**ASME B31.3 Spreadsheet User Guide (PRELIMINARY)**

# Introduction

The spreadsheet is a tool used to calculate stress values pertinent to ASME B31.3. Other commercial codes exist for this purpose such as CAESAR however unique non-standard tubing configurations can lead to errors in results and can sometimes simply not be modeled or ran in CAESAR. This spreadsheet does not perform the finite element method but instead utilizes NASTRAN for solving the finite element model created in FEMAP. The spreadsheet is a post processing tool using output from NASTRAN to calculate the necessary stresses to show tubing meets the B31.3 standard. FEMAP is used to summarize NASTRAN output to easily transfer data into the spreadsheet.

There are 8 tabs in the spreadsheet that require user input. Six of the tabs are for user input: “output”, “outputSL”, “elements”, “properties”, “Additional Input”, and “nodes”. The tab called “stress” is where the button to run the program resides as well as being the sheet that will display all output. The program is written in VBA which is embedded in the spreadsheet. The “BvsC” tab is used for background calculations and “tubeNodes” is a tab with a program still under development to put together margin charts.

For projects that have been run so far it has been found the program can take anywhere from 1- 15 minutes depending mainly on how many elements are being analyzed. All FEMAP instructions in this guide are shown using FEMAP 11.3 although they are basic features that shouldn’t change considerably from version to version. When taking data from FEMAP to the spreadsheet it is noticeable that some irrelevant data is copied over. When extracting data from FEMAP it is easier in some circumstances to pull this extra data. The program knows which data to use and which to disregard. For example, in FEMAP when you select elements to the data table the following columns are pulled:

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Prop ID | Type | Topology | Orientation Node | Orientation Vector | Color | Layer | Formulation | C1 | C2 |

The only columns actually used by the program are “ID”, “Prop Id”, “C1”, and “C2”. The other columns are disregarded.

This guide will go through each of the input tabs showing what info goes into each and how to extract it from FEMAP. From that point all that is left to do is hit run and the program will complete. There are a few cells on the “stress” tab near the bottom to monitor progress through various subroutines. Lastly, the guide will go through the code behind the spreadsheet highlighting key calculations. All equations used in this guide are numbered and their ASME B31.3 equivalent equation number or paragraph number is listed where applicable in the equation section of this guide. All symbols used in equations in this guide are defined in the Nomenclature section of this guide.

Contents

[Introduction 1](#_Toc497995486)

[Equations 3](#_Toc497995487)

[Nomenclature 3](#_Toc497995488)

[Input 4](#_Toc497995489)

[“properties” tab 4](#_Toc497995490)

[“elements” tab 6](#_Toc497995491)

[“nodes” tab 7](#_Toc497995492)

[“output/outputs” tab 7](#_Toc497995493)

[“Additional Input” tab 8](#_Toc497995494)

[Output 8](#_Toc497995495)

[“stress” tab 8](#_Toc497995496)

[Program Outline 9](#_Toc497995497)

[Sample Calculations 12](#_Toc497995498)

[Straight Tube Example 12](#_Toc497995499)

[Tube Bend Example 16](#_Toc497995500)

# Equations

|  |  |
| --- | --- |
| **User Guide Equation Number** | **Equivalent ASME B31.3 Equation Number or paragraph number** |
| 1 | N/A |
| 2 | N/A |
| 3 | Appendix D |
| 4 | Appendix D |
| 5 | Appendix D |
| 6 | 320.2 |
| 7 | 320.2 |
| 8 | N/A |
| 9 | (18) |
| 10 | 319.4.4, 320.2 |
| 11 | 319.4.4, 320.2 |
| 12 | (17) |
| 13 | (23b) |
| 14 | (23c) |
| 15 | (23d) |
| 16 | (23a) |
| 17 | (3d) |
| 18 | (3e) |
| 19 | (3a) |
| 20 | (3b) |
| 21 | (3c) |
| 22 | 302.3 |
| 23 | (1c) |
| 24 | (1a) |
| 25 | (1b) |

# Nomenclature

Ro,i = Radius of tube (outer, inner)

D = Diameter of tube

t = tube thickness

ii = in plane flexibility stress intensity factor, ASME B31.3 319.4.4

io = out of plane flexibility stress intensity factor, ASME B31.3 319.4.4

h = flexibility characteristic, Appendix D

Ii = in plane flexibility stress intensity factor, ASME B31.3 320.2

Io = out of plane flexibility stress intensity factor, ASME B31.3 320.2

Z = Section modulus

Sb = Bending Stress

St = Torsional Stress

Sa = Axial Stress

SE = Flexibility Combined Stress

it = Torsional flexibility stress intensity factor, ASME B31.3 319.4.4

ia = Axial flexibility stress intensity factor, ASME B31.3 319.4.4

It = Torsional flexibility stress intensity factor, ASME B31.3 320.2

Ia = Axial flexibility stress intensity factor, ASME B31.3 320.2

IIntrados, extrados = Strength reduction factor at tube bend (inner/outer wall of tube)

P = Internal design gage pressure

S = stress value for material, Table A-1

Ec = Casting quality factor, Table A-1A

Ej = Weld Quality factor, Table A-1B

E = Quality Factor

W = Weld joint strength reduction factor, ASME B31.3 302.3.5(e)

Y = coefficient from table 304.1.1

*f* = stress range factor

*fm* = maximum value of stress range factor

N = Maximum number of full displacement cycles over life

Sc = basic allowable stress at minimum temperature, Table A-1

Sh = basic allowable stress at maximum temperature, Table A-1

SL = stress due to sustained loads

# Input

First step is to open the .modfem for the model being analyzed. Next unlock the data table and display it. Next turn the select toolbar on if it is not already on. Now follow the instructions for each of the following input tabs.

## “properties” tab

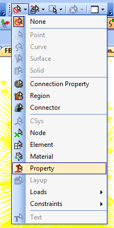
This tab takes all data that appears on any property card relevant to the tubing beams. The columns are as follows:

|  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Title | Type | Shape | Material | Color | Layer | End A Area | Y Shear Area | Z Shear Area | I1 | I2 |

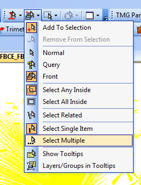
|  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- |
| I12 | NS Mass/Len | Warp Const | J | Perimeter | Neutral Axis Off A: Y | Neutral Axis Off A: Z |

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| Neutral Axis Off B: Y | Neutral Axis Off B: Z | Pt 1: Y | Pt 1: Z | Pt 2: Y | Pt 2: Z | Pt 3: Y | Pt 3: Z | Pt 4: Y | Pt 4: Z |

To query this data the user must know which properties are used for the tubing. Once the property card IDs are known make the selector entity correspond to property.



Make the selector mode correspond to multiple.



Choose selector actions to be dialog.



Add each relevant property ID card to the dialog box and click ok.

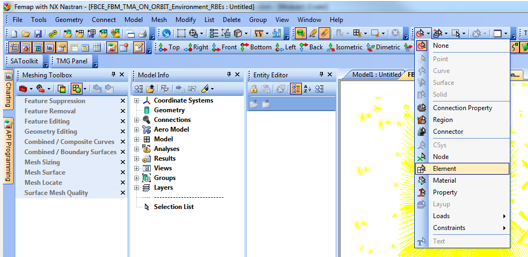
The property data will now be shown in the data table. Right click in the data table and select all, right click and select copy rows, and paste into cell A1 of the “properties” tab.

## “elements” tab

This tab takes each element that represents a tube that needs to be analyzed. The columns are as follows:

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Prop ID | Type | Topology | Orientation Node | Orientation Vector | Color | Layer | Formulation | C1 | C2 |

These columns are the default columns queried when selecting elements in FEMAP to the data table.

Make select entity correspond to “element” 

Choose selector action to be dialog. Change the “method” to “Property” and enter all property card IDs used in the properties tab and click ok. All relevent element data will now be shown in the data table. Select all data in the data table and copy and paste it into cell A1 of the “elements” tab.

## “nodes” tab

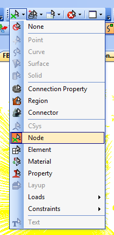
This tab contains all data for each node appearing on any element in the elements tab.

The columns in this tab are as follows:

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Color | Layer | Out CSys | Type | Superelement ID | Def CSys | X-Def | Y-Def | Z-Def |

|  |  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| Active Csys | X-Act | Y-Act | Z-Act | 1 | 2 | 3 | 4 | 5 | 6 | CSys for Output List | Out-Deform Dir 1 | Out-Deform Dir 2 | Out-Deform Dir 3 |

To populate this tab first highlight all element IDs from the first column of the elements tab. Next in FEMAP make selector entity correspond to node.



Choose selector actions to be dialog. Change the method to “on elements” and click “pick ^” 🡪 “paste”. Click ok and the data table should populate with all necessary nodes. Copy the data table and paste it into cell A1 of the “nodes” tab.

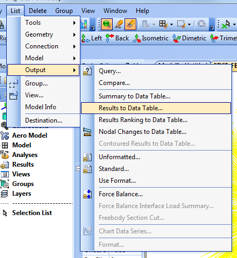
## “output/outputs” tab

These tabs take the moments in two orthogonal planes for each node of each element, the axial force on each node of each beam, and the torques on each node of each element for two load cases.

The columns are as follows:

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | CSys ID | 1002..Thermal Loads, 3014..Beam EndA Plane1 Moment | 1002..Thermal Loads, 3015..Beam EndA Plane2 Moment | 1002..Thermal Loads, 3016..Beam EndB Plane1 Moment | 1002..Thermal Loads, 3017..Beam EndB Plane2 Moment | 1002..Thermal Loads, 3022..Beam EndA Axial Force | 1002..Thermal Loads, 3023..Beam EndB Axial Force | 1002..Thermal Loads, 3024..Beam EndA Torque | 1002..Thermal Loads, 3025..Beam EndB Torque |

This data is required for each beam element being analyzed. Click “List”🡪 “output” 🡪 “Results to Data Table”



Click ok. The mark the check box next to the output set corresponding the load case for displacement loads. Select the check box next to output vectors: “3014..Beam EndA Plane1 Moment”, “3015..Beam EndA Plane2 Moment”, “3016..Beam EndB Plane1 Moment”, “3017..Beam EndB Plane2 Moment”, “3022..Beam EndA Axial Force”, “3023..Beam EndB Axial Force”, “3024..Beam EndA Torque”, “3025..Beam EndB Torque”. Click ok. Similar to how the nodes were queried, select and copy all element IDs from the “elements” tab. Click “pick ^” 🡪 “paste”. Click ok. The data table should populate with all output requested. Copy and paste this data into cell A1 of the “output” tab.

If there is a sustained loads case this process needs redone with that corresponding output set and data should be placed in the “outputSL” tab. If there is no sustained load case this tab can be left blank.

## “Additional Input” tab

This tab has space for the user to input the following parameters: pressure, material allowable at room temperature, material allowable at maximum temperature, weld joint strength reduction factor, table 304.1.1 coefficient, mechanical allowance, the maximum value of the stress range factor, number of cycles, casting quality factor, and weld quality factor.

# Output

## “stress” tab

Next go to the “stress” tab and click the “RUN” button near cell C61. See the % complete cells (H58 🡪M58 🡪 B58) for progress through various subroutines. The last step of the program is complete when the borders are drawn on the stress tab. All tube thicknesses are listed in rows 32-34. They are highlighted in green if they are larger than required and red if not. All Sustained loads stresses are in row 57 and are highlighted green if lower than the material allowable found in row 44 and highlighted red if not. The displacement stresses are located in row 52 and are highlighted green if lower than the displacement stress allowable found in row 48 and highlighted red if not.

The elements that fail due to sustained load stress are found listed below A59, the elements that fail due to displacement stress are listed below B59.

# Program Outline

The first thing the program does is clear the data from the “Stress” tab. So any previous results must be saved before running a new analysis. The next step for the program is to iterate through each element and determine if it is part of a bend or straight section of tubing.

To do this the program loops through each node from the “Node” tab. For each node it loops through each element checking for elements that contain that node. When an element is found that contains said node the 3-D coordinates of each of that elements end nodes are pulled from the “nodes” tab. A unit vector in the direction of the element is calculated and stored. The program than begins searching for any other elements that contain said node. When another element with said node is found, similar to before, the 3D coordinates for each end nodes of that element are pulled and a unit vector is calculated and stored. If the two unit vectors for elements sharing the common node are the same a tally is added to an array for each of the two elements. At this point the program moves on to the next node and repeats the process finding unit vectors of each element sharing the common node and adding a tally to each element that had the same unit vector. This process creates an array with a row for each element and a tally next to it representing how many elements that share a node with it have the same unit vector along their slope. Elements with a tally of 0 are parts of a bend because they have a differing slope than each adjacent elements.

The next portion of the program checks for the bend radius of each element identified as being part of a bend. The sub routine called radofCurve loops through each element. For each element it checks if it is a bend or it is part of a straight section by looking at the output of the bendstraightCheck subroutine. If the element is part of a straight section the sub routine goes to the next element. If the element is part of a bend it will first pull the 3-d coordinates of each end node as well as the end node of an adjacent element. The following equation is used to calculate the radius of curvature of a curve through these three points. The 3-D coordinates of each node are used to calculate necessary parameters.

Equation

The bend radius is than displayed on the “stress” tab in row 25 and the routine moves on to the next element.

The next portion of the program begins writing more data to the “stress” tab. It loops through each element on the “elements” tab. The Element ID, node 1, and node2 are pulled from the “elements” tab and written to rows 1, 2 and 3 of the “stress” tab. Next NASTRAN output, including “Beam end A plane 1 moment”, “Beam end A plane 2 moment”, “Beam end B plane 1 moment”, “Beam end B plane 2 moment”, “Beam end A axial force”, “Beam end B axial force”, “Beam end A torque”, “Beam end B torque” is pulled from the “output” tab and written to rows 4-11 on the “stress” tab. Next the tube’s outer radius and area is pulled from the property cards stored in the “properties” tab and written to rows, 23 and 53. The thickness is calculated with the outer radius and the area of the tube using the following equation.

Equation

This thickness is written on row 22 of the “stress” tab. The inner radius is than calculated and written to row 24 on the “stress” tab. Next the flexibility stress intensity factors for tube bends are calculated. The equations from ASME B31.3 for in plane and out of plane stress intensity factors are the following.

Equation

Equation

Where h is calculated using the following equation found in appendix D.

Equation

These values are written to rows 35, 36, and 38 on the “stress” tab. For straight sections of tube ii, and io are set to 1. Next the sustained loads stress intensity factors are calculated using the following equations.

Equation

Equation

The minimum for these values is 1, so if the calculated stress intensity factor is below 1 then it is replaced with 1. These values are written to rows 37 and 39.

Next the stress intensity factor on torsional stress and axial stress is written to rows 40 and 41 which is equal to 1 without applicable data.

Next the section modulus is calculated using the following equation.

Equation

This value is written to row 42 on the “stress” tab.

Next flexibility bending, torsional, and axial stress are calculated and then used to calculate flexibility combined stress using the following equations.

Equation

Equation

Equation

Equation

These values are written to rows 49-52 on the “stress” tab.

Next sustained bending, torsional, and axial stress are calculated and then used to calculate sustained combined stress using the following equations.

Equation

Equation

Equation

Equation

These values are written to rows 49-52 on the “stress” tab.

Next the material allowable stress, pressure, quality factor, and weld joint strength reduction factor is written to rows 21 and 26-28. The strength reduction factor due to a tube bend is calculated at the intrados and extrados of the tube using the following equations.

Equation

Equation

The maximum of these is used for tubing thickness calculations and is written to row 29 on the “stress” tab.

Next the material temperature reduction factor, Y, is written to row 30 of the “stress” tab. These values are tabulated in table 304.1.1 of ASME B31.3.

Next the minimum allowable tubing thicknesses are calculated using one of the following equations depending on if the element is part of a straight piece of tubing or a bend. The lower of the two calculated straight tube thicknesses is used to calculate margins.

Equation

Equation

Equation

Equation

These values are written to rows 32-34. Only applicable values are shown, non-applicable values are left blank. These values are compared to actual thicknesses and are color coated based on if the tube thickness meets ASME B31.3 satisfactorily. The cell will be green if it satisfactorily meets thickness requirements and red if not.

Next the material allowable stress at room temperature, casting quality factor, longitudinal weld joint factor, and the stress range factor are written to rows 43-47. Material allowables are found in table A-1, casting quality factors can be found in table 302.3.4, longitudinal weld joint factors are found in table 30.3.4, and the stress range factor is calculated using the following equation.

Equation

Next the flexibility stress allowable is calculated. If the sustained load stress is greater than the material allowable at maximum operating temperature the following equation is used to calculate flexibility stress allowables.

Equation

If the material allowable at maximum operating temperature is greater than the sustained load stress than the flexibility stress allowable is calculated using the following equation.

Equation

The next portion of the program compares actual stress to allowable stress levels. For flexibility stress Se is compared to SA and for sustained load stress Sl is compared to Sh. If the actual stress is less than the allowable stress the cell is colored green and if not the cell is colored red.

At this point the program goes on to the next element and repeats this process. This will continue until all elements are processed.

For elements that fail the program will write the element ID, max flexibility or sustained stress depending on which fails, allowable stress to the bottom left of the “stress” tab. The program will also calculate margins of safety on those stresses.

# Sample Calculations

Two arbitrary examples are now shown. One is the analysis of an element representing a straight tube and the other is of an element representing a tube bend showing all input from FEMAP and NASTRAN, all calculations pertinent to ASME B31.3, and finally results. This section serves the dual purpose of validating the spreadsheet as well as providing clarity to the process of the spreadsheet. This example assumes only a flexibility stress case is present and no sustained load case.

## Straight Tube Example

ELEMENT ID: 742420

“Output” Tab data for element 742420

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | CSys ID | 2..NX NASTRAN Case 1, 3014..Beam EndA Plane1 Moment | 2..NX NASTRAN Case 1, 3015..Beam EndA Plane2 Moment | 2..NX NASTRAN Case 1, 3016..Beam EndB Plane1 Moment | 2..NX NASTRAN Case 1, 3017..Beam EndB Plane2 Moment | 2..NX NASTRAN Case 1, 3022..Beam EndA Axial Force | 2..NX NASTRAN Case 1, 3023..Beam EndB Axial Force | 2..NX NASTRAN Case 1, 3024..Beam EndA Torque | 2..NX NASTRAN Case 1, 3025..Beam EndB Torque |
| 742420 | 0 | 20 | 9 | 22 | 8 | 14 | 14 | 0.3 | 0.3 |

“elements” Tab data for element 742420

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Prop ID | Type | Topology | Orientation Node | Orientation Vector | Color | Layer | Formulation | C1 | C2 |
| 742420 | 114..111002\_TUBE\_ASSY\_FSML\_FSMU-1\_ASM | BEAM | Line, 2-noded | 0 | 0., 1., 0. | 124 | 1 | None | 742387 | 742447 |

“properties” Tab data for element 742420

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Title | Type | Shape | Material | Color | Layer | End A Area | Y Shear Area | Z Shear Area | I1 |
| 114 | 111002\_TUBE\_ASSY\_FSML\_FSMU-1\_ASM | BEAM | Circular Tube | 143..304 CRES | 110 | 1 | 0.02 | 0.0106 | 0.0106 | 0.0001 |

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| I2 | I12 | NS Mass/Len | Warp Const | J | Perimeter | Neutral Axis Off A: Y | Neutral Axis Off A: Z | Neutral Axis Off B: Y | Neutral Axis Off B: Z | Pt 1: Y |
| 0.0001 | 0 | 0 | 0 | 2E-04 | 0.785 | 0 | 0 | 0 | 0 | 0 |

|  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- |
| Pt 1: Z | Pt 2: Y | Pt 2: Z | Pt 3: Y | Pt 3: Z | Pt 4: Y | Pt 4: Z |
| -0.125 | 0.125 | 0 | 0 | 0.125 | -0.125 | 0 |

“nodes” Tab data for element 742420

|  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Color | Layer | Out CSys | Type | Superelement ID | Def CSys | X-Def | Y-Def | Z-Def | Active Csys | X-Act | Y-Act | Z-Act | 1 | 2 | 3 | 4 | 5 | 6 |
| 742387 | 46 | 1 | 0 | 0..Node | 0 | 0 | -4.311 | 16.3 | 8.32 | 0..Global Rectangular | -4.311 | 16.3 | 8.32 | 0 | 0 | 0 | 0 | 0 | 0 |
| 742447 | 46 | 1 | 0 | 0..Node | 0 | 0 | -4.189 | 16.3 | 8.32 | 0..Global Rectangular | -4.189 | 16.3 | 8.32 | 0 | 0 | 0 | 0 | 0 | 0 |

“stress” Tab data for element 742420

|  |  |  |  |
| --- | --- | --- | --- |
| Element Data |  | Element Id | 742420 |
|  | point 1 | 742387 |
|  | point 2 | 742447 |
| Nastran Output |  | Beam end A plane 1 moment | 20 |
|  | Beam end A plane 2 moment | 9 |
|  | Beam end B plane 1 moment | 22 |
|  | Beam end B plane 2 moment | 8 |
|  | Beam end A axial force | 19 |
|  | Beam end B axial force | 19 |
|  | Beam end A torque | 12 |
|  | Beam end B torque | 12 |
| Nastran Output (Sustained Loads) |  | Beam end A plane 1 moment, SL | 0 |
|  | Beam end A plane 2 moment, SL | 0 |
|  | Beam end B plane 1 moment, SL | 0 |
|  | Beam end B plane 2 moment, SL | 0 |
|  | Beam end A axial force, SL | 0 |
|  | Beam end B axial force, SL | 0 |
|  | Beam end A torque, SL | 0 |
|  | Beam end B torque, SL | 0 |
| Property |  | Property Card | 114..111002\_TUBE\_ASSY\_FSML\_FSMU-1\_ASM |
| Pressure | (all at 60 for now) | Pressure | 60 |
| Element Geometry |  | thickness, as designed | 0.029 |
|  | r2 = r | 0.125 |
|  | r1 = r-t | 0.096 |
|  | Bend Radius |  |
| Material stress allowable |  | S | 16700 |
| Quality Factor |  | E | 0.8 |
| Weld Reduction factor |  | W | 1 |
| Max(Intrados, extrados) |  | I=max(3d, 3e) | 0.5 |
| 304.1.1 |  | Y | 0.4 |
| Allowances | ??? | c | 0 |
| Calculated thickness minimums | 304.1.2, 304.2.1 | thickness, straight(3a) | 0.0005 |
| 302.3.5 | thickness, straight(3b) | 0.0005 |
|  | thickness, bends (3c) |  |
| Stress Intensity Factors, moment indices, force indices | 320.1 | h = t\*R/r | 0 |
|  | ii = .9/h^(2/3) | 1 |
|  | Ii = .75 ii or 1, greater of 2 | 1 |
|  | io = .75/h^(2/3) | 1 |
|  | Io = .75 io or 1, greater of 2 | 1 |
|  | It = 1 w/o applicable data | 1 |
|  | Ia = 1 w/o applicable data | 1 |
| Section Modulus |  | Z = pi\*(r2^4-r1^4)/(4\*r2) | 0.001 |
| Limit stresses due to sustained loads and displacement strains | 319.3.4 | Sc = allowable at room temp | 25000 |
|  | sh = allowable at max temp | 16700 |
|  | Ec | 0.8 |
|  | Ej | 1 |
|  | f | 1 |
|  | sa(allowable) | 52125 |
| Flexibility Stress | 319.4.4 | Sb(18) | 45310 |
|  | st(17) | 12000 |
|  | sa(17) axial | 1900 |
|  | Se | 52960 |
| Sustained Loads Stress | 320.2 | A | .02 |
|  | Sb | 0 |
|  | St | 0 |
|  | Sa(axial) | 0 |
|  | Sl | 0 |

Starting from the top of the “stress” tab you can see the element ID, 742420, is listed from the “elements” tab. Next the end nodes are listed from the “elements” tab. Next the load values are pulled from the “output” and “outputSl” tab. Next the property card is pulled from the “properties” tab. Next the pressure is listed. The pressure is pulled from cell L61. As of now the pressure is the same for each element. Any tubing section that requires a different pressure needs to be a separate spreadsheet. Next the element geometry is calculated and pulled from the “property” tab.

r2 = .125

A = .02

Equation

r1 = r2 – t = .125-.029 = .096

The bend radius is blank because this element is part of a straight tube.

Next the material allowable stress, quality factor, and weld reduction factor are listed. These are pulled from the additional input section of the “stress” tab. Next the larger of the Intrados or extrados bend reduction factor is listed. These are equal to zero because this element is part of a straight tube.

Next the Table 304.1.1 coefficient and mechanical allowance is listed. These are pulled from the additional input section of the “stress” tab. Next the thicknesses are calculated.

Equation

Equation

The thickness of the tubes is larger than these calculated minimums so the cells are green.

Next the stress intensity factors, bending stress intensity factors, and axial stress intensity factors are calculated. Because this is a straight tube the value of h is 0. This causes io and ii to be 0; however the minimum value of ii and io is 1. Ii is the greater of .75ii and 1, which is 1. Io is the greater of .75io and 1, which is also 1. It and Ia are always 1 without applicable testing data. Next the section modulus is calculated.

Equation

After this the limit stresses and applicable quality factors are listed. These are all pulled from the additional input section of the “stress” tab. Because the stress due to sustained loads is less than the material allowable stress equation 24 is used to calculate the flexibility allowable stress.

= Equation

Next the flexibility stresses are calculated.

Equation

Equation

Equation

Equation

The flexibility stress is higher than the allowable so the cell is colored red.

Next the sustained load stresses are calculated. The process is the same as it was for calculating flexibility stress however a sustained load case is used. For this example there was no sustained load case so Sl = 0. Sl is lower than the material allowable so the cell is colored green.

## Tube Bend Example

ELEMENT ID: 742450

“Output” Tab data for element 742450

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | CSys ID | 2..NX NASTRAN Case 1, 3014..Beam EndA Plane1 Moment | 2..NX NASTRAN Case 1, 3015..Beam EndA Plane2 Moment | 2..NX NASTRAN Case 1, 3016..Beam EndB Plane1 Moment | 2..NX NASTRAN Case 1, 3017..Beam EndB Plane2 Moment | 2..NX NASTRAN Case 1, 3022..Beam EndA Axial Force | 2..NX NASTRAN Case 1, 3023..Beam EndB Axial Force | 2..NX NASTRAN Case 1, 3024..Beam EndA Torque | 2..NX NASTRAN Case 1, 3025..Beam EndB Torque |
| 742450 | 0 | 0.955482 | -0.194954 | 0.918873 | -0.31517 | 2.751794 | 2.751794 | -0.432276 | -0.432276 |

“elements” Tab data for element 742450

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Prop ID | Type | Topology | Orientation Node | Orientation Vector | Color | Layer | Formulation | C1 | C2 |
| 742450 | 114..111002\_TUBE\_ASSY\_FSML\_FSMU-1\_ASM | BEAM | Line, 2-noded | 0 | -0.09802, -0.99518, 0. | 124 | 1 | None | 742471 | 742470 |

“properties” Tab data for element 742450

|  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| Title | Type | Shape | Material | Color | Layer | End A Area | Y Shear Area | Z Shear Area | I1 |
| 111002\_TUBE\_ASSY\_FSML\_FSMU-1\_ASM | BEAM | Circular Tube | 143..304 CRES | 110 | 1 | 0.019528 | 0.0106242 | 0.0106242 | 0.0001 |

|  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| I2 | I12 | NS Mass/Len | Warp Const | J | Perimeter | Neutral Axis Off A: Y | Neutral Axis Off A: Z | Neutral Axis Off B: Y | Neutral Axis Off B: Z | Pt 1: Y |
| 1E-04 | 0 | 0 | 0 | 0.0002 | 0.785398 | 0 | 0 | 0 | 0 | 0 |

|  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- |
| Pt 1: Z | Pt 2: Y | Pt 2: Z | Pt 3: Y | Pt 3: Z | Pt 4: Y | Pt 4: Z |
| -0.125 | 0.13 | 0 | 0 | 0.125 | -0.13 | 0 |

“nodes” Tab data for element 742450

|  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |  |
| --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- | --- |
| ID | Color | Layer | Out CSys | Type | Superelement ID | Def CSys | X-Def | Y-Def | Z-Def | Active Csys | X-Act | Y-Act | Z-Act | 1 | 2 | 3 | 4 | 5 | 6 |
| 742470 | 46 | 1 | 0 | 0..Node | 0 | 0 | -4 | 15 | 8 | 0..Global Rectangular | -3.8 | 15 | 8.3 | 0 | 0 | 0 | 0 | 0 | 0 |
| 742471 | 46 | 1 | 0 | 0..Node | 0 | 0 | -4 | 15 | 8 | 0..Global Rectangular | -3.7 | 15 | 8.3 | 0 | 0 | 0 | 0 | 0 | 0 |

“stress” Tab data for element 742450

|  |  |  |  |
| --- | --- | --- | --- |
| Element Data |  | Element Id | 742450 |
|  | point 1 | 742471 |
|  | point 2 | 742470 |
| Nastran Output |  | Beam end A plane 1 moment | 2 |
|  | Beam end A plane 2 moment | 3 |
|  | Beam end B plane 1 moment | 9 |
|  | Beam end B plane 2 moment | 4 |
|  | Beam end A axial force | 10 |
|  | Beam end B axial force | 10 |
|  | Beam end A torque | 6 |
|  | Beam end B torque | 6 |
| Nastran Output (Sustained Loads) |  | Beam end A plane 1 moment, SL | 0 |
|  | Beam end A plane 2 moment, SL | 0 |
|  | Beam end B plane 1 moment, SL | 0 |
|  | Beam end B plane 2 moment, SL | 0 |
|  | Beam end A axial force, SL | 0 |
|  | Beam end B axial force, SL | 0 |
|  | Beam end A torque, SL | 0 |
|  | Beam end B torque, SL | 0 |
| Property |  | Property Card | 114..111002\_TUBE\_ASSY\_FSML\_FSMU-1\_ASM |
| Pressure | (all at 60 for now) | Pressure | 3296 |
| Element Geometry |  | thickness, as designed | 0.027999934 |
|  | r2 = r | 0.125 |
|  | r1 = r-t | 0.097000066 |
|  | Bend Radius | 0.624929939 |
| Material stress allowable |  | S | 16700 |
| Quality Factor |  | E | 0.8 |
| Weld Reduction factor |  | W | 1 |
| Max(Intrados, extrados) |  | I=max(3d, 3e) | 1.125 |
| 304.1.1 |  | Y | 0.4 |
| Allowances | ??? | c | 0 |
| Calculated thickness minimums | 304.1.2, 304.2.1 | thickness, straight(3a) |  |
| 302.3.5 | thickness, straight(3b) |  |
|  | thickness, bends (3c) | 0.031 |
| Stress Intensity Factors, moment indices, force indices | 320.1 | h = t\*R/r | 0.139983979 |
|  | ii = .9/h^(2/3) | 3.33828661 |
|  | Ii = .75 ii or 1, greater of 2 | 2.503714957 |
|  | io = .75/h^(2/3) | 2.781905508 |
|  | Io = .75 io or 1, greater of 2 | 2.086429131 |
|  | It = 1 w/o applicable data | 1 |
|  | Ia = 1 w/o applicable data | 1 |
| Section Modulus |  | Z = pi\*(r2^4-r1^4)/(4\*r2) | 0.000977733 |
| Limit stresses due to sustained loads and displacement strains | 319.3.4 | Sc = allowable at room temp | 25000 |
|  | sh = allowable at max temp | 16700 |
|  | Ec | 0.8 |
|  | Ej | 1 |
|  | f | 1 |
|  | sa(allowable) | 52125 |
| Flexibility Stress | 319.4.4 | Sb(18) | 41253 |
|  | st(17) | 6000 |
|  | sa(17) axial | 1000 |
|  | Se | 43096 |
| Sustained Loads Stress | 320.2 | A | .02 |
|  | Sb | 0 |
|  | St | 0 |
|  | Sa(axial) | 0 |
|  | Sl | 0 |

Starting from the top of the “stress” tab you can see the element ID, 742420, is listed from the “elements” tab. Next the end nodes are listed from the “elements” tab. Next the load values are pulled from the “output” and “outputSl” tab. Next the property card is pulled from the “properties” tab. Next the pressure is listed. The pressure is pulled from the additional input section of the “stress” tab. Next the element geometry is calculated and pulled from the “property” tab.

r2 = .125

A = .02

Equation

r1 = r2 – t = .125-.029 = .096

Bend Radius = .625”

Next the material allowable stress, quality factor, and weld reduction factor are listed. These are pulled from the additional input section of the “stress” tab. Next the larger of the Intrados or extrados bend reduction factor is listed.

Equation

Equation

1.125 is used because it is the largest.

Next the Table 304.1.1 coefficient and mechanical allowance is listed. These are pulled from the additional input section of the “stress” tab. Next the thicknesses are calculated.

Equation

The thickness of the tubes is smaller than the calculated minimum so the cell is colored red.

Next the stress intensity factors, bending stress intensity factors, and axial stress intensity factors are calculated.

Equation

Equation

Equation

Equation

Equation

Ii is the greater of .75\*ii and 1, which is 2.5. Io is the greater of .75\*io and 1, which is also 2.1. It and Ia are always 1 without applicable testing data. Next the section modulus is calculated.

Equation

After this the limit stresses and applicable quality factors are listed. These are all pulled from the additional input section of the “stress” tab. Because the stress due to sustained loads is less than the material allowable stress equation 24 is used to calculate the flexibility allowable stress.

= Equation

Next the flexibility stresses are calculated.

Equation

Equation

Equation

Equation

The flexibility stress is lower than the allowable so the cell is colored green.

Next the sustained load stresses are calculated. The process is the same as it was for calculating flexibility stress however a sustained load case is used. For this example there was no sustained load case so Sl = 0. Sl is lower than the material allowable so the cell is colored green.