steels per API 579 F.5.3 by 4.4 times.

Flawed Region has been Post Weld Heat Treated

☐ Flawed region has been post weld heat treated.

This factor is only used in the evaluation of crack-like flaws and will reduce the effect of residual stress when the flaw is in the proximity of a weld. If PWHT has been performed the residual stresses in the weld are reduced to 20% of their non-PWHT values.

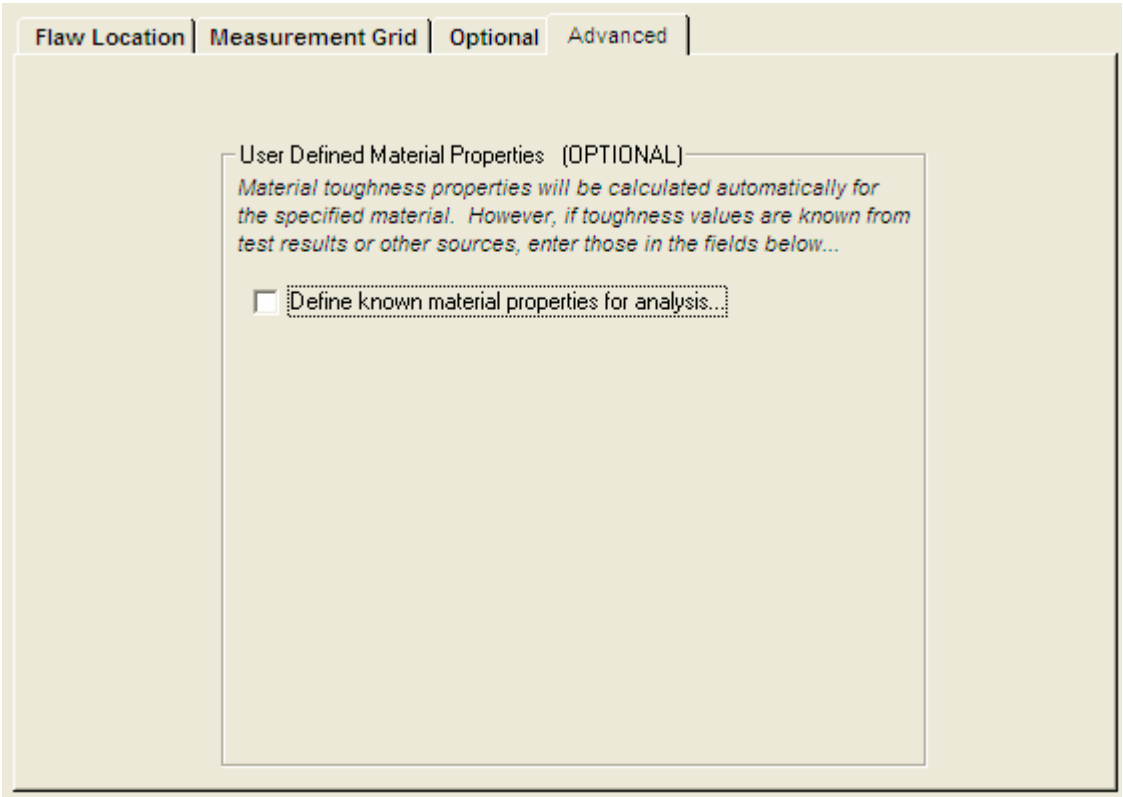
Secondary Loads are Applied Dynamically

☐ Secondary Loads are Applied Dynamically

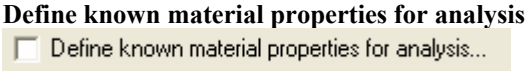
Only used for the evaluation of crack like flaws. Used when some portion of the operating load or secondary stress is applied dynamically. In this case the K_{IC} value will be adjusted based on the temperature, and the Dynamic Ramp loading time. The user can override this calculation by entering the Dynamic Critical fracture toughness at operating temperature if a better value is available.

Advanced Input

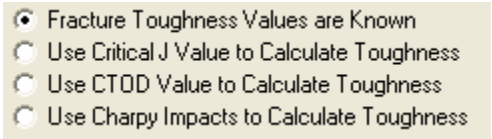
The advanced input tab provides additional control over the material properties to be used in the FFS calculations. Mainly, control over the material properties relating to the evaluation of crack like flaws is provided.



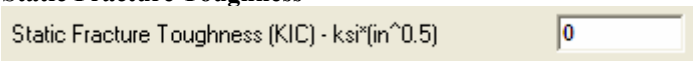
Define known material properties for analysis



If material properties for the material of construction are known, then select this option to access the material definition options. Nozzle/PRO provides the following options for material definitions:

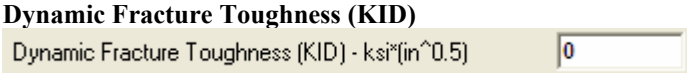


Static Fracture Toughness



Available only if the option "Fracture Toughness Values are Known" is selected.
K_{IC} value at operating conditions. This value will be estimated by the program based on the type of material input if not entered. Only used for crack type flaws.

Dynamic Fracture Toughness (KID)



Available only if the option "Fracture Toughness Values are Known" is selected.
K_{IC} value at operating conditions for dynamic loadings. Only used if the "Secondary Loads are Applied Dynamically" check box is marked in the Optional input tab. If not entered the program will calculate a dynamic K_{IC} based on loading time and temperature. Only used for crack-type flaws.

Critical J Value

Critical J Value - ksi-in 0

Available only if the option "Use Critical J Value to Calculate Toughness" has been selected.
If a J integral value is entered it will be used to compute the K_{IC} per API 579 Appendix F.4.2. Only used for crack-type flaws.

CTOD Value from Test

CTOD Value from Test - in 0

Available only if the option "Use CTOD Value to Calculate Toughness" has been selected.
If a crack tip opening displacement value is available from a CTOD test of the material then this value may be entered as per API 579 Appendix F.4.2.

Charpy Impact Energy

Charpy Impact Energy - ft-lbs 0

Available only if the option 'Use Charpy Impacts to Calculate Toughness" has been selected.
Enter the Charpy energy at operating temperature if available. This value can be converted into the K_{IC} value to be used in crack-type flaw evaluations.

Fitness for Service Example

The following example illustrates the API 579 fitness for service tool in Nozzle/PRO. This particular example is API 579 example 4.11.1 and will illustrate the evaluation of a localized corrosion region in a pressure vessel, removed from any gross structural discontinuities, but located at a longitudinal weld seam.

This sample model may be found in the "Samples1" folder of the installation directory with filename: "NozzlePRO_FFS_LTA.nozzlepro"

A region of localized corrosion has been found on a pressure vessel during a scheduled turnaround. The local metal loss area passes through a longitudinal weld seam.

Design Conditions = 300 psi @ 350F
Inside Diameter = 48 in.
Fabricated Thickness = 0.75 in.
Uniform Metal Loss = 0.0 in.
FCA = 0.1 in. (Future Metal Loss [Corrosion Allowance])
Material = SA 516 Grade 70
Weld Joint Efficiency = 0.85

The corroded area was NOT in the circumferential weld seam.
Effective Longitudinal Flaw Length = 9.75 in.
Effective Circumferential Flaw Length = 9.0 in.
Minimum thickness = 0.45 in.

Thickness Measurements for the local metal loss are given below. There are 8 measurement points in the longitudinal (C) direction, and 7 measurement points in the circumferential (M) direction.

	C1	C2	C3	C4	C5	C6	C7	C8	Circ CTP
M1	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75
M2	0.75	0.48	0.52	0.57	0.56	0.58	0.60	0.75	0.48
M3	0.75	0.57	0.59	0.55	0.59	0.60	0.66	0.75	0.55
M4	0.75	0.61	0.47	0.58	0.36	0.58	0.64	0.75	0.36
M5	0.75	0.62	0.59	0.58	0.57	0.48	0.62	0.75	0.48
M6	0.75	0.57	0.59	0.61	0.57	0.56	0.49	0.75	0.49
M7	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75
Long CTP	0.75	0.48	0.47	0.55	0.36	0.48	0.49	0.75	

Step 1	Specify geometry for the pressure vessel. Note that the option "No Attachment" has been selected since the flaw is located in the shell removed from any gross structural discontinuities. **Base Shell Type**: Hemi Head, Elliptical Head, Conical Head, Cylinder, Dished Head, Flat Head **Nozzle / Attachment Type**: Straight, Pad, Barrel, Structure 1, Saddle, Shoe, No Attachment, Gusset **Cylinder Geometry**: Outside Diameter (in.) 49.5; Wall Thickness (in.) 0.75; Total Length (in.) 400
Step 2	Click "Loads" to define the loadings acting on the pressure vessel. Input the values given below. When finished, click OK to return to the main Nozzle/PRO interface. Pressure (psi) 300 Nozzle loads may be omitted and Nozzle Pro will still calculate stiffnesses, stress intensification factors and allowable loads. Pressure SHOULD BE ENTERED. Nozzle Inside Temperature (deg.F) 350 Shell Inside Temperature (deg.F) 350 Nozzle Outside Temperature (deg.F) 350 Shell Outside Temperature (deg.F) 350
Step 3	Click "Materials" to define the material of construction for the pressure vessel shell. Once finished, click OK to return to the main Nozzle/PRO interface screen. Users with a Mat/PRO license may use the "Import Material from Mat/PRO" button to import all of the material properties. Otherwise, simply input the values given below: Cold Allowable Stress (psi) 20000 Hot Allowable Stress (psi) 20000 Fatigue Curve Low Carbon Steel Elastic Modulus (psi) 29.400e6 Poissons Ratio 0.3 Expansion Coefficient (in./in./deg.F) 7.0000e-6 Cold Yield Stress (psi) 38000 Hot Yield Stress (psi) 33050 Cold Tensile Stress (psi) 70000
Step 4	Open the API 579 FFS definition screen by clicking the API 579 FFS icon: API 579 FFS...

Step 5	In the API 579 FFS input form, define a description and select the Flaw Location. Note that only one option is available in the Flaw Location list. This is because the Nozzle/PRO attachment type was "No Attachment". Description: API 579 Example 4.11.1 (Corrosion at longitudinal weld seam) Flaw Location: Anywhere in Shell
Step 6	Since the flaw to be evaluated is a local corroded region, select the option "Local Metal Loss" from the evaluation type input field. Evaluation Type: Local Metal Loss
Step 7	The flaw is located in a welded region, therefore the option "Weld Region" must be selected for the Proximity to Weld input field. Proximity to Weld: Weld Region
Step 8	Define the flaw definition basis. This input instructs Nozzle/PRO how the flaw dimensions will be provided. In this example, a thickness survey has been provided and will be used in the flaw evaluation. Therefore, the option "Maximum measured flaw depth/length" should be selected. When "Maximum measured flaw depth/length" is selected, Nozzle/PRO will expect the user to provide a thickness survey input in the Measurement Grid input tab. Nozzle/PRO will use the largest flaw dimensions in circumferential and longitudinal directions in the FFS calculations. Basis: Maximum measured flaw depth/length Flaw Depth (a) - in: 0.3867 Flaw Length (2c) - in: 8.7107 **Nozzle/PRO will calculate these values and insert them into these input fields for you.**
Step 9	Click the Measurement Grid tab to access the thickness survey input form. Measurement Grid
Step 10	In the Measurement Grid input tab, fill in the various input fields as shown below: Measurement Details Inside Diameter at Flaw - in: 48 Min Req'd Thk \ Nominal Thk - in: 0.492 \ 0.75 Future Corrosion Allowance - in: 0.1 Remaining Strength Factor: 0.90 # Circ. Points \ Spacing - in: 8 \ 1.5 # Long. Points \ Spacing - in: 7 \ 1.5

Step 11

Fill in the Thickness Survey spreadsheet with the thickness measurements provided at the beginning of this example:

Note that the critical thickness planes (CTP's) in the circumferential and longitudinal directions will be determined as the input is completed. Also, the minimum measured thickness is highlighted by red font.

	C1	C2	C3	C4	C5	C6	C7	C8	Circ CTP
L1	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75
L2	0.75	.48	.52	.57	.56	.58	.6	0.75	0.48
L3	0.75	.57	.59	.55	.59	.60	.66	0.75	0.55
L4	0.75	.61	.47	.58	.36	.58	.64	0.75	0.36
L5	0.75	.62	.59	.58	.57	.48	.62	0.75	0.48
L6	0.75	.57	.59	.61	.57	.56	.49	0.75	0.49
L7	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75	0.75
Long CTP	0.75	0.48	0.47	0.55	0.36	0.48	0.49	0.75	

Step 12

Once all of the thickness survey data is entered, the flaw size calculations should be available in the Critical Flaw Dimension results frame. The maximum of either the circumferential or longitudinal dimensions will be used in the calculation. Recall that these will be automatically inserted in the Flaw Depth and Flaw Length input fields as discussed in Step #6.

Critical Flaw Dimensions

The following are the critical flaw lengths and depths available for analysis based on the measured thicknesses provided below.

Critical Circ. Flaw Depth - in	0.3867
Critical Circ. Flaw Length - in	7.2107
Critical Long. Flaw Depth - in	0.3867
Critical Long. Flaw Length - in	8.7107

Step 14

Click the "Optional" input tab to define any option input values.

Optional

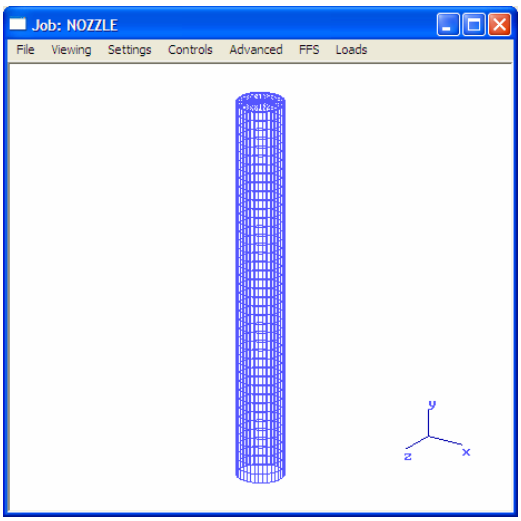
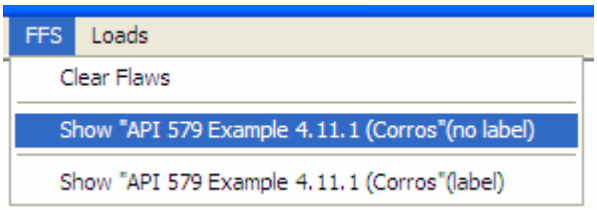
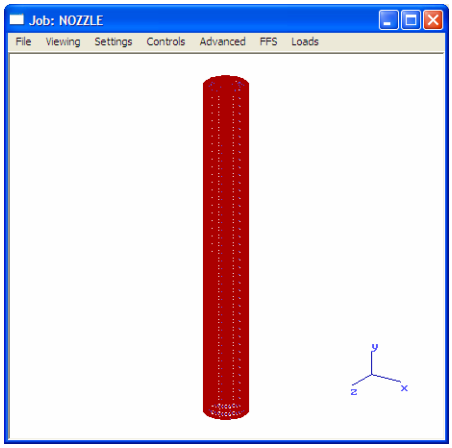
The only requirement for this example is the Weld Joint Efficiency. Nozzle/PRO defaults to a Weld Joint Efficiency value of 0.70. However, in this example the Weld Joint Efficiency is 0.85.

Weld Joint Efficiency 0.85

Step 15

Click OK to save the flaw data and return to the main Nozzle/PRO input screen.

OK

Step 16	In the main Nozzle/PRO interface screen, click "Plot Only" to generate a graphical plot of the model to be analyzed. The result should be as shown below:  
Step 17	When the graphical image appears, the portions of the model to be included in the FFS calculations can be reviewed using the "FFS" menu option. The nodes of the model to be included in the FFS calculation will be highlighted by the maroon colored dots as shown below. Close the plot window when finished viewing the model.  

Step 18	In the main Nozzle/PRO screen, click Run FE to perform the finite element analysis and FFS calculations. **Run FE**
Step 19	After the analysis has completed, the API 579 Fitness for Service calculations will be available in the output report. To access the reports, click on the following menu items in the report's table of contents: • FFS Results Summary • FFS for Flaw# 1 for Region:Cylindrical Shell Refer to the following discussion for interpretation of the analysis results.

A Pass/Fail summary of the FFS calculations is provided in the FFS Results Summary report. This report provides a quick review of the calculation results for each of the user defined flaws. In this example, the summary report tells the user that Flaw #1 (the local thin area defined in this example) is not acceptable and exceeds the allowable limits by a factor of 1.331.

```
FFS Results Summary

Flaw# 1 Region:Cylindrical Shell Primary Metal Loss: 1.331                    NOT OK
Flaw# 1 Region:Cylindrical Shell Primary MetSecondary Metal Loss: 0.000
Criteria NOT Satisfied
```

Details of the FFS calculations for Flaw #1 can be reviewed by clicking the "FFS for Flaw# 1 for Region: Cylindrical Shell" link in the report's table of contents. This report is shown below.

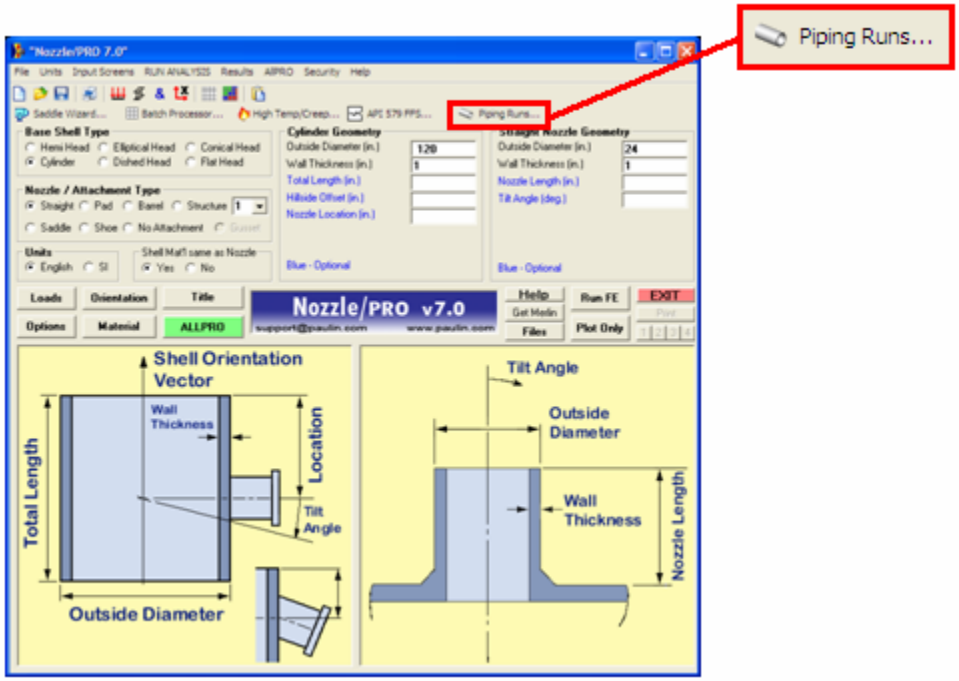
```
FFS for Flaw# 1 for Region:Cylindrical Shell

API 579 Fitness for Service Evaluation
----------------------------------------------

Conservative assumptions were made when implementing the
fitness for service rules of API579 Sections 5.0 and 9.0.
It is the users responsibility to review and check the
results printed herein to verify that they apply and are
valid for the particular problem studied.

Descr:  API 579 Example 4.11.1 (Corros

Yield Stress at Room Temperature                 =      38.000 ksi
Yield Stress at Operating Temperature            =      33.050 ksi
Flow Stress at Operating Temperature             =      43.050 ksi
Flow Stress at ROOM Temperature                  =      48.000 ksi
Modulus of Elasticity at Room Temperature        =   29400.000 ksi
Modulus of Elasticity at Operating Temperature   =   28100.000 ksi

Internal Pressure                                =     300.000 psi
Operating Temperature                            =     350.000 degF

Local Primary Membrane Stress in Area of Flaw    =      10.118 ksi
Local Primary Bending Stress in Area of Flaw     =       1.292 ksi
Local Secondary Membrane Stress in Area of Flaw  =       0.003 ksi
Local Secondary Bending  Stress in Area of Flaw  =       0.007 ksi

Initial Crack Depth                              =       0.387 in.
Initial Crack Half-Length                        =       4.355 in.
Component Wall Thickness at Flaw                 =       0.750 in.
```



```
Component Inside Radius at Flaw Location          =      24.750 in.

Flaw is in an area that contains a weld or HAZ.

Longitudinal Weld Joint Efficiency                =       0.850
Probability of Failure                            = 0.000001000

Coefficient of Variation                          =       0.100
(Primary Loads and Stresses are
 computed and well known.)

Poissons Ratio used in this analysis              =       0.300


API 579 Section 5.0 Assessment for Local Metal Loss
---------------------------------------------------

5.54 Membrane  Stress due to Primary Loads        =      32.992 ksi
5.54 Allowable Stress due to Primary Loads        =      24.788 ksi

Primary Membrane Stress at Flaw EXCEEDS limit     =     133.100 %

5.54 Membrane  Stress due to Secondary Loads      =       0.009 ksi
5.54 Allowable Stress due to Secondary Loads      =      49.575 ksi

Secondary Membrane Flaw Stress WITHIN allowable   =       0.017 %
```

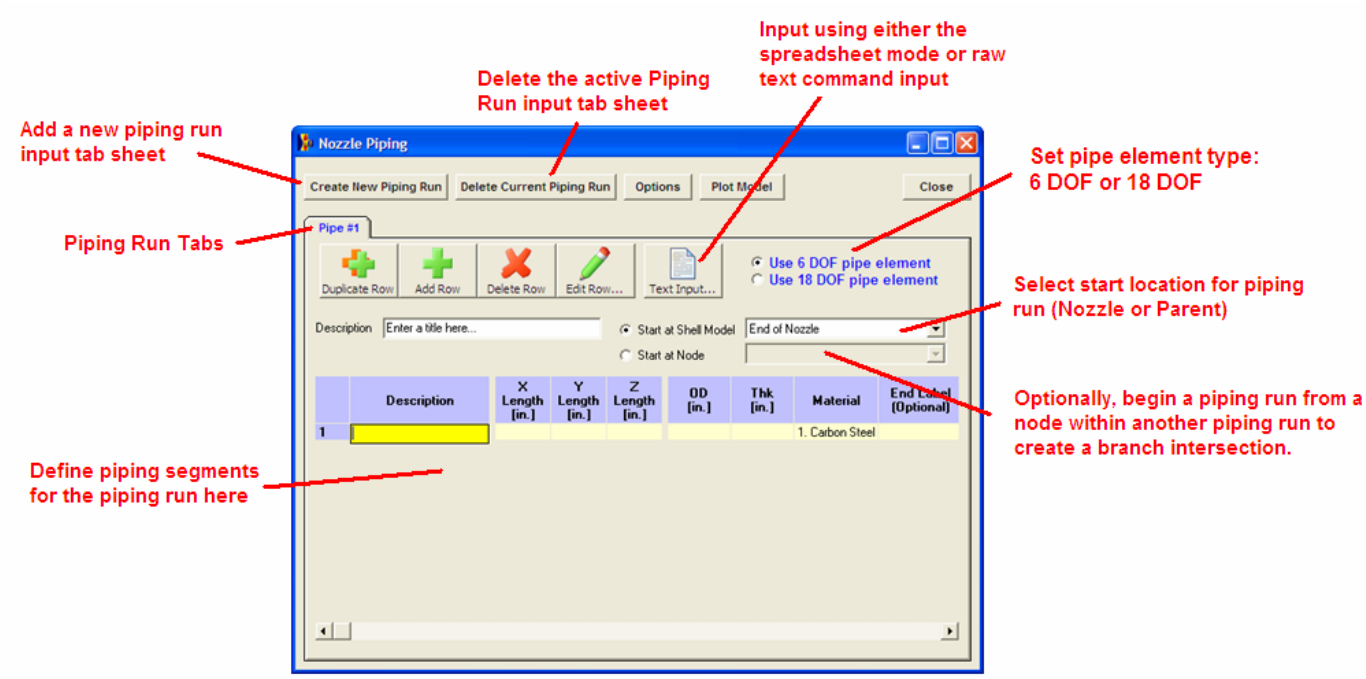
Section 10:Nozzle/PRO Piping Input Screens

Piping may be attached to Nozzle/PRO shell models using the piping input screens accessed via the "Piping Runs…" icon as shown below. Up to ten unique piping runs may be included with the Nozzle/PRO model.

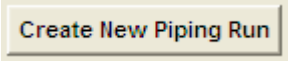


Nozzle/PRO Piping

All piping is modeled using the Nozzle/PRO Piping input form. Each of the inputs will be described in the following sections.

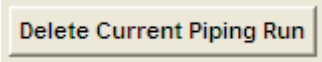


Create New Piping Run



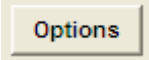
Additional piping runs are created using the "Create New Piping Run" button. Each time the "Create New Piping Run" button is clicked, a new piping run tab will be added to the input form. Up to ten unique piping runs may be defined for the model. Each piping run may contain virtually any number of piping segments (elements).

Delete Current Piping Run



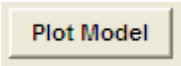
Piping runs may be deleted from the input form using the "Delete Current Piping Run" button. The piping run which will be deleted will be the active piping run currently selected in the piping tab sheet.

Options



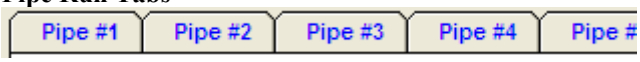
There are a number of optional settings the user may specify which will control the Nozzle/PRO piping solution and reports. These options are accessed via the "Options" button.

Plot Model



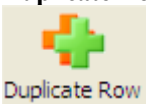
Prepares the model for analysis and generate a graphical plot of the model. The user may also click the "Plot Only" button in the main Nozzle/PRO interface screen to achieve the same results.

Pipe Run Tabs



Each tab indicates a piping run included in the Nozzle/PRO model. Access the input for the specific piping run by selecting one of the available pipe run tabs.

Duplicate Row

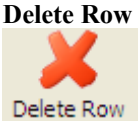


Used to duplicate an existing input row in the piping input spreadsheet.

Add Row

Add a new row to the piping input spreadsheet.

Delete Row



Delete the active row from the piping input spreadsheet.

Edit Row

Use the Edit Row button to perform various editing operations on the current row and piping input spreadsheet such as cut, copy, paste, insert, etc.

Text Input… (or Grid Input…)

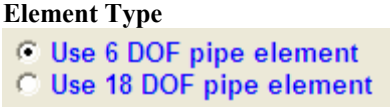
The user may construct each piping run using either the standard spreadsheet input or a raw text file with Beamer file commands. To switch into text input mode, just click the Text Input button. Nozzle/PRO can convert the existing spreadsheet input into text input, but can not convert text input back into the spreadsheet input. Therefore, any changes made in the Text Input mode can never be converted back into the Grid Input mode.

The user may switch between Text Input mode and Grid Input mode without losing either input. For instance, even when Text Input mode is being used, the original spreadsheet data will be saved so that the user can revert back at any time. However, keep in mind that the spreadsheet will not be revised with any Text Input changes.

Nozzle/PRO Piping Grid Input

The Grid Input mode is the primary input mode and recommended for most users. The Grid Input mode allows the user to construct the Nozzle/PRO piping model using a familiar spreadsheet input format. The following describes the input fields available for the Grid Input mode.

Element Type

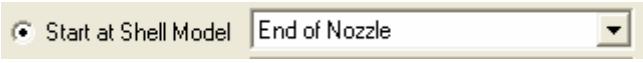


Designates the finite element type to be used in the piping model. Two element types are available, the standard 6 DOF beam elements or an advanced 18 DOF beam element. The 6 DOF beam element is widely used in piping analysis programs and will replicate traditional piping analysis results. The advanced 18 DOF piping element permits ovalization degrees of freedom and therefore produces more accurate solutions where local flexibilities are important such as in large D/t piping. The 18 DOF elements also produce more accurate stiffness interaction results for close coupled elbow-elbow pairs and intersection models.

Description

The user may provide a descriptive name for the current piping run here.

Start at Shell Model

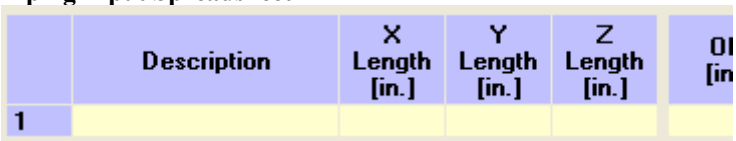


Each piping run must begin at a defined location in the model. The "Start at Shell Model" is used to begin a piping run from an attachment point on the Nozzle/PRO shell model. Options are typically at the end of the nozzle or some point on the parent geometry (ends of the header or bottom of a head, etc).

Start at Node

Piping runs may also begin from intersection locations within other piping runs. This option will typically be used when the user needs to construct branch piping intersections. The selection list here is a listing of all "End Nodes" defined in other piping runs. End nodes must be defined within other piping runs before a new piping run can be created and begin from an existing "End Node".

Piping Input Spreadsheet



The piping input spreadsheet allows the user to construct the piping model. Each row represents a segment within the piping run. For instance, a straight section of piping would be defined by its length, diameter, and other properties. The various columns within the Piping Input Spreadsheet are described below:

Input Column	Description
Description	The description field should be used to provide a unique identifier for the piping input row which will help the user when reviewing the input or results at a later date.
X Length	Describes the length of the piping segment in the global X direction. For straight sections, this is the length to the ends of the straight section or to the tangent intersection points if a bend is located at the end of the straight section.
Y Length	Describes the length of the piping segment in the global Y direction. For straight sections, this is the length to the ends of the straight section or to the tangent intersection points if a bend is located at the end of the straight section.
Z Length	Describes the length of the piping segment in the global Z direction. For straight sections, this is the length to the ends of the straight section or to the tangent intersection points if a bend is located at the end of the straight section.
OD	Describes the outside diameter of the piping segment.
Thk	Describes the design thickness of the piping segment. This thickness should be the nominal thickness of the pipe less any mechanical tolerances, corrosion allowance, or other margins.
Material	Specify the material of construction for the piping segment. See the Piping Material input screen for further description.
End Label (Optional)	• The end label is used to define a connection point for other piping runs. For instance, end node labels are typically defined where other piping runs will begin or end. These are usually where piping branch intersections occur. • The End Label may be a Base Node ID defined as part of a restraint in another portion of the piping model. If the End Label is a Base Node ID, then the two nodes will be tied together by the stiffness defined for the restraint containing the Base Node ID. • This input is optional.
Pressure	The operating pressure for the piping segment.
Temperature	The operating temperature for the piping segment.
Fluid Density	Describes the density of the contents of the piping segment. If a fluid density is provided, Nozzle/PRO will assume that the entire piping segment is filled.
Restraints	Used to apply restraints to the piping segment. See the Piping Restraints screen for more details on this feature.
End Forces	Define any point loads using this input option. The end loads will always be applied at the end of the current piping segment. For piping segments with bends at the end, the end forces are applied with the same philosophy as the restraints.
Bend at End?	If a piping bend exists at the end of the piping segment, then mark this check box. All bends are assumed to be standard bends with a bend radius equal to 1.5 times the outside diameter of the piping segment.
Bend Radius	The user may override the default bend radius of 1.5*D by specifying a user defined bend radius in this field.
Flanged Ends	If rigid components exist near the bend which may influence the flexibility of the bend, then select the number of ends affected by these rigid components.
Number Miter Cuts	For mitered bends, specify the number of miter cuts which exist along the bend.
Rigid Element?	If the current piping segment should be considered a rigid element, then mark the Rigid Element check box.
Rigid Weight	For rigid elements only, the user may define the total weight of the rigid element. If the rigid weight is specified as zero, then Nozzle/PRO will assume that a rigid construction element should be generated. If the rigid weight is greater than zero, then the total weight of the rigid will be equal to the specified rigid weight.

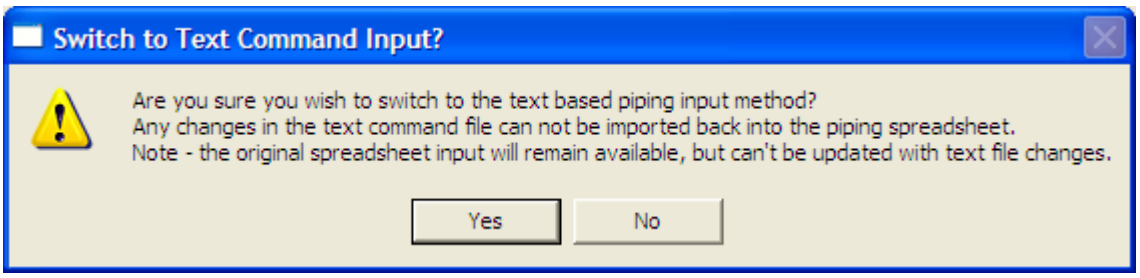
Insulation Thickness	If external insulation exists on the piping segment, define the thickness of the insulation here.
Insulation Density	Describes the density of the piping insulation.
Refractory Thickness	If internal refractory exists on the piping segment, define the refractory thickness here.
Refractory Density	Describes the density of the piping refractory.
No Output	Permits the user to turn off output reporting for the selected pipe segment. If selected, then the piping segment will not be reported as part of the solution report. This is typically used for rigid construction elements where the results are not of interest.

Nozzle/PRO Piping Text Input

The second input mode available for constructing Nozzle/PRO piping models is the Text Input mode. In the Text Input mode, Beamer command text strings are used to instruct Nozzle/PRO how to construct the piping model. This input method is only recommended for advanced users familiar with the Beamer input file command system. Users can gain experience by first constructing the model using the Grid Input mode and then converting to Text Input mode as described below. It is highly recommended that user's attempting to utilize the Text Input mode refer to the Text Input mode help file by clicking the "Text Help" button.

New and experienced users may find it convenient to use the "Insert New Command" option. This option provides a simple interface to construct and insert the various Beamer commands into the Text Input mode. The Command Builder interface permits the user to input familiar information while letting Nozzle/PRO take care of the proper formatting of the various Text Input commands.

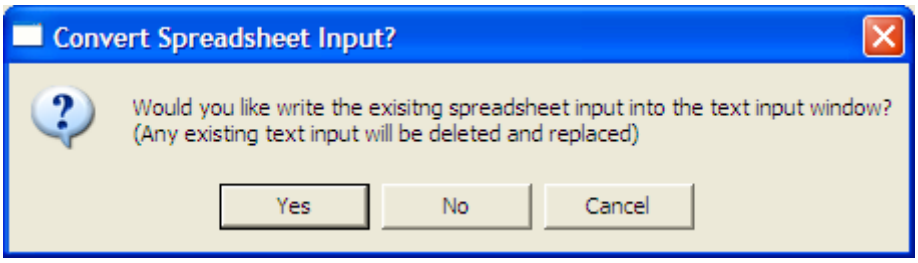
When switching into Text Input mode, Nozzle/PRO will display the following warning message. This message is intended to warn the user that any input or changes provided in the Text Input mode can not be converted back into the Grid Input spreadsheet. The original spreadsheet input data will remain available, but will not be revised with the Text Input revisions.



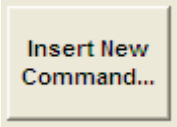
If existing spreadsheet input is already available, then the user will be presented with an opportunity to convert the spreadsheet input data into Text Input. This is often useful if the user wishes to quickly construct the model using the Grid Input mode and then make final alterations using the Text Input method prior to analysis. It is also useful when learning to use the Text Input mode since the converted spreadsheet data provides a good example to follow.

- Click YES to convert the existing Grid Input data into the Text Input format.
- Click NO to start with a clear Text Input format.

- Click Cancel to quit and return to the Grid Input mode.

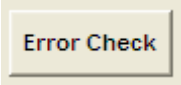


Insert New Command…



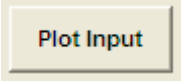
Launches the Command Builder interface. This allows the user to construct and insert Beamer commands into the Text Input interface without having to learn the strict format required for by the Text Input mode.

Error Check



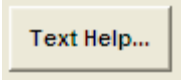
When Text Input mode is being used, it is easy to make simple input errors. The Error Check button will perform an error check on the text input and open the error report file for the user to review. It is recommended that this is used frequently and before any analysis is attempted.

Plot Input



Generate a graphical plot of the current input when using the Text Input mode.

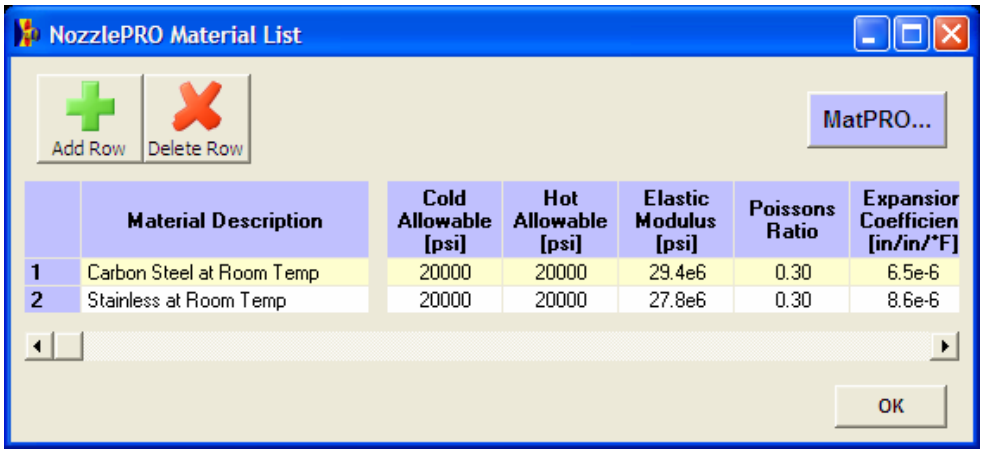
Text Help…



Opens the Beamer command help file.

Nozzle/PRO Piping Material List

The Nozzle/PRO Piping Material List is used define all piping materials which will be used in the piping models.

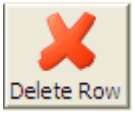


	Material Description	Cold Allowable [psi]	Hot Allowable [psi]	Elastic Modulus [psi]	Poissons Ratio	Expansior Coefficien [in/in/°F]
1	Carbon Steel at Room Temp	20000	20000	29.4e6	0.30	6.5e-6
2	Stainless at Room Temp	20000	20000	27.8e6	0.30	8.6e-6

Add Row

Additional materials may be defined using the Add Row button. Clicking the Add Row button will create a new input row in the Material Spreadsheet.

Delete Row



Materials may be deleted from the input form using the "Delete Row" button. The material which will be deleted is the active row in the material spreadsheet.

Material Input Spreadsheet

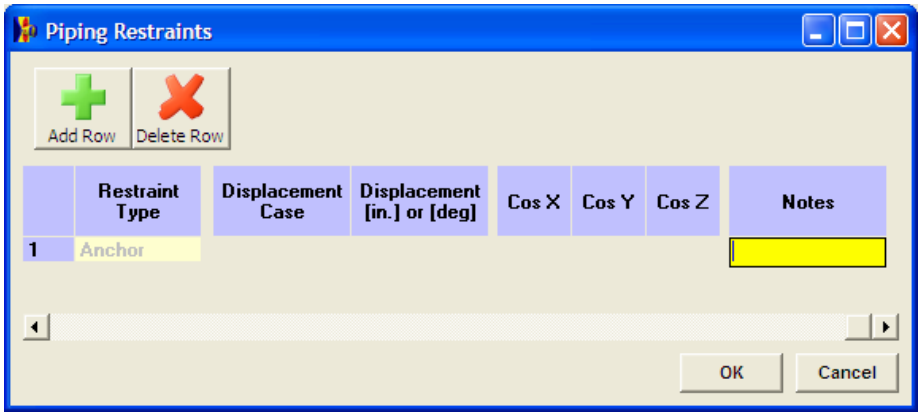
	Material Description	Cold Allowable [psi]	Hot Allowable [psi]	Fati
1	Carbon Steel	20000	20000	Low C
2	Stainless at Room Temp	20000	20000	Auster

The material input spreadsheet allows the user to define unique materials to be used in the piping model. Each row represents a different material. The material properties are described using the various columns of the spreadsheet. Each of these material property columns is described in the following table:

Input Column	Description
Material Description	Provides a unique description for each material defined in the material spreadsheet. This description will be available in the piping input spreadsheet when each piping segment is assigned a material of construction.
Cold Allowable	Defines the cold allowable stress for the material. The cold allowable should be the allowable stress at the minimum temperature of the operating cycle.
Hot Allowable	Defines the hot allowable stress for the material. The hot allowable should be the allowable stress at the maximum temperature of the operating cycle.
Elastic Modulus	Specify the elastic modulus of the material.
Poisson's Ratio	Defines the Poisson's ratio of the material.
Expansion Coefficient	Defines the thermal expansion coefficient of the material. This should be the mean thermal expansion between 70°F (21°C) and the operating temperature.
Density	Specify the density of the material.
Cold Yield	Defines the cold yield stress for the material. The cold yield should be the allowable stress at the minimum temperature of the operating cycle.
Hot Yield	Defines the hot yield stress for the material. The hot yield should be the allowable stress at the maximum temperature of the operating cycle.
Cold Tensile	Defines the cold tensile strength for the material. The cold yield should be the allowable stress at the minimum temperature of the operating cycle.

Nozzle/PRO Piping Restraints

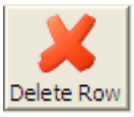
The Piping Restraints input form is accessed through the Nozzle/PRO Piping Input form's piping spreadsheet using the pop-up button in the Restraints column. The Piping Restraints input form is used to apply restraints to a specified piping segment.



Add Row

Additional piping restraints may be defined using the Add Row button. Clicking the Add Row button will create a new input row in the Piping Restraints spreadsheet.

Delete Row



Restraints may be deleted from the input form using the "Delete Row" button. The restraint which will be deleted is the active row in the material spreadsheet.

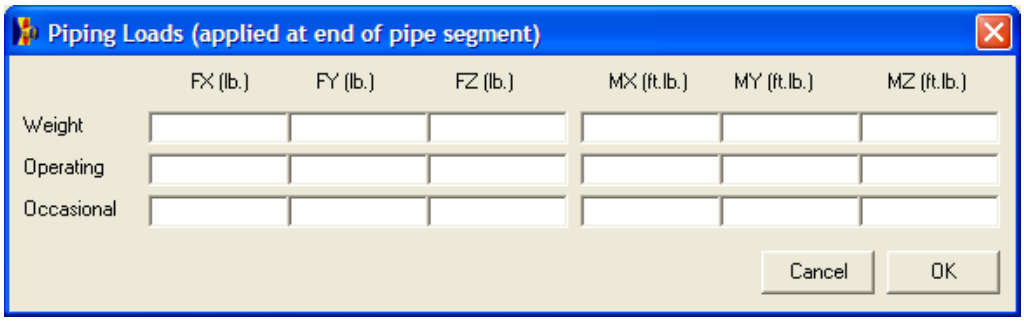
Input Column	Description
Restraint Type	Select the type of restraint to be applied to the piping segment.
Location	Defines the location for the restraint on the piping segment. The default (blank) is to apply the restraint to the "TO" end of the piping segment. The user may specify that the restraint should act at the "FROM" end of the piping segment by selecting the option "Start Node"
Stiffness	**Optional** Defines the translational or rotational stiffness of the user defined restraint. A default value of 1e15 will be used if no stiffness is specified. This option will typically be used to define a spring can or hanger with known spring stiffness. In this case, the user should define the linear spring stiffness defined by the manufacturer.
Initial Load	**Optional** Defines the initial load acting on the restraint.

	A common application of this input field is to define a spring support with an initial load.
Base Node ID	**Optional** The Base Node ID is used to tie degrees of freedom together between various portions of the model using the defined stiffness value.
Displacement Case	**Optional** Use to define the load case in which the displacement acting on the restraint will be applied. Note that displacements may only be defined for directional restraints such as X, Y, Z, Rx, Ry, and Rz. Displacements are not permitted for restraint types such as ANCHOR. If the user wishes to define a displacement on an anchor, then the anchor must be defined by six independent restraints (one for each degree of freedom), each with their own properties.
Displacement	**Optional** Defines the displacement magnitude to be applied to the restraint. See Displacement Case description above for additional considerations.
Cos X	**Optional** The vector component in the Global X direction for a skewed restraint type.
Cos Y	**Optional** The vector component in the Global Y direction for a skewed restraint type.
Cos Z	**Optional** The vector component in the Global Z direction for a skewed restraint type.
Notes	**Optional** Insert any descriptive text here to clarify the purpose of the restraint. These notes will appear as part of the output report and input echo.

Nozzle/PRO Piping Loads

The Piping Loads input form is accessed thru the Nozzle/PRO Piping Input form's piping spreadsheet (see the End Forces column in the piping spreadsheet discussion). The piping loads input form is used to define applied forces acting on the end of the selected piping segment.

All loads are defined in the global coordinate system and applied at the end of the specified piping segment.



Weight Loads
Weight loads include any sustained type loads acting in the installed case.

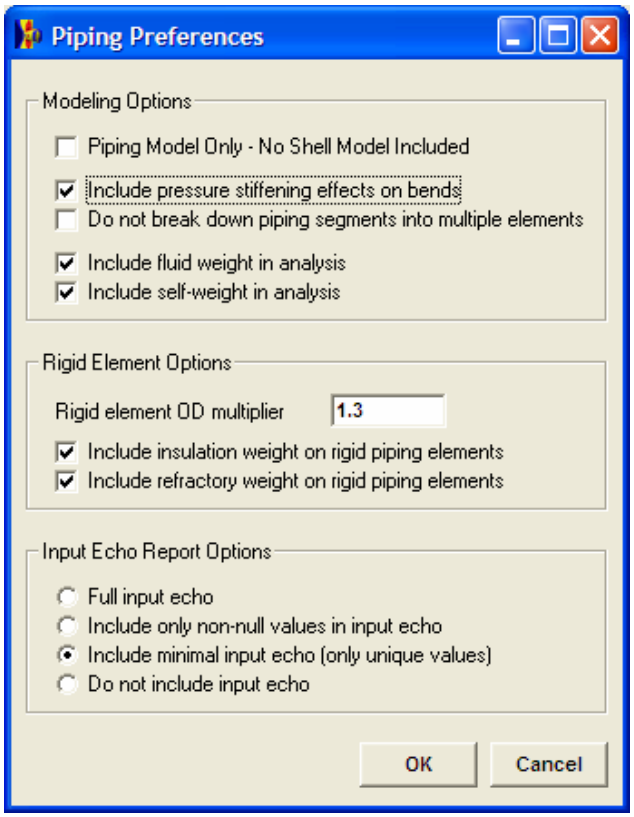
Operating Loads
Operating loads are any applied loads which are present during the operating case being evaluated. The operating loads should include the weight loads defined above. The difference between the operating loads and weight loads will be used to define the range case for fatigue analysis.

Occasional Loads
Occasional loads are usually due to wind, seismic, or other cases not defined as part of the typical operating conditions. Occasional loads can be evaluated either as contributing to primary type failures or fatigue failures. See Section 2 of the Nozzle/PRO manual for more discussion of these options.

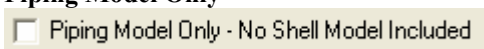
Note that the occasional loads should not include any portion of the weight or operating load cases. The occasional loads will be combined automatically by Nozzle/PRO where appropriate.

Nozzle/PRO Piping Preferences

Several options are available for controlling the solution and output format when using the Nozzle/PRO piping feature. The Nozzle/PRO Piping Preferences input form may be accessed through



Piping Model Only

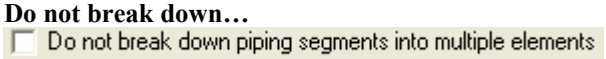


A piping only model with no Nozzle/PRO shell elements may be created by selecting this option. This option is typically used when the user wishes to analyze the piping elements only without including any shell model as part of the solution.

Include Pressure Stiffening on Bends

Indicates whether the pressure stiffened flexibility factors should be used according to ASME B31.3. The default is to include pressure stiffening effects.

Do not break down…



By default, Nozzle/PRO will break piping segments into several elements in the FEA analysis. This is to improve the stiffness model and dynamic solutions. The user may specify that the piping segments are not broken down into multiple elements by selecting this option.

Include Fluid Weight

☑ Include fluid weight in analysis

Defines whether or not fluid weight should be included in the piping analysis.

Include Self Weight

☑ Include self-weight in analysis

Defines whether or not weight loads due to material density should be included in the piping analysis.

Rigid Element OD Multiplier

Rigid element OD multiplier 1.3

Controls the diameter of rigid elements in the piping model. Note that this option will not be considered for rigid elements where the rigid weight has been specified as zero since these elements are represented as weightless rigid links in the model. Also, the rigid element OD multiplier will only affect the stiffness (via the element thickness) of the rigid piping element and not the weights calculated for the rigid.

Include Insulation Weight on Rigid Piping Elements

☑ Include insulation weight on rigid piping elements

Determines whether or not weight due to insulation will be added to the user defined rigid element weight. The insulation weight included on the rigid will be 1.75 times the weight of insulation on equivalent straight pipe.

Include Refractory Weight on Rigid Piping Elements

☑ Include insulation weight on rigid piping elements

Determines whether or not weight due to refractory will be added to the user defined rigid element weight. The refractory weight included on the rigid will be 1.75 times the weight of refractory on equivalent straight pipe.

Input Echo Report Options

Input Echo Report Options

- ○ Full input echo
- ○ Include only non-null values in input echo
- ◉ Include minimal input echo (only unique values)
- ○ Do not include input echo

Input Echo Report Option	Description
Full input echo	All available input fields will be reported in the user input echo report.
Include only non-null values in input echo	Only input fields which contain input explicitly defined by the user will be included in the input echo. Null fields will not be included as part of the input echo report.
Include minimal input echo	Only non-null unique values will be reported. Input values common between adjacent input rows in the piping input spreadsheet will not be included in the input echo report. This option will provide the most efficient output report since only the pertinent input is reported.
Do not include input echo	No input echo will be included as part of the solution report.

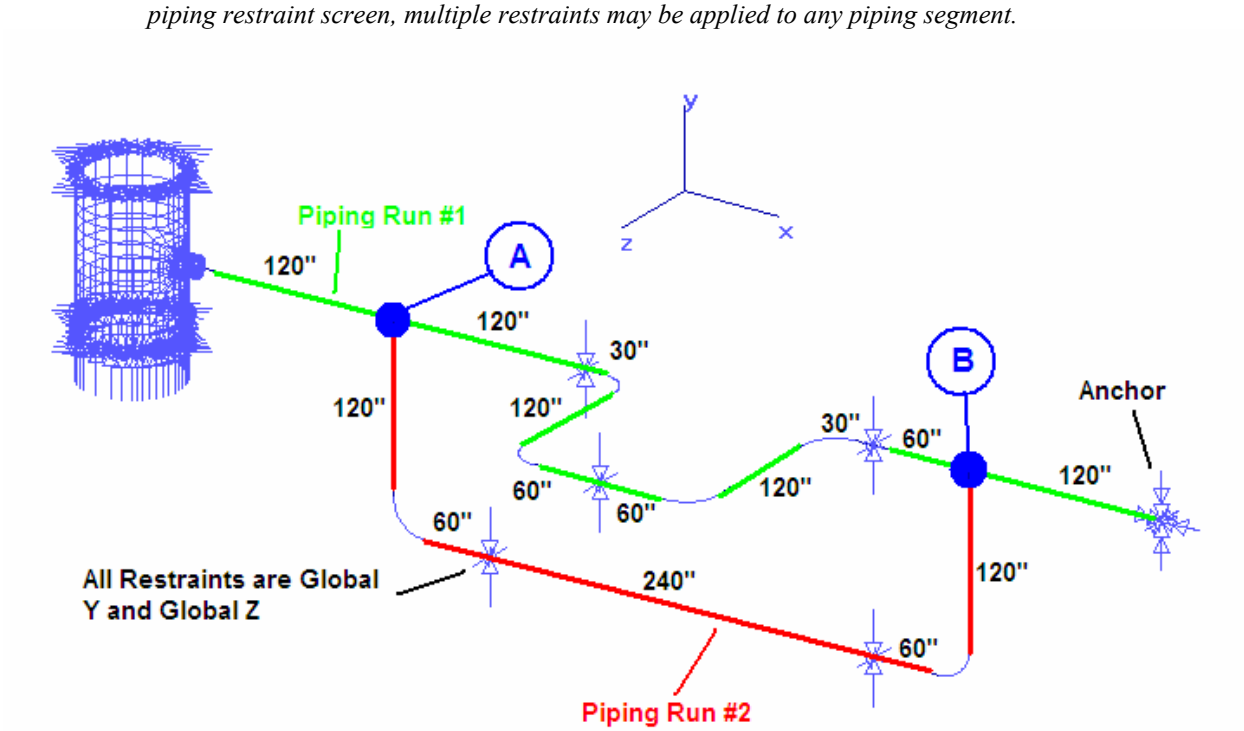
Piping Example #1- Using End Node Labels

The following example will illustrate the basic steps necessary to construct simple piping geometries attached to the Nozzle/PRO shell models. In this example, the piping layout as shown below will be used to demonstrate the usual input operations. For simplicity, default material properties will be used for the vessel, nozzle, and attached piping.

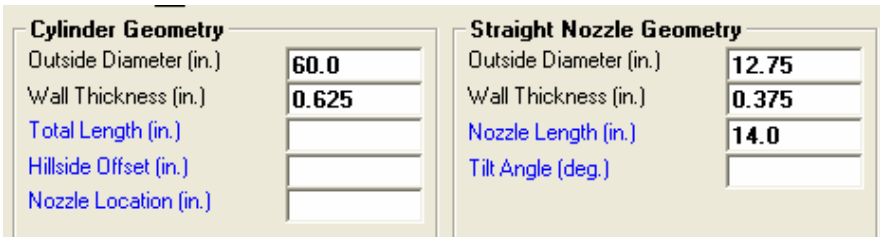
This sample model may be found in the "Samples1" folder of the installation directory with filename "NozzlePRO_Piping.nozzlepro"

In this example, several key concepts will be covered:

- *Creating "End Node Labels" which permit individual piping runs to connect to one another. In this example, Piping Run #2 begins at a branch connection along Piping Run #1 and also ends at a branch connection along Piping Run #1.*
- *Creating multiple boundary conditions at the end of a single piping segment. Using the piping restraint screen, multiple restraints may be applied to any piping segment.*

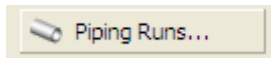


The first step is to construct the shell element model of the vessel and nozzle, where the first piping run will be attached. In this case, the vessel is 60" OD x 0.625" with a 12.75" OD x 0.375" x 14.0" long nozzle. A screen shot of the vessel input values are given below.

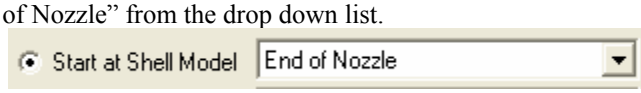


Once the vessel and nozzle geometry has been defined, the remaining work is to define the geometry of the attached piping runs. The following steps outline the general procedure to construct the piping model:

1. Click the "Piping Runs…" icon in the main Nozzle/PRO interface, located just above the nozzle geometry input frame.



2. Since Piping Run #1 will begin at the end of the nozzle, select the option "Start at Shell Model" and then select "End of Nozzle" from the drop down list.



3. Input the dimensions and geometry info for Piping Run #1. The inputs for Piping Run #1 are shown below. Some important features to note are:
   a. To create a new row in the spreadsheet, click the "Add Row" icon in the toolbar.
   b. The first pipe segment, which is the length between the nozzle and the intersection to Piping Run #2, has an "End Label" defined at the end of the pipe segment. This end node label defines a connection point where other piping nodes may be attached. In this example, Piping Run #2 will begin from End Label "A".
   c. Note that Row #12 also has an "End Label" defined. In this case, the end label is "B". End Label "B" will be the termination point for Piping Run #2.
   d. Restraints are created by selecting the blinking button in the Restraints column.
      i. When the piping restraints form appears, click "Add Row" to create a new piping restraint then select the appropriate restraint type and fill in the remaining properties for that restraint.
   e. Note that some input items are column duplicated. Column duplicated row entries are indicated by the light gray text. For instance, the pipe OD is defined only for the first input row and this defined value is automatically inherited by each row following.

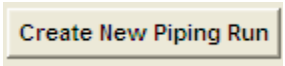
Input for Piping Run #1 in Example

	Description	X Length [in.]	Y Length [in.]	Z Length [in.]	OD [in.]	Thk [in.]	Material	End Label (Optional)	Pressure [psi]	Temp [°F]	u	Restraints	End Forces	Bend at End?
1	Start of piping run at	120			12.75	0.375	1. Carbon Steel	A	300	250				☐
2		120			12.75	0.375	1. Carbon Steel		300	250		Global Y		☐
3	Bend #1 in loop	30			12.75	0.375	1. Carbon Steel		300	250				☑
4	Bend #2 in loop			120	12.75	0.375	1. Carbon Steel		300	250				☑
5		60			12.75	0.375	1. Carbon Steel		300	250		Global Y		☐
6	Bend #3 in loop	60			12.75	0.375	1. Carbon Steel		300	250				☑
7	Bend #4 in loop			-120	12.75	0.375	1. Carbon Steel		300	250				☑
8		30			12.75	0.375	1. Carbon Steel		300	250				☐
9	Intersection with Run #2	60			12.75	0.375	1. Carbon Steel	B	300	250		Global Y		☐
10	End of Run at Anchor	120			12.75	0.375	1. Carbon Steel		300	250		Anchor		☐

Sample Piping Restraint for a Global Y Restraint in Piping Run #1.

	Restraint Type	Location
1	Global Y	End Node

4. After completing all the input for Piping Run #1, click the "Create New Piping Run" button to create a new input tab sheet for Piping Run #2. This should add a new tab to the tab list with title "Pipe #2".



5. Since Piping Run #2 will begin from a node within another piping run and not a point on the shell model, select the "Start at Node" option and set the starting node label to "A".

6. Next, input the piping geometry for Piping Run #2. The input spreadsheet should be as shown below for Piping Run #2:
   a. *Important – Piping Run #2 intersects Piping Run #1 and ends at the branch connection within Piping Run #1. Therefore, Piping Run #2 must connect to End Label "B" which was previously defined in Piping Run #1 To connect to node "B", the End Label in row 7 is defined as "B" as shown below.*
   b. *Global Y & Z restraints have been provided in the Restraints column for Rows #2 and #3 This is indicated by the word "Multiple". Click on the blinking button to define the multiple restraints to applied to the piping model. An example of the Global Y and Z restraints are given below.*

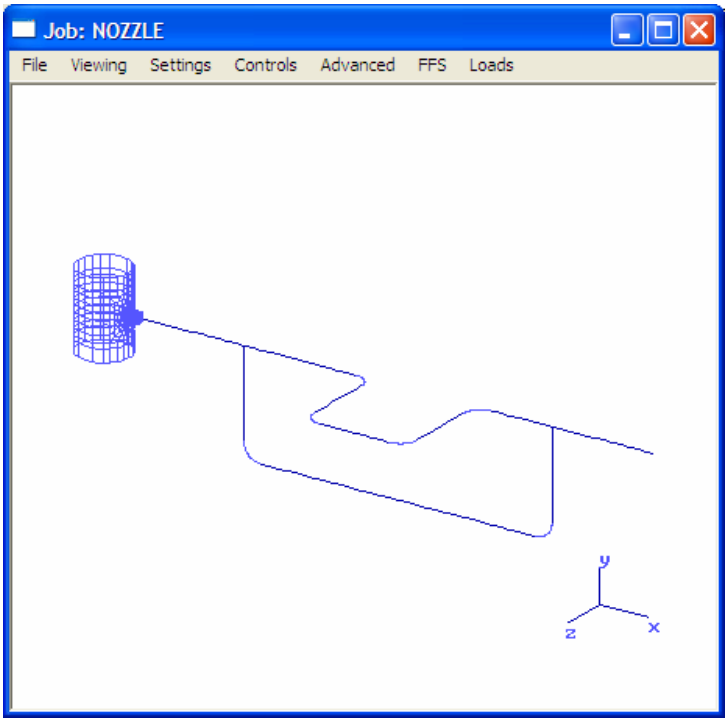
Input for Piping Run #2 in Example

	Description	X Length [in.]	Y Length [in.]	Z Length [in.]	OD [in.]	Thk [in.]	Material	End Label (Optional)	Pressure [psi]	Temp [°F]	Restraints	End Forces	Bend at End?
1	Start of Piping Run at "A"		-120		12.75	0.375	1. Carbon Steel		300	250			☑
2	Global Y & Z Restraints	60			12.75	0.375	1. Carbon Steel		300	250	Multiple		☐
3	Global Y & Z Restraints	240			12.75	0.375	1. Carbon Steel		300	250	Multiple		☐
4		60			12.75	0.375	1. Carbon Steel		300	250			☑
5	End of Piping Run at "B"		120		12.75	0.375	1. Carbon Steel	B	300	250			☐

Example of Global Y & Z Restraints for Piping Run #2

	Restraint Type	Location
1	Global Y	End Node
2	Global Z	End Node

7. After all of the piping model input has been specified, click CLOSE to return to the main Nozzle/PRO interface. The model may now be plotted or analyzed as normal with Nozzle/PRO models. A plot of the above input should yield the following model:



Piping Example #2 – Using Base Node ID's

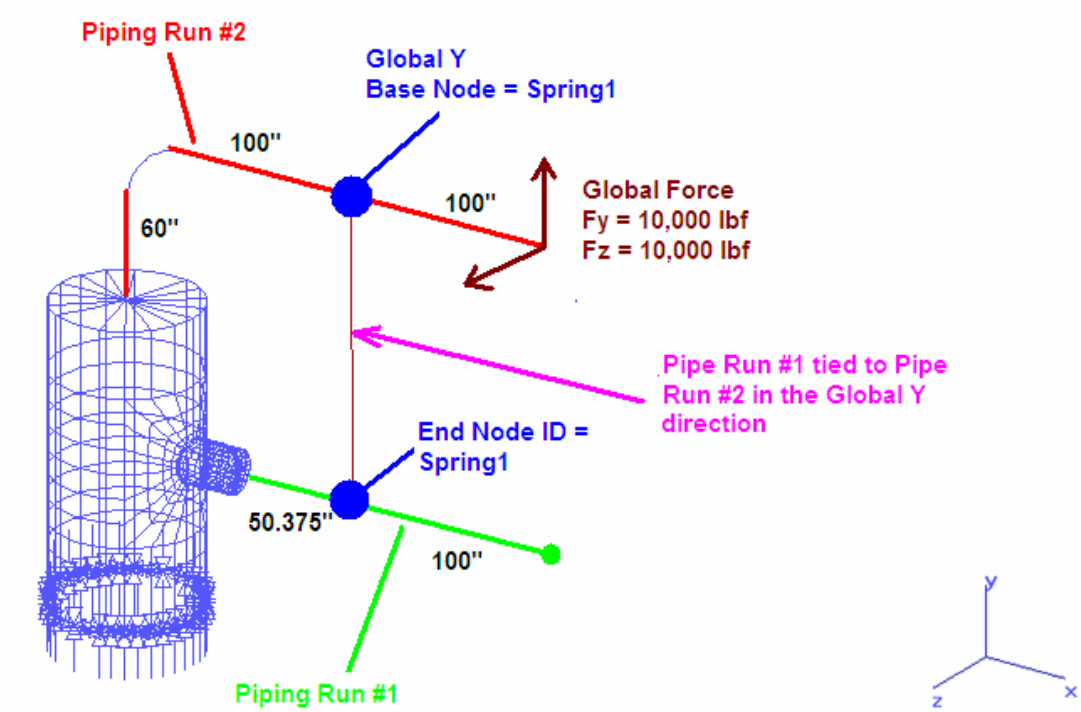
The following example will illustrate the use of Base Node ID's in Nozzle/PRO piping models. Base Node ID's are defined in the Piping Restraint screens and can be used in the piping input spreadsheet or within other Piping Restraint definitions. The Base Node ID's tie specific degrees of freedom (translational or rotational) together.

In this example, the goal is to tie two piping runs together to simulate a lower pipe supported from the upper pipe as shown below. The upper and lower pipes will be tied together using a user defined Base Node ID.

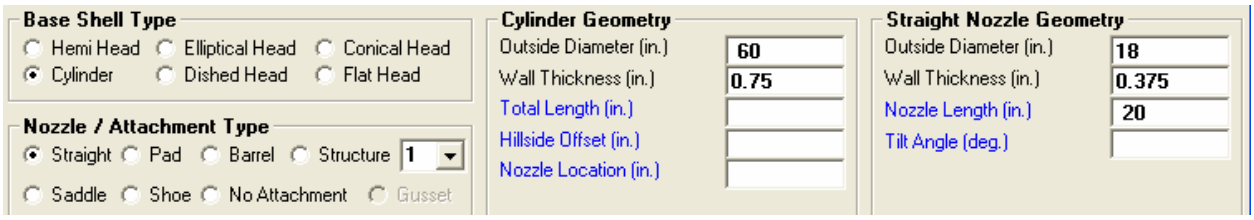
This sample model may be found in the "Samples1" folder of the installation directory with filename: "NozzlePRO_Base_Node.nozzlepro"

This example will illustrate the following concept:

- *Creating "Base Node IDs" which are used to tie degrees of freedom together. In this example, Pipe Run #1 will be "slaved" to Pipe Run #2 in the Global Y direction thru Base Node ID "Spring".*
- *Applying user defined loads to the piping model.*

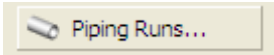


The first step is to construct the shell element model of the vessel and nozzle, where the first piping run will be attached. In this case, the vessel is 60” OD x 0.75” with a 18.0” OD x 0.375” x 20.0” long nozzle. A screen shot of the vessel input values are given below.

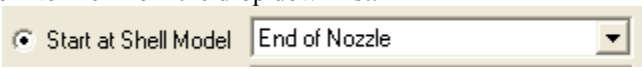


Once the vessel and nozzle geometry has been defined, the remaining work is to define the geometry of the attached piping runs. The following steps outline the general procedure to construct the piping model:

1. Click the “Piping Runs…” icon in the main Nozzle/PRO interface, located just above the nozzle geometry input frame.

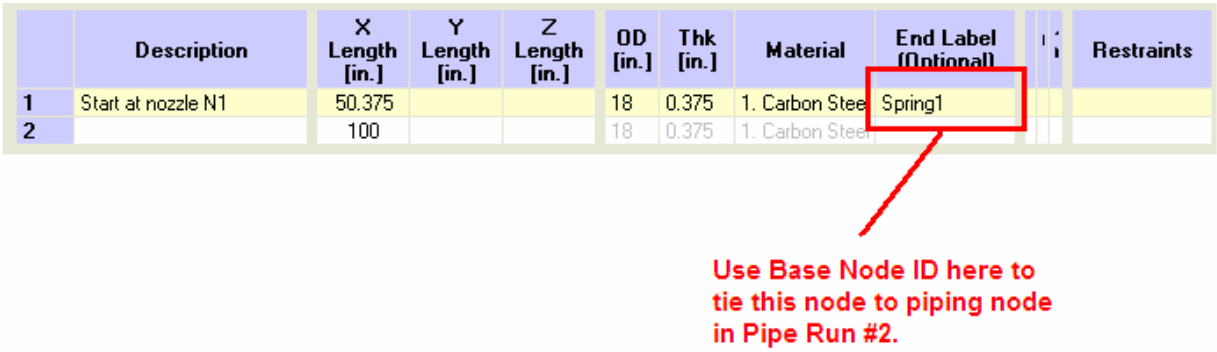


2. Since Piping Run #1 will begin at the end of the nozzle, select the option “Start at Shell Model” and then select “End of Nozzle” from the drop down list.

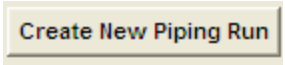


3. Input the dimensions and geometry info for Piping Run #1. The inputs for Piping Run #1 are shown below. Some important features to note are:
   a. To create a new row in the spreadsheet, click the “Add Row” icon in the toolbar.
   b. The first pipe segment, which is the length between the nozzle and the intersection to Piping Run #2, has an “End Label” defined at the end of the pipe segment. This End Label is defined as “Spring1”, which will be the same name assigned to the Base Node ID in Piping Run #2. Therefore, by defining “Spring1”, the pipe segment will be slaved to Piping Run #2 thru Base Node ID “Spring1”.

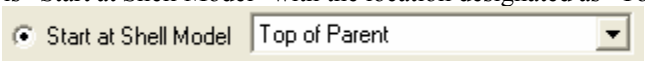
Input for Piping Run #1



4. After completing all the input for Piping Run #1, click the "Create New Piping Run" button to create a new input tab sheet for Piping Run #2. This should add a new tab to the tab list with title "Pipe #2".

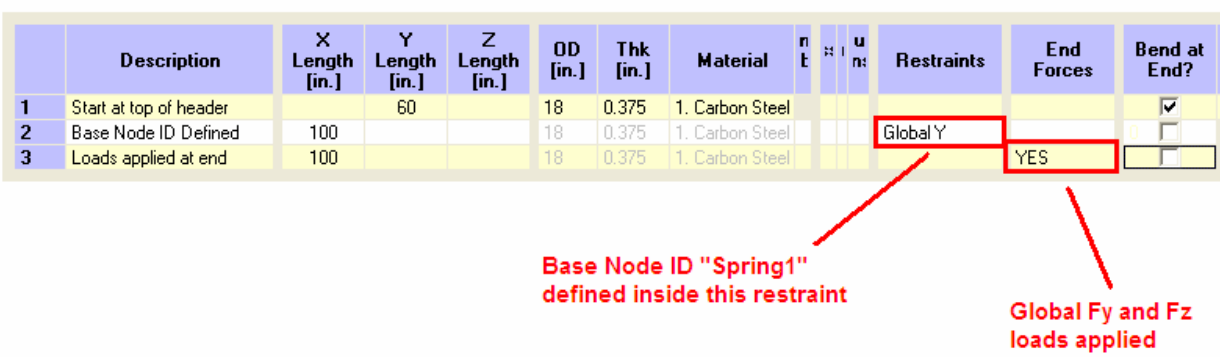


5. Piping Run #2 will begin from the top of the shell model. Therefore, the appropriate selection for the start location is "Start at Shell Model" with the location designated as "Top of Parent".



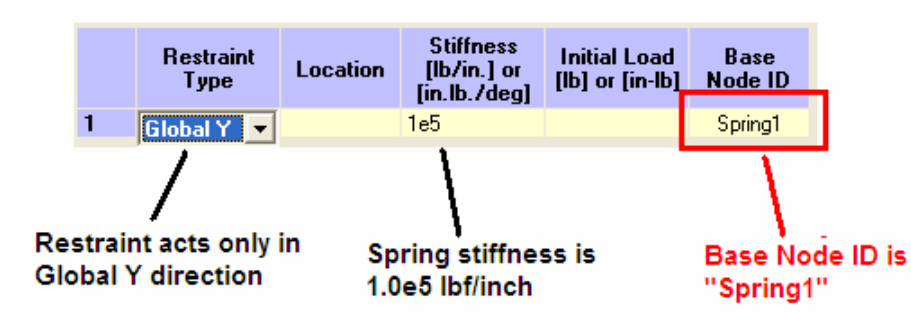
6. Next, input the piping geometry for Piping Run #2. The input spreadsheet should be as shown below for Piping Run #2 (see image of input screen below for additional guidance):
    a. *Important – In Row #2, the Global Y restraint with the Base Node ID which ties Piping Run #2 and Piping Run #1 together must be defined. To do this, follow these steps:*
        i. *Open the Piping Restraints screen selecting the cell in Row #2 within the "Restraints" column, then click the blinking arrow button.*
        ii. *When the Piping Restraints screen appears, click "Add Row" to generate a new restraint for the pipe segment.*
        iii. *Since the restraint should act in the Global Y direction only, select "Global Y" from the Restraint Type column.*
        iv. *The spring which is being simulated will have a linear stiffness of 1.0e5 lbf/inch. Specify this value in the Stiffness input column.*
        v. *Specify the Base Node ID which is used to uniquely identify this Base Node. In this example, the Base Node ID is "Spring1". Recall that this same variable name was used as the End Label within Piping Run #1. Now that the Base Node ID and restraint is created, Piping Run #1 and Piping Run #2 are tied together by the user defined stiffness in the Global Y direction.*
        vi. *Click OK to return to the main piping input form.*

Piping Input for Piping Run #2



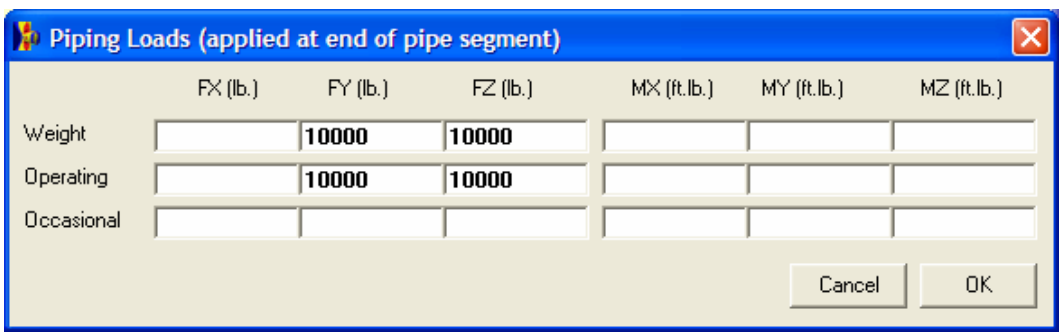
	Description	X Length [in.]	Y Length [in.]	Z Length [in.]	OD [in.]	Thk [in.]	Material	Restraints	End Forces	Bend at End?
1	Start at top of header		60		18	0.375	1. Carbon Steel			☑
2	Base Node ID Defined	100			18	0.375	1. Carbon Steel	Global Y		☐
3	Loads applied at end	100			18	0.375	1. Carbon Steel		YES	☐

Piping Restrain Input for Piping Run #2, Row #2

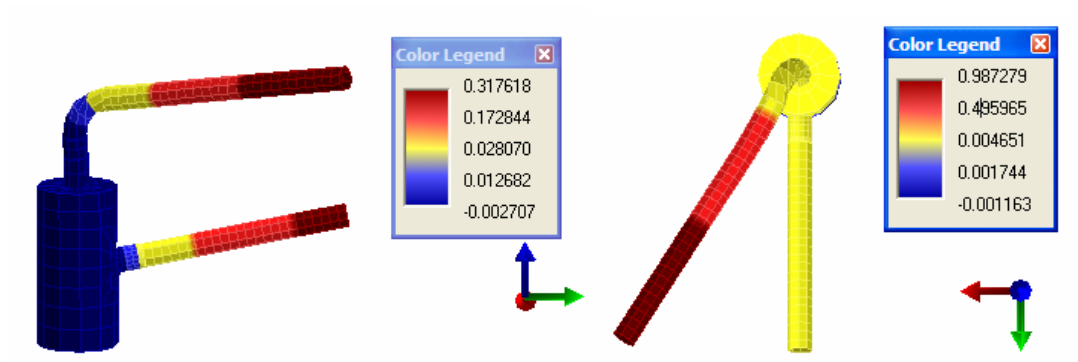


7. To illustrate the way in which the degrees of freedom will be tied together between Piping Run #1 and #2, the user can apply two directional loads to the free end of Piping Run #2. One load will be in the Global Y direction and one load in the Global Z direction. To define these end loads, use the following steps:
   a. *In Row #3, the last input segment for Piping Run #2, click on the cell within the column End Forces, then click the blinking arrow button* .
   b. *When the Piping Loads screen appears, specify 10,000 lbf in the weight case for the Global Y and Global Z directions, in the Weight and Operating load cases.*
   c. *Click OK to return to the main piping input form.*

Piping Restrain Input for Piping Run #2, Row #3



8. All of the input should now be complete. Next, click the RUN FE button in the main Nozzle/PRO interface screen to run the analysis. The results should indicate that only the Global Y degree of freedom has been linked between Piping Run #1 and #2.
   a. *Since only the Global Y direction degree of freedom is tied between Piping Run #1 and #2, there should only be displacement in the Global Y direction for Piping Run #1. There should be no displacement in the Global Z direction (other than a very small amount translated thru the shell model due to torsion loading of the shell model by Piping Run #2).*
   b. *Results from the FEA are shown below. Global Y displacements are shown in the figure at left. As expected, the Base Node has tied the Global Y degree of freedom between Piping Run #1 and #2, resulting in Y displacements. Since the Base Node has only linked the Global Y displacement, there are no other displacements translated through the base node tie. Therefore, even though a Z direction load is applied to Piping Run #2, it is not translated to Piping Run #1 as shown in the right-hand figure.*



Left – Global Y displacement showing affect of Base Node. Loads are transferred in Global Y direction.
Right – Global Z displacement. Note that Base Node did not transmit Global Z loads or displacements.

