Table of Contents

TRIFLEX® Windows User Manual

Introduction to TRIFLEX®Windows .... Chapter 1
Tutorial .... Chapter 2
CHAPTER 3 7
Creating a TRIFLEX Window icon...7
3.1.1 Main Screen Layout 8
3.1.2 Commands for Graphical Operations 10
3.1.3 Accessing Data from Piping Model..11
3.1.4 Using the Manual and Help Command 12
3.2 Opening and Importing Example Piping Model Files 12
3.2.1 Processing a Previously Built Piping Model 14
3.2.2 Printing Output Reports 15
3.2.3 Append, Insert and Replace Mode 16
3.3 Coding a Problem 18
3.3.0 Starting TRIFLEX Windows 18
3.3.1 Setup a Project 18
3.3.2 Setup Input/Output units 21
3.3.3 Setup Modeling Defaults 23
3.3.4 Setup Case Definition Data...29
3.3.5 Occasional Loading Data 45
3.3.6 Mode Shapes and Frequencies 46
3.3.7 Graphic Color Settings..47
3.3.7.1 Graphic Color Settings...47
3.3.8 Graphic Preferences 49
3.3.9 Save Graphic Setting..50
3.3.10 Restore Setting 50

3.4 Coding Anchor Components 50
3.4.1 Discussion of Anchor Component – Type/Location Tab 50
3.4.2 Discussion of Anchor Component – Init. Mvmts and Rotations Tab 53
3.4.3 Discussion of Anchor Component, Pipe Properties Tab 56
3.4.4 Discussion of Anchor Component, Process Tab...............................64
3.4.5 Discussion of Anchor Component, Code Compliance 66
3.4.5.1 B31.1 Code Compliance 68
3.4.5.2 B31.3 Code Compliance and DIN 2413 71
3.4.5.3 B31.4 Code Compliance 74
3.4.5.4 B31.8 Code compliance 76
3.4.5.5 U.S Navy General Specifications for Ships, Section 505 78
3.4.5.6 ASME Section III, Division I (Subsection NC) – Class 2 80
3.4.5.7 ASME Section III, Division I (Subsection ND) – Class 3 82
3.4.5.8 SPC1- Swedish Piping Code (Method 1, Section 9.4) 84
3.4.5.9 SPC2- Swedish Piping Code (Method 2, Section 9.5) 86
3.4.5.10 TBK5-6 - Norwegian General Rules for Piping System (Annex D- Alternative Method) 89
3.4.5.11 TBK5-6 - Norwegian General Rules for Piping System (Section 10.5) 91
3.4.5.12 DNV –Submarine Pipeline System -DnV, 1981 Edition 94
3.4.5.13 DNV –Submarine Pipeline System -DnV, 1996 Edition 96
3.4.5.14 DNV – Offshore Standard OS-F101 Submarine Pipeline System -DnV, 2000 Edition 98
3.4.5.15 Polska Norma PN-79 / M-34033 100
3.4.5.16 SNIP 2.05-06-95 FSU Transmission Piping Code 104
3.4.5.17 Fiberglass Reinforced Plastic Pipe 106
3.4.5.18 UKOOA- UK Offshore Operator Association 114
3.4.5.19 BS8010 Code Compliance 122
3.4.5.20 NPD Guidelines for Submarine Pipelines and Risers 124
3.4.5.21 Statoil Design, Specifications Offshore Pipeline Systems 126
3.4.5.22 European Standard prEN 13480-3 128
3.4.6 Discussion of Anchor Component, Soil Load Tab 130
3.4.7 Discussion of Anchor Component, Wind Loads 135

3.5 Coding Piping Data 139
3.5.1 Coding Piping Data, Piping Data 139
3.5.2 Coding Pipe Data, Restraint Tab 142
3.6 Coding Elbow Data 167
3.6.1 Coding Elbow Data, Elbow Data Tab 167
3.7 Coding Branch Connection, Branch Connection....................................173
3.7.1 Coding Branch Connection, Branch Connection Tab.....................173
3.8 Coding Valve Data, Valve Data Tab 179
3.8.1 Coding Valve Data, Valve Data Tab 179
Welded Valve 183
Flanged Valve 184
3.9 Coding Flange Data 186
3.9.1 Coding Flange Data, Flange Data Tab...186
Single Flange 189
3.10 Coding Reducer Data 191
3.10.1 Coding Reducer Data, Reducer Data Tab 191
3.11 Coding Joint Data..195
3.11.1 Coding Joint Data, Joint Data Tab 195
3.12 Coding Expansion Joint 201
3.12.1 Coding Expansion Joint, Expansion Joint Tab 201
3.13 Release Element 206
3.13.1 Coding Release Element, Release Element Tab 206
3.14 Executing a Static Analysis...210
3.15 View Run Output 212
3.1.6 Printing..216
APPENDIX A..218

TRIFLEX® Windows Theory Manual

Outputs ....Chapter 6
Rotating Equipment Compliance Reports ....Chapter 7
Triflex®Windows Piping Code Compliance Reports ....Chapter 8
Triflex®Windows Dynamic Capabilities ....Chapter 9

Related Engineering Data .... Appendix

List of Figures

Figure 3.1.0-1 Demo IU1.dta Example.. 7
Figure 3.1.1-2 Status Bar Indicators 9
Figure 3.1.2-1 Graphic Toolbar Buttons 10
Figure 3.1.3-1 Viewing Anchor Component Properties 11
Figure 3.1.3-2 Worksheet.. 11
Figure 3.2.0-1 Display of an Imported Model 13
Figure 3.2.1-1 Calculation Log or Dayflies 14
Figure 3.2.2-1 Print Report Preview Options 15
Figure 3.2.2-2 Printing Options 15
Figure 3.3.1-1 Main Screen – Setup Options... 18
Figure 3.3.1-2 Project Data 19
Figure 3.3.2.1 – Input Output Units 21
Figure 3.3.3-3 Main Screen – Setup Options.. 23
Figure 3.3.4 Setup Case Definition.. 29
Figure 3.3.5 - Setup Occasional Loading Data 45
Figure 3.3.6 – Dynamic Data Entry... 46
Figure 3.3.7 - Graphic Color Settings 47
Figure 3.3.7.1 –Color Selection 48
Figure 3.3.8 – Graphical Preferences... 49
Figure 3.4-1 Anchor Components, All Tabs.. 50
Figure 3.4. Node 1000, Anchor Dialog, Initial Mvmt/Rot Tab 53
Figure 3.4.3 - Anchor Component, Pipe Properties Tab.. 56
Figure 3.4.4 – Anchor Component, Process Tab 64
Figure 3.4.5 - Anchor Component, Code Compliance Tab 66
Figure 3.4.5.1 - Anchor Component, Code Compliance Tab, B31.1 68
Figure 3.4.5.2 - Anchor Component, Code Compliance Tab,B31.3 71
Figure 3.4.5.3 - Anchor Component, Code Compliance Tab, B31.4 74
Figure 3.4.5.4 - Anchor Component, Code Compliance Tab, B31.8 76
Figure 3.4.5.5 - Anchor Component, Code Compliance Tab, US Navy 78

Figure 3.4.5.6 - Anchor Component, Code Compliance Tab, Class 2 80
Figure 3.4.5.7 - Anchor Component, Code Compliance Tab, Class 3 82
Figure 3.4.5.8 - Anchor Component, Code Compliance Tab, SPC1 84
Figure 3.4.5.9 - Anchor Component, Code Compliance Tab, SPC2 86
Figure 3.4.5.10 - Anchor Component, Code Compliance Tab, TBK 56 89
Figure 3.4.5.11 - Anchor Component, Code Compliance Tab, TBK 5-6 Method 2 91
Figure 3.4.5.12 - Anchor Component, Code Compliance Tab, DNV 1981 94
Figure 3.4.5.13 - Anchor Component, Code Compliance Tab, DNV 1996 96
Figure 3.4.5.14 - Anchor Component, Code Compliance Tab, DNV 2000 98
Figure 3.4.5.15 - Anchor Component, Code Compliance Tab, Polska Norma 100
Figure 3.4.5.16 - Anchor Component, Code Compliance Tab, SNIP 104
Figure 3.4.5.17.1 – Modeling Default, FRP 106
Figure 3.4.5.17.2 - Anchor Component, Pipe Properties Tab, FRP 108
Figure 3.4.5.17.3 - Anchor Component, Process Tab, FRP 109
Figure 3.4.5.17.4 - Anchor Component, Code Compliance Tab, FRP 112
Figure 3.4.5.18.1 – Modeling Default, FRP 114
Figure 3.4.5.18.2 - Anchor Component, Pipe Properties Tab, FRP 116
Figure 3.4.5.18.3 - Anchor Component, Process Tab, FRP 117
Figure 3.4.5.18.4 - Anchor Component, Code Compliance Tab, FRP 120
Figure 3.4.5.19 - Anchor Component, Code Compliance Tab, BS 8010 122
Figure 3.4.5.20 - Anchor Component, Code Compliance Tab, NPD 124
Figure 3.4.5.21 - Anchor Component, Code Compliance Tab, STOL 126
Figure 3.4.6 – Anchor Component, Soil Load Tab 130
Figure 3.4.7 – Anchor Component, Wind Load Tab 135
Figure 3.5 – Anchor Component, Pipe Properties Tab 139
Figure 3.5.2-2 XYZ and ABC Coordinate Systems 144
Figure 3.6.1 – Coding Elbow Data, Elbow data Tab 167
Figure 3.7.1 Coding Branch Connection, Branch Connection Tab 173
Figure 3.8.1 Coding Valve Data, Valve Data Tab 179
Figure 3.9.1 Coding Flange Data, Flange Data Tab 186
Figure 3.10.1 Coding Reducer Data, Reducer Data Tab 191
Figure3.11.1 Coding Joint Data, Joint Data Tab 195
Figure 3.12.1 Coding Expansion Joint, Expansion Joint Tab 201

Figure 3.11.1 Coding Joint Data, Joint Data Table 206
Figure 3.14.1 Main Screen, Calculate Pull-Down Menu 210
Figure 3.14.2 Main Screen, Calculation Ready/Stop Icon 210
Figure 3.15-1 Output Pull-Down Menus 212
Figure 3.15-2 – Output Report, View results 212
Figure 3.15-3 – Output Report, Type Report Selector 213
Figure 3.15-4 Output Code Compliance Report 214
Figure 3.15-6 – Output Display Dialog 215
Figure 3.15-7 – Output Display Deformed Graphics 215
Figure 3.16-1 – Print Report 216
Figure 3.3.6-2 Report Print Menu 217

CHAPTER 3

Creating a TRIFLEX Window icon

To create a TRIFLEX Windows Icon on your desktop, do the following:

1. Click on the **START** button in the lower left corner of your screen.
2. Highlight **Find** and click on **Files or Folders**.
3. Enter **TriflexWindows.exe** in the ***Named*** field; select all hard drives in the ***Look in*** field and click on Find Now. The default path is:

   C:\Program Files\PipingSolutions\TriflexWindows

4. Right click on the TriflexWindows.exe file name
5. Highlight ***Create Shortcut*** and left click
6. Click **YES** to respond to the Windows Message to place the TRIFLEX Windows Icon on the desktop.

To execute TRIFLEX Windows, double click on the TRIFLEX Windows Icon on the desktop.

To open an Existing Piping Model, click on **FILE** and from the pop-up menu, select **OPEN**. From the path (**c:\ProgramFiles\PipingSolutions \TriflexWindows\Samples\Tutorial01**), open **Tutorial01.DTA** file.

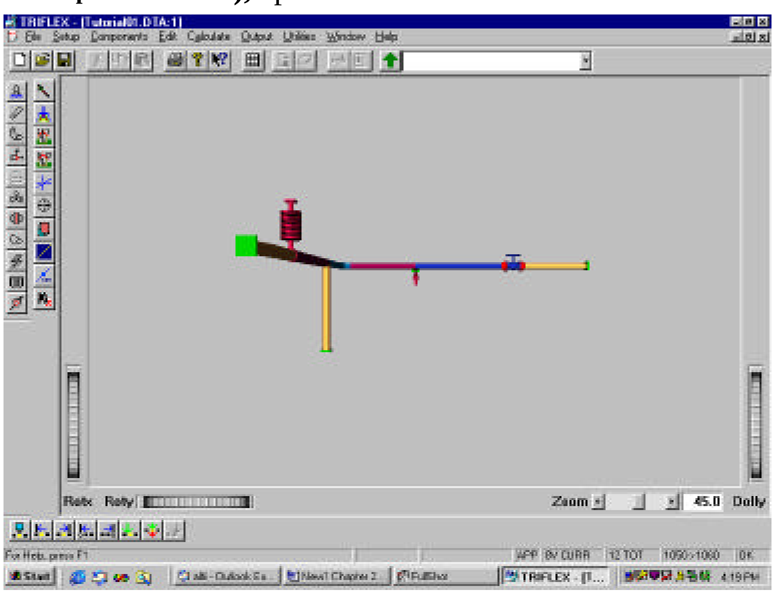
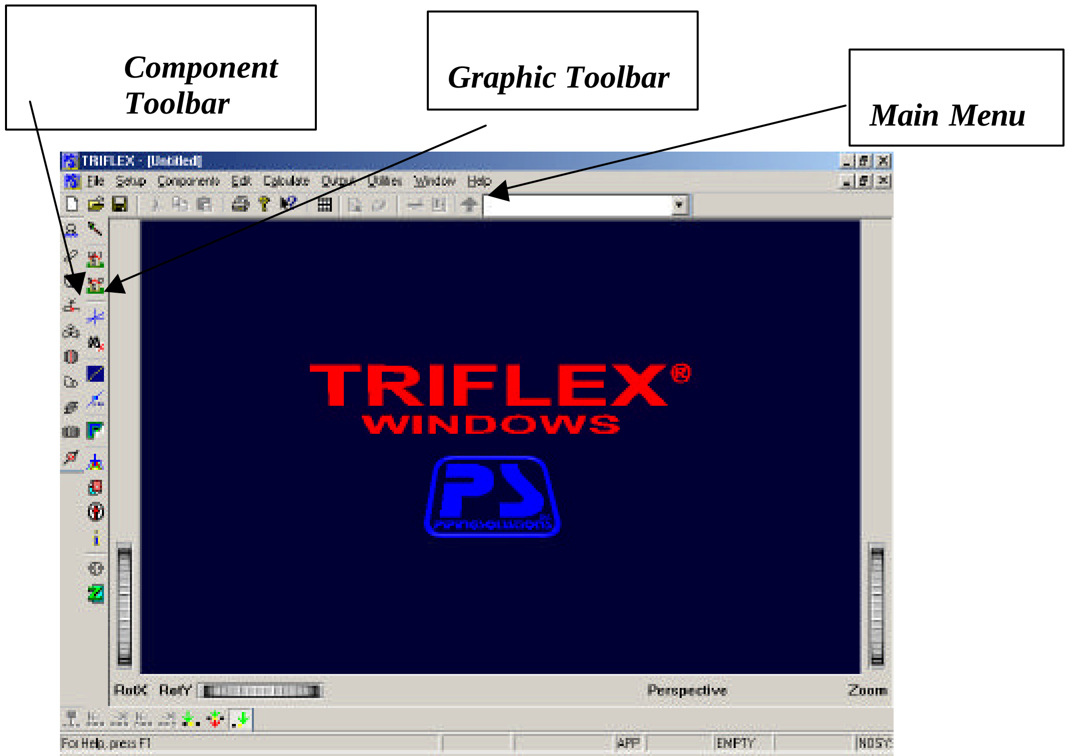


Figure 3.1.0-1 Demo IU1.dta Example

3.1.1 Main Screen Layout

When TRIFLEX is first brought up, the TRIFLEX introduction screen as shown in Figure 3.1.1-1 appears.



The **Component toolbar** buttons are the same as the components listed at the bottom of the Components pop-up menu. To create a component, click on one of the component buttons or select and click on Component on the Main menu, and then highlight the component you wish and click on it.

Thumb-wheels: The window also includes three thumb-wheels labeled **Rotx**, **Roty**, and **Dolly**. At the bottom right of the window is a slider control labeled **Zoom**.

Thumb-wheels & Zoom Slider

There are three wheels on the screen. The two thumb-wheels in the lower left corner: **Rotx, Roty** will rotate the piping system around x-axis and y-axis respectively. There is a third thumb wheel located on the lower right corner. In Orthogonal mode, the thumb wheel will be labeled Zoom and will allow the user to zoom in and out on the model. In Perspective mode, the third thumb-wheel will be labeled Dolly and will enable the user zoom in and out in walk-through style. In Perspective mode, a slider is also provided in the lower right corner to enable the user to zoom in and out.

Note: +y axis is always up (vertical) in a piping model in TRIFLEX.

Toolbars and Menus

On left side of the screen, two Toolbars are provided. The buttons in the left column make up the Components Toolbar. The buttons in the right column make up the Graphic Toolbar.

Along the top of the screen, two rows of the Main Menu are provided. They are similar in style to the standard Microsoft Menu Layout and provide editing facilities, file services, graphic facilities, etc.

Status Bar

This is located on the bottom view of the screen (Figure 3.1.1-2).

Figure 3.1.1-2 Status Bar Indicators

APP - Refers to Append Mode as opposed to INS (Insert) Mode.

EMPTY – Appears when a piping model has not yet been created or loaded.

When a piping model has been created or loaded, the following two items will appear:

3B CURR- Current Component is No. 3 and is a Branch from node **1010** to **1020**.

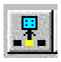
12 TOT- Refers to the piping model having a total of 12 Components.

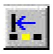
NOSYS - Appears when a piping model has not been created or loaded.

When a piping model has been created or loaded, the following two items will appear to indicate the status of the geometry of the system:

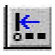
OK- indicates that there is no geometry error.

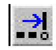
ERR – indicates that there is a geometry error.

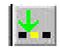
 Edit current component

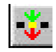
 Previous component

 Next component

 First component

 Last component

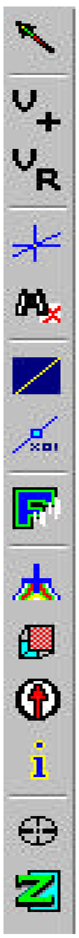
 Insert ahead

 Replace current

 Append following

3.1.2 Commands for Graphical Operations

Execute the following commands to become familiar with the Graphic Toolbar. We suggest you start with the Node Labels icon and work up to the Select/View icon, Figure 3.1.2-1.



Select/View – Arrow used to point at a component and select it / Hand used to move or rotate the piping mode.

Set View – Allows user to define a view of the piping model as the default view

Recall View – Brings desired view on screen

Toggle Axis – Draws X, Y, Z-axis - size and position can be changed

Node Locate – Locates the node specified and places axis at node point

Line/Render – Line or 3D shapes –component colors can be changed

Node Numbers – Turns on/off node numbers of components

Freeze Frame – Turns on/Off components from being drawn

View All - Brings entire piping model into view on screen

Ortho/Perspective – Right angle view or panorama view

Orient View – Orients view in different planes selections

ISO View – sets the model in Isometric view

Zoom Point – Brings user specified point in the piping model closer

Windows Zoom – Zooms in on the window that is specified

Figure 3.1.2-1 Graphic Toolbar Buttons

Note: To Pan – hold down the shift key, click on the object you wish to move and move the object. In order to complete this the user must be in graphics mode (the hand as opposed to the arrow must be showing).

3.1.3 Accessing Data from Piping Model

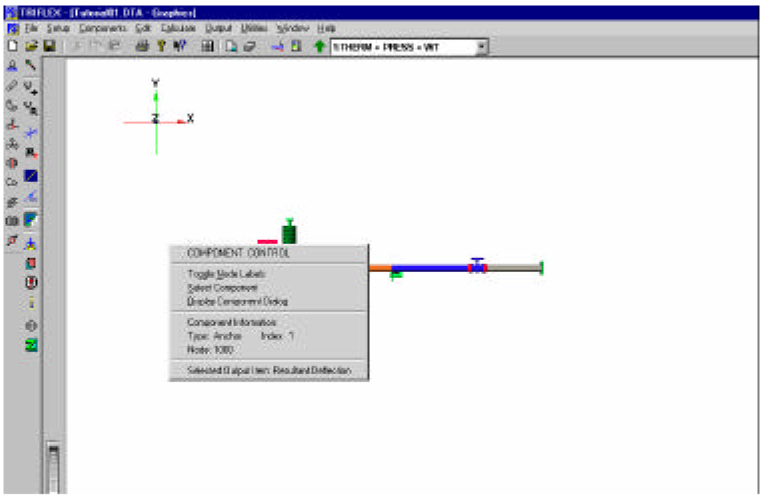


Figure 3.1.3-1 Viewing Anchor Component Properties

To investigate the properties of a piping model, clicking (left mouse button) on the particular component of interest. For instance, clicking on the Anchor will yield a menu such as shown in Figure 3.1.3-1. To modify any property on this component, click on **Display Component Dialog** and enter the desired data in the component dialog from the keyboard. An in-depth discussion can be found in Section 3.0.

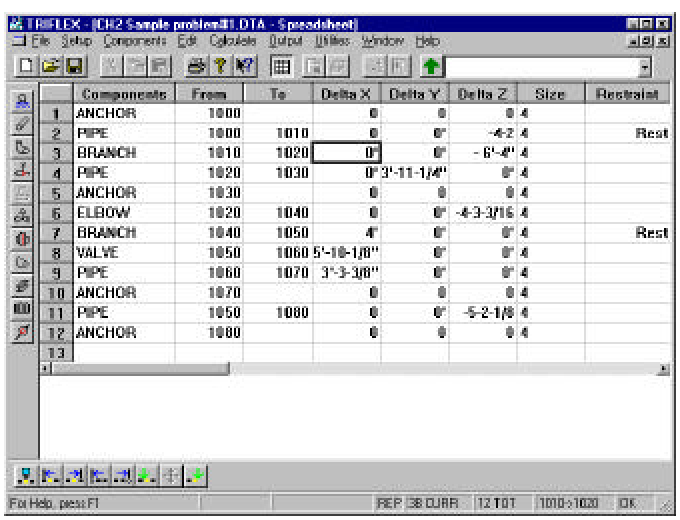


Figure 3.1.3-2 Worksheet

To view entered data for the piping model, including node numbers, delta dimensions, pipe sizes, restraint indicators, pipe material, insulation material, and temperature and pressure for all load cases, click on the component button icon Worksheet, located in the Main Menu. Figure 3.1.3-2. Pressing the Ctrl + Tab keys allows the user to toggle between different screens.

Note: If your Company runs CAD from this system, then check to see what commands is "Hot Keyed".

3.1.4 Using the Manual and Help Command

To access assistance with specific topics; click on **Help** on the Main Menu. **Index** and **User Manual** will then appear. Clicking on **Index** will show a list of topics to select from to obtain more detail about any specific topic listed. Clicking on **User Manual** will show a list of the chapters available for viewing.

The electronic TRIFLEX User's Manual is located in the default directory:

c:\ProgramFiles\PipingSolutions\TriflexWindows\Manual

The manual is furnished electronically in Adobe Acrobat (*.pfd) format and linked by chapter, figures and index. Click on a chapter and the chapter will appear on the screen.

3.2 Opening and Importing Example Piping Model Files

To open a previously created piping model, click on **File** in the **Main Menu,** select option **Open** and then select the file you wish. By default, the extension of TRIFLEX data files is "**.dta**". The complete path is:

c:\ProgramFiles\PipingSolutions
\TriflexWindows\Samples\Tutorial01\Tutorial01.dta

To import a previously created TRIFLEX DOS piping model, click on **Utilities** in the **Main Menu,** select option **Import File** and then click on **DOS Triflex Job**. By default, the extension of DOS TRIFLEX data files is "**.job**". The complete path is:

c:\Program Files\PipingSolutions
\TriflexWindows\Samples\Tutorial01\Tutorial01.job

To display the spreadsheet and the piping model simultaneously on a split screen as shown in Figure 2.2.0-1, open a piping model. The piping model will be displayed on the screen. Click on **Windows** on the **Main Menu** and select **Tile Vertical.** The user will see two windows; one with the piping model and the other will be blank. The user should then click on the Spreadsheet Icon in the Main Menu to obtain the spreadsheet in the blank screen. Click on any component in the piping model and the data for that component will be highlighted in the spreadsheet. Similarly, by clicking on a node in the spreadsheet, the component on the piping model will be highlighted. This is useful in identifying components in a piping model for copying, inserting and deleting.

Note: Models may be built using the spreadsheet and/or in graphic mode as described in section 2.3.0 of this User's Manual.

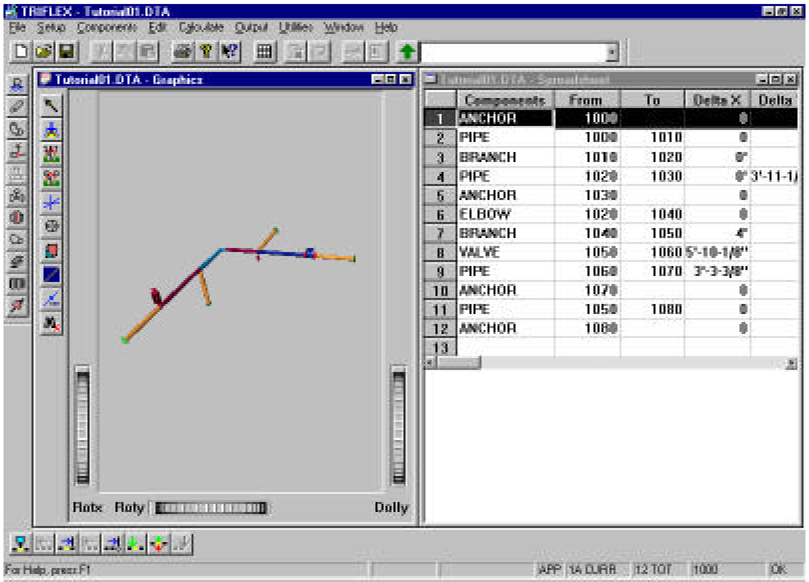
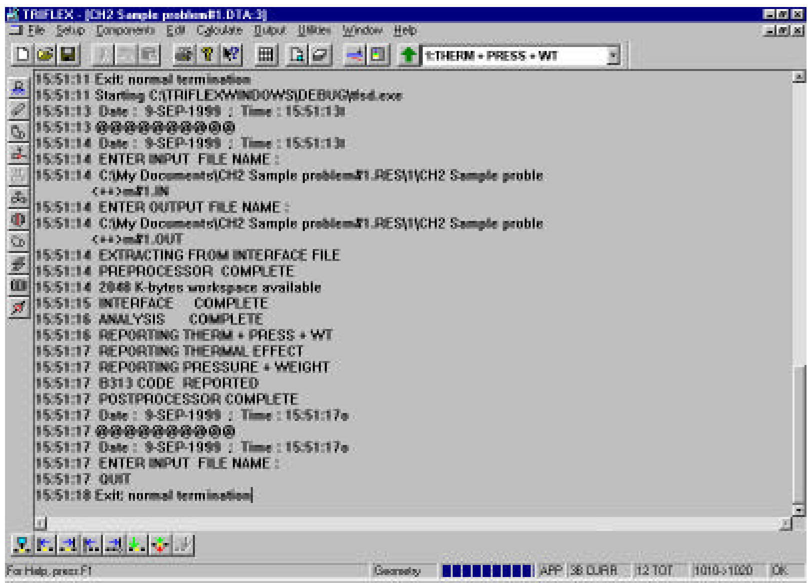


Figure 3.2.0-1 Display of an Imported Model

3.2.1 Processing a Previously Built Piping Model

There are two methods of processing a piping stress analysis. The first is to go to the **Main Menu**, select **Calculate** and then select **Basic Calculation**. The second method is to click on the green arrow on the **Main Toolbar**. Figure 3.2.1-1 depicts the "run time log" or "calculation log" sometimes known as the "Dayflies". While the program is executing or after the program has executed, the user should look for the following terms: <u>*ERROR; QUIT; EXIT:*</u>



<u>*Normal Termination*</u>

Figure 3.2.1-1 Calculation Log or Dayflies

If the last two lines of the dayflies are "**QUIT**" and "**Exit: normal termination**", then TRIFLEX Windows is telling you that the execution was successfully completed. If the word "**ERROR**" appears, then you must examine your input data to find the error and make corrections. Please note that TRIFLEX generates this report in another window for viewing. To return to the piping model, you must delete or minimize this window.

NOTE: If you have imported a DOS TRIFLEX data file, you must re-define the required case data. To do so, Click on **Setup** on the **Main Menu** and then click on the **Case Definition**. The user must enter the desired case data on the dialog provided.

3.2.2 Printing Output Reports

To print output reports, click on **Output** on the **Main Menu** and then click on **Print Reports** on the **Pull down Menu**. The screen in Figure 3.2.2-1 will appear.

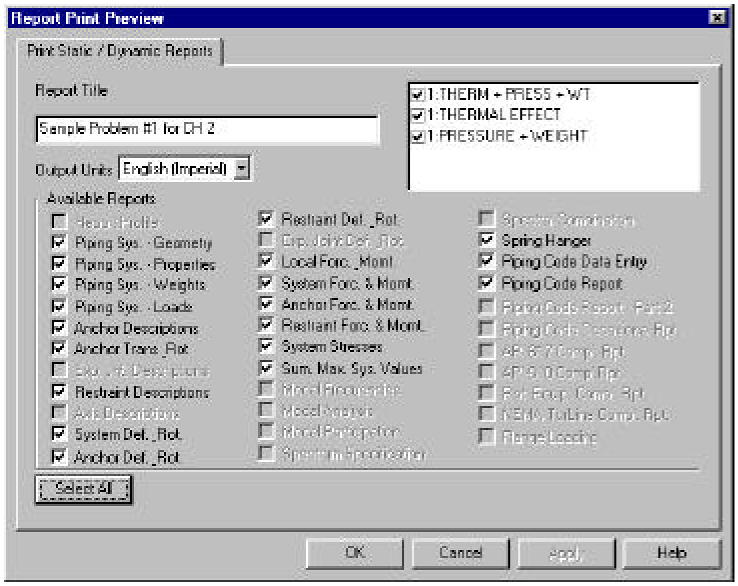


Figure 3.2.2-1 Print Report Preview Options

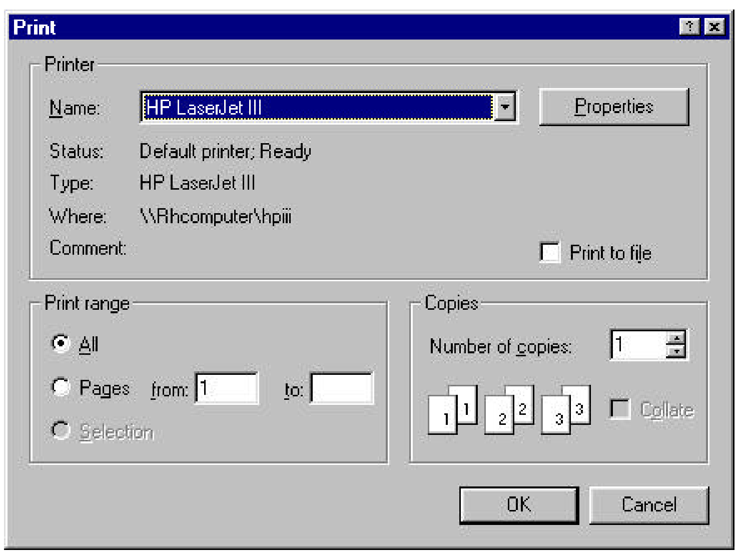


Figure 3.2.2-2 Printing Options

Select the desired load cases and check the reports you wish to review and click the OK button. TRIFLEX will then give you an opportunity to select the printer and printing options as shown in Figure 3.2.2-2 and will then print the reports for you.

3.2.3 Append, Insert and Replace Mode

In order to demonstrate the modification capabilities of TRIFLEX Windows, it is best to either create a short model or refer to Figure 3.1.0-1. TRIFLEX Windows can operate in **APPEND** mode, **INSERT** mode or **REPLACE** mode. To change this mode, click on **Components** on the **Main Menu** and then click on the desired mode - Append, Insert or Replace. Alternatively, the user can click on the icons located in the bottom left corner of Main Screen to change the operating mode. See Figure 3.1.2-2 for an explanation of these Icons.

The three modes for modeling components are as follows: ***Insert*** (creates component prior to highlighted or current component), ***Append*** (creates component following last component in a branch) and ***Replace*** (replaces highlighted or current component). When building a new piping model, the user must be in Append mode. When the user wishes to insert a new component in an existing piping model prior to a highlighted component, the Insert mode should be selected. When the user wishes to replace one highlighted component, the user should select the replace mode. Insert and Replace also are functional for current or last coded components when no component is highlighted. The selected mode will remain the same until the user selects a different mode.

To **Insert** one or more components, do the following:

1. Turn on the node numbers by clicking on the **Node Numbers** Icon on the Graphic Toolbar while viewing the piping model.

2. Highlight the component before which you wish to place a new component. Alternatively, you can select this component on the spreadsheet.

3. Click on the **Insert** Icon in the lower left corner.

4. Select the component you wish to insert from the component toolbar and the desired dialog will appear for you to define the component. Then click ***OK*** or press ***Enter***.

Similarly, to **Append** a component following **the last component** (must be last component of a branch), click on the desired component on the component toolbar and enter the data on the dialog that appears. Then click ***OK*** or press ***Enter***.

To **Replace** a component, do the following: Turn on the node numbers by clicking on the **Node Numbers** Icon on the Graphic Toolbar while viewing the piping model.

1. Highlight the component, which you wish to replace. Alternatively, you can select this component on the spreadsheet.

2. Click on the **Replace** Icon in the lower left corner.

3. Select the new component from the component toolbar. The desired dialog will appear for you to define the component. Then click ***OK*** or press ***Enter***.

Modifying (Delete, Cut, Paste, Copy and Undo)

The following procedures are recommended for graphically modifying components:

Deleting

1. Click on the component(s) to be deleted.

2. Press the **Del (Delete*)*** key.

Cutting (Ctrl + x)

1. Click on the component(s) that are to be cut.

2. Click on **Edit** on the **Main Menu** and click on **Cut.**

Copying (Ctrl + c)

1. Click on the component(s) that are to be copied.

2. Click on **Edit** on the **Main Menu** and click on **Copy.**

Pasting (Ctrl + v) May be used to append one or more components (previously cut or copied components) to the TO node of the highlighted component.

1. Click on the component to which the component(s) are to be pasted.

2. Click on **Edit** on the **Main Menu** and click on **Paste.**

Undo (Ctrl +z) To undo the last operation, click on **Edit** on the **Main Menu** and click on **Undo.**

Note: In order to **PAN** hold down the SHIFT key and left Click on the mouse on the model dragging the chosen area of the model to the center position Appendix A lists Keyboard Control Key.

3.3 Coding a Problem

3.3.0 Starting TRIFLEX Windows

The purpose of this section is to demonstrate the entry of data into the TRIFLEX Windows dialogs and to build a small piping model.

A piping model will be generated using the interactive screen capabilities. This model will illustrate a portion of the TRIFLEX Windows features and will provide a solid basis for utilizing all of the TRIFLEX Windows capabilities.

Begin by double clicking on the TRIFLEX Windows icon on your desktop.

After the logo screen appears for a few seconds, the main screen of TRIFLEX Windows will be displayed.

3.3.1 Setup a Project

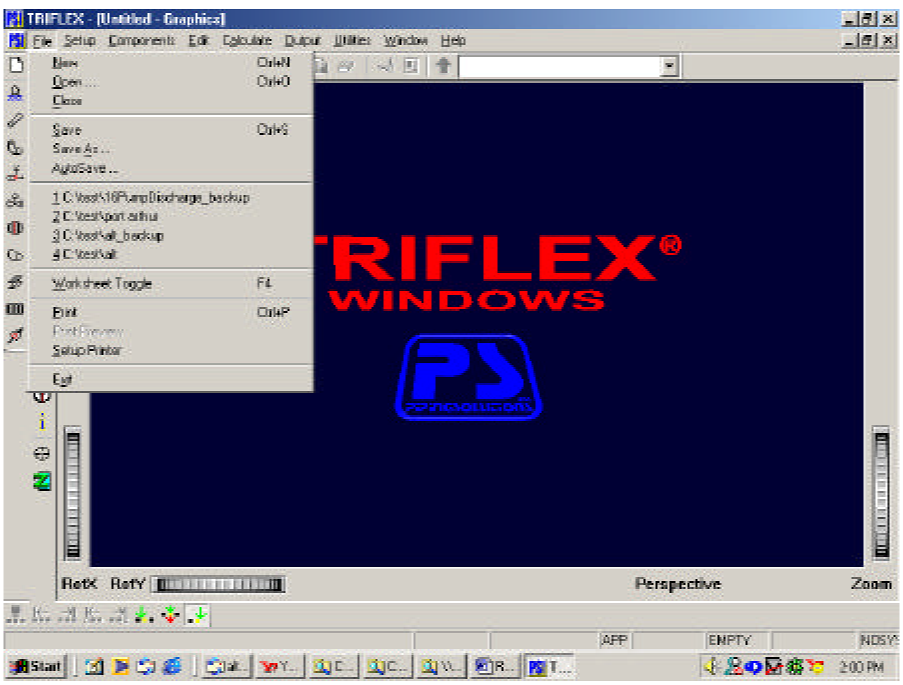


Figure 3.3.1-1 Main Screen – Setup Options

From **Setup** menu, select **Project** as shown in Figure 3.3.1-1, complete the fields to define Project Name, Project Account No., Project Cost Code, Engineer's Name/Initials, etc., as shown in Figure 3.3.1-2. These fields are not mandatory to execute an analysis the above model. A detail discussion appears below.

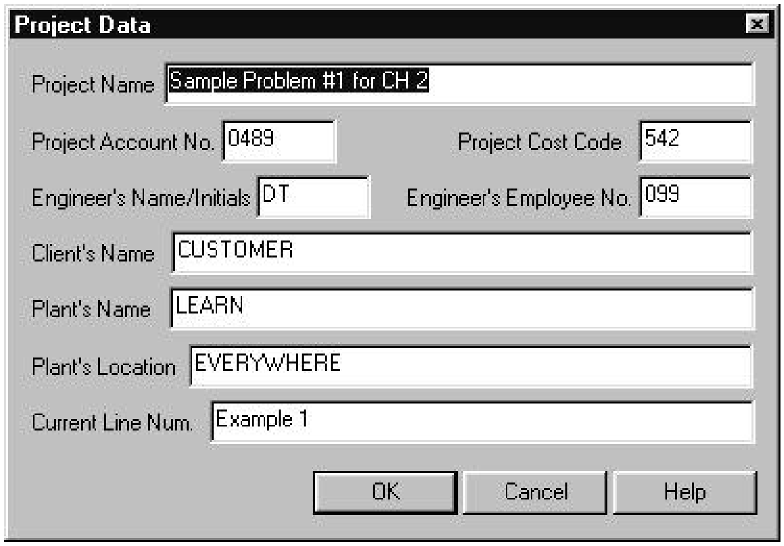


Figure 3.3.1-2 Project Data

The first step in creating a new piping model data file is to provide TRIFLEX with descriptive information about the piping system being modeled. To access the Project Data dialog, the User must click on Setup on the main menu and then Project on the drop down combo list. A Project dialog will then be presented to the User. Enter the data as noted below:

Project – In this field, the User should enter a descriptive title for the piping system being modeled. This line of title will appear at the top of every page of output.

Project Account No. – In this field, the User may enter a project account number, if desired.

Project Cost Code – In this field, the User may enter a project cost code, if desired.

Engineer's Name / Initials – In this field, the User may enter his or her initials.

Engineer's Employee No. – In this field, the User may enter his or her employee number.

Client's Name - In this field, the User may enter the name of the client for whom the work is being performed.

Plant's Name - In this field, the User may enter the name of the plant at which the piping system being analyzed is located.

Plant's Location - In this field, the User may enter the location of the plant where the piping system is being analyzed.

Current Line Num. - In this field, the User may enter the line number of the piping system being analyzed.

3.3.2 Setup Input/Output units

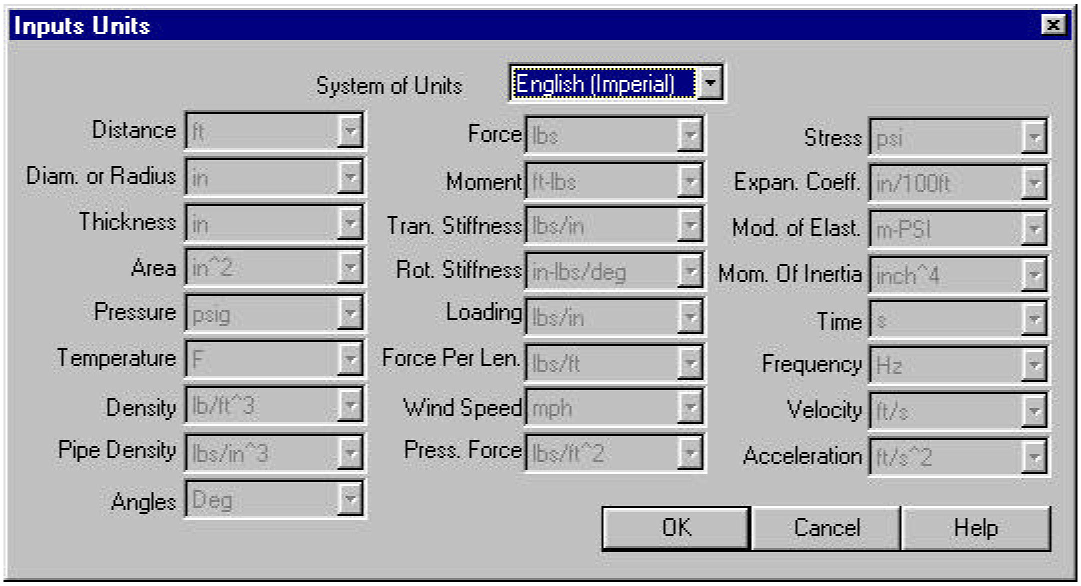


Figure 3.3.2.1 – Input Output Units

The next step in creating a new piping model data file is to select the system of units that will be used throughout the piping model to define the piping system and all related data. The systems of units may not be changed once the piping model is started. To access the Input Units dialog, the User must click on Setup on the main menu and then Input Units on the drop down combo list. An Input Units dialog will then be presented to the User. Enter the data as noted below:

System of Units – In this field, a drop down combo list is provided for the User to click on and select the desired system of units. Four systems of units are available:

1. English (Imperial)
2. SI Metric
3. MKS Metric
4. IU1 Metric

When the User selects the desired system of units, the screen immediately below the drop down combo list will display the variables in the calculations performed by TRIFLEX and the units that TRIFLEX will use for each. To see the units used by TRIFLEX for each variable, simply select the system of units and then see the desired field for the units that will be used by TRIFLEX for the specific variable.

3.3.3 Setup Modeling Defaults

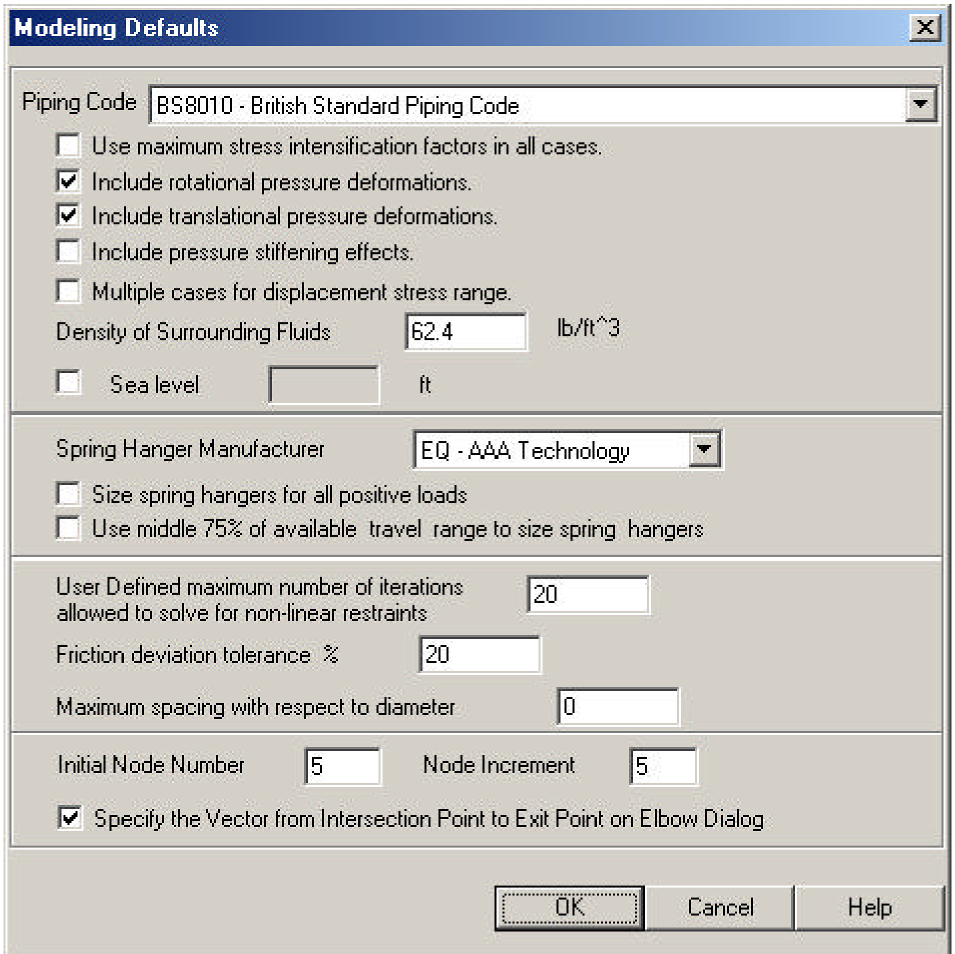


Figure 3.3.3-3 Main Screen – Setup Options

The next step in creating a new piping model data file is to define the modeling defaults that will be used throughout the piping model as the User defines it. If the modeling defaults are changed after the initial modeling has been completed, then the new values will be applied on all cases processed after the defaults were changed. To access the Modeling Defaults dialog, the User must click on Setup on the main menu and then Modeling Defaults on the drop down combo list. A Modeling Defaults dialog will then be presented to the User. Enter the data as noted below:

Piping Code – TRIFLEX contains the guidelines and rules for computing the deflections, rotations, forces, moments and stresses in a piping system based upon a number of different piping codes. Each piping code has its own unique rules for performing the calculations to determine if a piping system meets the code requirements. For instance, stress intensification and flexibility factors are calculated by TRIFLEX in accordance with the piping code selected by the User and applied at bends, miters, reducers and branch connections through out the piping system. A piping code must be selected by the User from the drop down combo list in this field, even if a code compliance analysis is not requested. The piping codes currently included in TRIFLEX Windows are as follows:

B31.1 - ASME Power Piping Code

B31.3 - ASME Process Piping Code

B31.4 - ASME Pipeline Transportation Systems for Liquid Hydrocarbons and

Other Liquids Code

B31.8 - (DOT Guidelines) ASME Gas Transmission & Distribution Systems Code

U.S. Navy - General Specifications for Ships, Section 505

Class 2 – ASME Section III, Subsection NC Code

Class 3 – ASME Section III, Subsection ND Code

TBK51 - Norwegian General Rules for Piping Systems (Annex D - Alternative

Method)

TBK52 - Norwegian General Rules for Piping Systems (Section 10.5)

SPC1 - Swedish Piping Code (Method 1 - Section 9.4)

SPC2 - Swedish Piping Code (Method 2 - Section 9.5)

DNV - DnV **(**Det Norske Veritas) Rules for Submarine Pipeline Systems, 1981 & 1996

POL1 – Polska Norma PN-79 / M34033 Steam and Water Piping

Additional piping codes will be incorporated in TRIFLEX Windows in the near future.

Use Maximum Stress Intensification Factors in all Cases - When a check is placed in this check box, TRIFLEX will apply the larger of the in plane and out-of-plane stress intensification factors for each node point in the analysis. This also applies to any Code Compliance Analysis calculations requested. The default is with the check box unchecked.

Include rotational pressure deformations - When a check is placed in this check box, TRIFLEX will consider the effect of internal pressure on the elbow or bend to cause the elbow to rotate to an angle that is greater than the installed angle. In essence, the elbow or bend opens up because of the effect of internal pressure. The default is with a check in the check box.

Include translational pressure deformations - When a check is placed in this check box, TRIFLEX will consider the effect of internal pressure on the pipe to cause the pipe to elongate or lengthen. The default is with a check in the check box.

Include Pressure Stiffening Effects - When a check is placed in this check box, TRIFLEX will consider, in accordance with the appropriate Piping Code, the stiffening effect of internal pressure on bends and elbows. The default is with the check box unchecked.

Multiple Cases for Displacement Stress Range - When a check is placed in this check box, TRIFLEX will calculate the thermal stress range by computing thermal loads in two or more operating analyses in combination and then using the largest stress range between all load sets as the thermal stress range for the code compliance calculations. The default is with the check box unchecked.

Density of Surrounding Fluid - This field is provided to enable a user to enter the density of the fluid surrounding the pipe when buoyancy effects are to be considered. The default is with this field blank as most piping stress analyses are of piping systems in an open-air environment.

Spring Hanger Manufacturer - If spring hangers are to be sized and selected in this piping stress analysis, then the manufacturer of the spring hangers from whose product line the selection and sizing is to be based must be selected by the User from the drop down combo list in this field. When one of the following vendors is selected, TRIFLEX will choose the proper size and series spring hangers from the selection available from the selected vendor. The selection of the required hangers will be based upon the load being carried and the required installed to operating travel as determined by TRIFLEX. Spring hangers from the following manufacturers are available in TRIFLEX:

Basic Engineers

Bergen & Paterson

Bergen - Power Piping

Comet Support Springs

Carpenter & Paterson

Equal (AAA Technology)

Flexider (Table 5)

Flexider (Table 6)

Flexider (Table 5, revised)

Grinnell

Inoflex

Lisega

Nordon

NPS

Stalowa Wola

Size Spring Hangers for all positive loads - To size spring hangers, TRIFLEX will first perform a **Weight Analysis** to determine the loads at each support point where the spring hangers are to be sized. When a check is placed in this check box and the support loads in the Weight Analysis are found to be positive (greater than zero), TRIFLEX will proceed with the **Operating Case Analysis** to determine the required support movements. For a detailed explanation of the procedure used by TRIFLEX to properly size spring hangers, see Section 5 of this user's manual.

When this check box is left unchecked and the support loads in the Weight Analysis are found to be less than fifty pounds (50), TRIFLEX will not proceed with the **Operating Case Analysis** to determine the required support movements. TRIFLEX will stop with the results of the Weight Analysis and will allow the user to make the necessary spring support decisions. Note that some spring hanger manufacturers do not supply spring hangers to carry loads less than 50 pounds. The default assumption made by TRIFLEX is with the check box unchecked.

Use middle 75% of available travel range to size spring hangers - When the user places a check mark in this check box, TRIFLEX will eliminate twelve and one half percent from the top of the working range and the same twelve and one half percent from the bottom of the working range. Then, TRIFLEX will size the spring hangers using the resultant seventy-five percent of the working range as shown on the Spring Hanger Size and Series Selection Table. Using this process, the user will typically get a more conservative spring hanger for the application. The default assumption is with the check box unchecked.

User Defined maximum number of iterations allowed to solve for non-linear restraints - In this field, the User may specify the maximum number of iterations to be allowed for TRIFLEX to converge on a solution. When using non-linear restraints in a piping system (one-way restraints, limit stops, soil properties, or when considering friction), TRIFLEX must iterate to find the resulting restraint action. The program defaults to a maximum of ten (10) iterations unless the desired number of iterations is entered in this field.

When coding jobs with soil parameters, it is recommended that the number of iterations be increased. Increasing the number of iterations does not significantly increase the time used to perform the analysis. It only increases the possibility of a more accurate solution. With a higher number of iterations specified, TRIFLEX will be allowed to iterate more times when trying to converge on a solution. For typical piping system analysis, twenty (20) iterations is generally more than enough to allow TRIFLEX to converge on a reasonably accurate solution.

Friction deviation tolerance (Percent) - The frictional force computed by TRIFLEX must satisfy the following conditions:

1. If the displacement is zero (or negligible), then the frictional force must be less than or equal to the limit.

2. Else, if sliding occurs, then the frictional force must be equal to the limit.

Coulomb as the product of the normal reaction, multiplied by the coefficient of friction, has established the above-mentioned “limit”.

While checking for condition (2) above, TRIFLEX allows for a tolerance. That is, if the User specifies a tolerance of 20%, any frictional force in the range of 0.8 to 1.2 times the limit will be accepted by TRIFLEX in the calculations.

Considering friction in a piping stress analysis is not an exact science. Specifying a very low friction deviation tolerance is generally not recommended unless the piping stress analyst has specific engineering data to support such assumptions.

Max. Spacing with Respect to Diameter - In this field, the User may specify the default maximum spacing between consecutive node points as a multiple of the nominal pipe diameter. When the Number of Intermediate Node Points or the Maximum Spacing between Node Points is specified on node point dialogs, such entries will override the value entered in this field. The Maximum Spacing with Respect to Diameter on this dialog defaults to zero (no automatic spacing of node points will be imposed) except when soil properties have been specified. Then, the Maximum Spacing with Respect to Diameter on this dialog defaults to three (3) nominal pipe diameters.

TRIFLEX will add the newly created node points to the input and use node point numbers that were generated by adding one to the previous **To** node point number for each node point to be generated. In cases where many node points will be generated by TRIFLEX, it is strongly recommended that the increment between node point numbers coded by the User be sufficiently large so that the generated node point numbers will not be numerically larger than the following To node points.

This feature is also very useful when building a piping model for dynamic (Modal) analysis. By stating the diameter spacing with respect to the nominal diameter, TRIFLEX will create intermediate node points at a spacing no greater than this value times the nominal pipe diameter. For instance, if a value of 7 is specified as the spacing with respect to the nominal pipe diameter and an 8-inch nominal pipe diameter is specified, there will be a maximum distance between node points of 56 inches.

Initial Node Number - In this field, the User may specify the desired node number for the first node point. TRIFLEX will default to an initial node number of “5”.

Node Increment - In this field, the User may specify a number that TRIFLEX is to add to the previous To node point to determine the next To node point number. It is highly recommended that an increment of at least three, if not five, be specified. TRIFLEX will default to an increment of 5, if the User does not enter a value in this field. When building a piping system with numerous soil restraints, it is recommended that a node increment of 30 or 40 be used.

3.3.4 Setup Case Definition Data

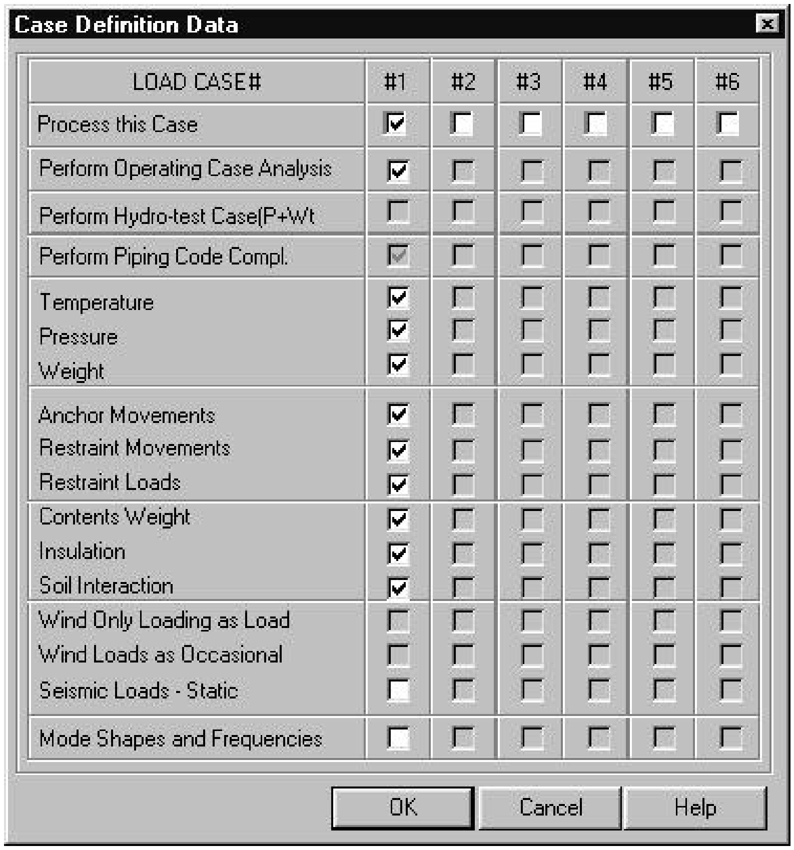


Figure 3.3.4 Setup Case Definition

In order for TRIFLEX to perform a piping flexibility analysis properly, the User must define what analyses are to be performed and what conditions and data are to be considered. To enter the required data, the User must click on Setup on the Main Menu and then on Case Definition on the drop down list. Upon clicking Case Definition, a Case Definition Data dialog will be presented to the User.

In the left column, the conditions and data that can be considered are listed. To the right of the left most columns, six columns of check boxes are provided for the User to define the conditions and data that are to be considered in each case. The conditions and data that may be considered in case number one can be checked in the number one column of check boxes. The conditions and data that may be considered in case number two can be checked in the number two column of check boxes, etc. The conditions and data that the User can instruct TRIFLEX to consider on a case-by-case basis are defined below:

Process this Case - When a check mark is placed in this check box, TRIFLEX will execute the analysis or analyzes specified with the load conditions selected in the column of check boxes directly below this check box. When this check box is left blank, no analysis will be processed for this load case. The default will be with the check box left blank.

Perform Operating Case Analysis - When a check mark is placed in this check box, TRIFLEX will set up the options to process an operating case analysis when the execute command is pressed after the piping model is built. When a check is placed in this check box, TRIFLEX automatically places a check in the following check boxes:

- ❑ Perform Hydro-test Case Analysis – Gray out
- ❑ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired
- ✓ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ✓ Anchor Movements Included; the check mark can be removed by the User.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ❑ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – the User will be allowed to check this box, if desired

Note: The User can only select one of the two Wind options – not both.

- ❑ Wind Loading as Occasional Load – the User will be allowed to check this box, if desired

- ❑ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if desired. The User may not select Seismic Loads if Wind loads have been selected.

- ❑ Mode Shapes and Frequencies – Gray out

TRIFLEX automatically grays out the fields that the User is not permitted to select. The User may select any of the conditions listed above so long as the program does not gray the check boxes out. Check boxes will be grayed out by TRIFLEX when a check mark in such a check box would be contrary to another option selected by the User or to a default set within TRIFLEX. When the Perform Operating Case Analysis check box is left blank, no operating case analysis will be processed for this load case. The default will be with the check box left blank.

- ❑ **Perform Hydro-test Case (P+Wt w/1.0)** – By placing a check in this field, the User instructs TRIFLEX to process a pressure plus weight analysis with the piping system filled with water and the temperature set to 70 degrees F or 21 degrees C. The check boxes with and without checks will be as follows:

- ❑ Perform Operating Case Analysis – Gray out

- ✓ Perform Hydro-test Case Analysis

- ❑ Perform Piping Code Compliance Analysis – Gray out

- ❑ Temperature – Gray out - Returns active again if check in hydro-test Case is removed.

- ✓ Pressure (User can't remove check mark) Gray out - Returns active again if check in hydro-test Case is removed.

- ✓ Pipe Weight (User can't remove check mark) Gray out - Make active again if check in hydro-test Case is removed.

- ✓ Contents Weight Included (User can't remove check mark) Gray out - Returns active again if check in hydro-test Case is removed.

- ❑ Insulation Weight Included; the user will be allowed to check this box, if desired

- ❑ Anchor Movements Included; the user will be allowed to check this box, if desired

- ❑ Restraint Movements Included; the user will be allowed to check this box, if desired

- ❑ Restraint Loads Included; check mark can be removed by the User.
- ❑ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – Gray out
- ❑ Wind Loading as Occasional Load – Gray out
- ❑ Seismic Loads – Static Equivalent – Gray out
- ❑ Mode Shapes and Frequencies – Gray out

In all cases above where the fields are defined as "grayed out", the User will be prohibited from entering a check mark in these check boxes. Where it is stated that the User will be allowed to place a check the check box, if desired, the User will be allowed to do so.

Perform Piping Code Compliance Analyses – By placing a check in this check box, the User instructs TRIFLEX to process the analyses necessary for TRIFLEX to generate a code compliance report. This is a pre-packaged combination of runs combined according to a pre-prescribed set of rules within TRIFLEX. The User can process only the analyses required for a code compliance analysis or the User can process an operating case analysis as well as the analyses required for a code compliance analysis. The check box combinations for each of these two scenarios are as follows:

With No Operating Case Analysis

- ❑ Perform Operating Case Analysis – the User will be allowed to check this box, if desired
- ❑ Perform Hydro-test Case Analysis – Gray out
- ✓ Perform Piping Code Compliance Analysis
- ❑ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.
- ❑ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.
- ❑ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.

- ✓ Anchor Movements Included; the check mark can be removed by the User.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ❑ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – the User will be allowed to check this box, if desired

The User can only select one of the two Wind options – not both.

- ❑ Wind Loading as Occasional Load – the User will be allowed to check this box, if desired
- ❑ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if desired. The User may not select Seismic Loads if Wind loads have been selected.
- ❑ Mode Shapes and Frequencies – Gray out

With Operating Case Analysis

- ✓ Perform Operating Case Analysis – the User will be allowed to check this box, if desired
- ❑ Perform Hydro-test Case Analysis – Gray out
- ✓ Perform Piping Code Compliance Analysis
- ✓ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ✓ Anchor Movements Included; the check mark can be removed by the User.

- ✓ Restraint Movements Included; check mark can be removed by the User.

- ✓ Restraint Loads Included; check mark can be removed by the User.

- ❑ Soil Interaction Included – the User will be allowed to check this box, if desired

- ❑ Wind included with Weight – the User will be allowed to check this box, if desired

The User can only select one of the two Wind options – not both.

- ❑ Wind Loading as Occasional Load – the User will be allowed to check this box, if desired

- ❑ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if desired. The User may not select Seismic Loads if Wind loads have been selected.

- ❑ Mode Shapes and Frequencies – Gray out

In all cases above where the check boxes are defined as "grayed out", the User will be prohibited from entering a check mark in these check boxes. Where it is stated that the User will be allowed to place a check mark in the check box, if desired, the User will be allowed to do so. In addition, the User is allowed to remove any of the check marks that the program generates automatically as a result of the selection of the pre-packaged "Perform Piping Code Compliance Analysis" option.

Temperature – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of a change in temperature. This field will automatically be checked when the User checks the Perform Operating Case Analysis option. When this option has been selected, the check mark will appear in this check box and will be grayed out.

When the User places a check mark in the Perform Hydro-test Case check box, this field will be made blank and will be grayed out.

When the User places a check mark in the Perform Piping Code Compliance Analysis check box, this field will be made blank and will be grayed out. However, when the User places a check mark in the Perform Piping Code Compliance Analysis check box and in the Perform Operating Case Analysis check box, a check mark will appear in this check box and will be grayed out.

If any of the pre-packaged analyses have been selected, then the User has no option to place a check mark in the Temperature, Pressure or Pipe Weight fields.

When the User places a check mark in this field, mode shapes and frequencies may not be selected. Therefore, the mode shapes and frequencies check box will be grayed out.

A check mark may be placed in this check box by the User to request that TRIFLEX process an analysis considering the effects of temperature change only. In addition, the User may place a check mark in this check box as well as either or both of the check boxes labeled Pressure and Pipe Weight. In such a case, the User can specify a Temperature plus Pressure analysis by placing a check mark in the Temperature and Pressure check boxes, or the User can specify a Temperature plus Pipe Weight analysis by placing a check mark in the Temperature and Pipe Weight check boxes, or the User can specify a Temperature plus Pressure plus Pipe Weight analysis by placing a check mark in the Temperature, Pressure and Pipe Weight check boxes

Pressure – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of a change in pressure. This field will automatically be checked when the User checks any of the following options: Perform Operating Case and Perform Hydro-test Case. When these options have been selected, the check mark will appear in this check box and will be grayed out.

When the User places a check mark in this field, mode shapes and frequencies may not be selected. Therefore, the mode shapes and frequencies check box will be grayed out. See the last paragraph under Temperature for combinations of Temperature, Pressure and Pipe Weight for a discussion of combination loadings.

Pipe Weight – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of weight. This field will automatically be checked when the User checks any of the following options: Perform Operating Case and Perform Hydro-test Case. When these options have been selected, the check mark will appear in this check box and will be grayed out.

When the User places a check mark in this field, mode shapes and frequencies may not be selected. Therefore, the mode shapes and frequencies check box will be grayed out. See the last paragraph under Temperature for combinations of Temperature, Pressure and Pipe Weight for a discussion of combination loadings.

Contents Weight - In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of the weight of the contents in the piping. This check box can be checked along with any other option including the mode shapes and frequencies so long as pipe weight is also checked. In other words, the user cannot specify contents weight to be considered without specifying that pipe weight be part of the piping analysis.

Insulation Weight - In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of the weight of the insulation on the piping. This check box can be checked along with any other option including the mode shapes and frequencies so long as pipe weight is also checked. In other words, the user cannot specify insulation weight to be considered without specifying that pipe weight be part of the piping analysis.

Anchor Movements - In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of anchor movements. A check mark is automatically placed in this check box when the User checks either of the following options: Perform Operating Case and Perform Piping Code Compliance. This check box is left unchecked for all other cases, but the User can place a check in this check box in combination with all other check boxes except the Mode Shapes and Frequencies check box. When the Mode Shapes and Frequencies check box has been checked, the check mark in this field should be removed.

Restraint Movements - In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of restraint movements. A check mark is automatically placed in this check box when the User checks either of the following options: Perform Operating Case and Perform Piping Code Compliance. This check box is left unchecked for all other cases, but the User can place a check in this check box in combination with all other check boxes except the Mode Shapes and Frequencies check box. When the Mode Shapes and Frequencies check box has been checked, the check mark in this field should be removed.

Restraint Loads - In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of restraint loads. A check mark is automatically placed in this check box when the User checks any of the following options: Perform Operating Case, Perform Hydro-test Case and Perform Piping Code Compliance. This check box is left unchecked for all other cases, but the User can place a check in this check box in combination with all other check boxes except the Mode Shapes and Frequencies check box. When the Mode Shapes and Frequencies check box has been checked, the check mark in this field should be removed.

Soil Interaction – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of the soil springs generated by TRIFLEX as a result of the User's soil specification data. This check box can be checked along with any other option other than the mode shapes and frequencies.

With Operating Case Analysis specified

- ✓ Perform Operating Case Analysis – , you must pass Oper or T1+P1+W to the calculator
- ❑ Perform Hydro-test Case Analysis – Gray out
- ❑ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired
- ✓ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.

- ✓ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ✓ Anchor Movements Included; the check mark can be removed by the User.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ✓ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – Gray out
- ❑ Wind Loading as Occasional Load – Gray out
- ❑ Seismic Loads – Static Equivalent – Gray out
- ❑ Mode Shapes and Frequencies – Gray out

With Operating Case Analysis with Piping Code Compliance Analysis

- ✓ Perform Operating Case Analysis
- ❑ Perform Hydro-test Case Analysis – Gray out
- ✓ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired
- ✓ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.

- ✓ Anchor Movements Included; the check mark can be removed by the User.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ✓ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – Gray out – Wind cannot be considered if Soil is.
- ❑ Wind Loading as Occasional Load – Gray out – Wind cannot be considered if Soil is.
- ❑ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if piping code is selected.
- ❑ Mode Shapes and Frequencies – Gray out

With Piping Code Compliance Analysis – No Operating Case Analysis

- ❑ Perform Operating Case Analysis
- ❑ Perform Hydro-test Case Analysis – Gray out
- ✓ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired
- ❑ Temperature – Gray out
- ❑ Pressure – Gray out
- ❑ Pipe Weight – Gray out
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ✓ Anchor Movements Included; the check mark can be removed by the User.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ✓ Soil Interaction Included – the User will be allowed to check this box, if desired

- ❑ Wind included with Weight – Gray out – Wind cannot be considered if Soil is.

- ❑ Wind Loading as Occasional Load – Gray out – Wind cannot be considered if Soil is.

- ❑ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if piping code is selected.

- ❑ Mode Shapes and Frequencies – Gray out

Wind included with Weight – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of wind loads in conjunction with the pipe weight. If a check mark is placed in this check box, the User may specify any of the following prepackaged analyses: Perform Operating Case Analysis or Perform Hydro-test Case Analysis or Perform Piping Code Compliance Analysis.

For piping code compliance analyses where only an operating case analysis is processed and results are compared with code allowable, the User is encouraged to select this option.

In addition, when the Wind included with Weight is checked, the Load Case options may be elected as follows:

- ✓ Perform Operating Case Analysis – User may place a check mark in this field.

- ❑ Perform Hydro-test Case Analysis – If Perform Operating Case Analysis or if Perform Piping Code Compliance Analysis is checked, this check box may not be checked.

- ✓ Perform Piping Code Compliance Analysis – User may place a check mark in this field.

- ✓ Temperature – User may place a check mark in this field if no prepackaged options are checked.

- ✓ Pressure – User may place a check mark in this field if no prepackaged options are checked.

- ✓ Pipe Weight – User must place a check mark in this field if no prepackaged options are checked.

  Contents Weight Included - User may place a check mark in this field

  Insulation Weight Included; - User may place a check mark in this field

  Anchor Movements Included; - User may place a check mark in this field

Restraint Movements Included - User may place a check mark in this field

Restraint Loads Included - User may place a check mark in this field

Soil Interaction Included – the User will be allowed to check this box, if desired

Wind included with Weight

❑ Wind Loading as Occasional Load – Gray out.

❑ Seismic Loads – Gray out.

❑ Mode Shapes and Frequencies – Gray out

Wind Loads as an Occasional Load – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of wind loads as an occasional load. This can only be selected when a user has requested a Piping Code Compliance analysis with or without an Operating Case Analysis. If this check box is checked, the User may not specify any of the following analyses (Perform Hydro-test Case Analysis.

In addition, the when Wind Loads as an Occasional Load is checked, the Load Case options are as follows:

✓ Perform Operating Case Analysis – User may place a check mark in this field

❑ Perform Hydro-test Case Analysis – Gray out

✓ Perform Piping Code Compliance Analysis – User must place a check mark in this field

Temperature – User may place a check mark in this field if no prepackaged options are checked.

Pressure – User may place a check mark in this field if no prepackaged options are checked.

✓ Pipe Weight – User must place a check mark in this field or in the Operating case Analysis or in the Piping Code Compliance Analysis

Contents Weight Included - User may place a check mark in this field

Insulation Weight Included; - User may place a check mark in this field

Anchor Movements Included; - User may place a check mark in this field

Restraint Movements Included - User may place a check mark in this field

Restraint Loads Included - User may place a check mark in this field

Soil Interaction Included – the User will be allowed to check this box, if desired

❑ Wind included with Weight – Gray out.

✓ Wind Loading as Occasional Load

❑ Seismic Loads – Gray out.

❑ Mode Shapes and Frequencies – Gray out

Static Equivalent Seismic Loading – In this field, the User may place a check mark to instruct TRIFLEX to consider the effects of User Specified percentages of gravity along the X, Y and Z-axes as an occasional load. By checking this check box, the User instructs TRIFLEX to apply the gravity loading multiplied by the factors entered by the User on the occasional loading dialog. To find this dialog, click on "Setup" and then "Occasional Loading". This check box can only be selected when the User has checked the Piping Code Compliance Analysis.

With Operating Case Analysis specified

✓ Perform Operating Case Analysis – , you must pass Oper or T1+P1+W to the calculator

❑ Perform Hydro-test Case Analysis – Gray out

❑ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired

✓ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.

✓ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.

✓ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.

✓ Contents Weight Included; check mark can be removed by the User.

✓ Insulation Weight Included; check mark can be removed by the User.

✓ Anchor Movements Included; the check mark can be removed by the User.

✓ Restraint Movements Included; check mark can be removed by the User.

✓ Restraint Loads Included; check mark can be removed by the User.

- ✓ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – Gray out
- ❑ Wind Loading as Occasional Load – Gray out
- ❑ Seismic Loads – Static Equivalent – Gray out – Not accessible with only an operating case
- ❑ Mode Shapes and Frequencies – Gray out

With Operating Case Analysis with Piping Code Compliance Analysis

- ✓ Perform Operating Case Analysis
- ❑ Perform Hydro-test Case Analysis – Gray out
- ✓ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired
- ✓ Temperature – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pressure – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Pipe Weight – Gray out – Make active again if check in Perform Operating Case is removed.
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ✓ Anchor Movements Included; the check mark can be removed by the User.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ✓ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – Gray out – Wind cannot be considered if Soil is.
- ❑ Wind Loading as Occasional Load – Gray out – Wind cannot be considered if Soil is.

- ✓ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if piping code is selected.
- ❑ Mode Shapes and Frequencies – Gray out

With Piping Code Compliance Analysis – No Operating Case Analysis

- ❑ Perform Operating Case Analysis
- ❑ Perform Hydro-test Case Analysis – Gray out
- ✓ Perform Piping Code Compliance Analysis – the User will be allowed to check this box, if desired
- ❑ Temperature – Gray out
- ❑ Pressure – Gray out
- ❑ Pipe Weight – Gray out
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ✓ Anchor Movements Included; check the User can remove mark.
- ✓ Restraint Movements Included; check mark can be removed by the User.
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ✓ Soil Interaction Included – the User will be allowed to check this box, if desired
- ❑ Wind included with Weight – Gray out – Wind cannot be considered if Soil is.
- ❑ Wind Loading as Occasional Load – Gray out – Wind cannot be considered if Soil is.
- ✓ Seismic Loads – Static Equivalent – the User will be allowed to check this box, if piping code is selected.
- ❑ Mode Shapes and Frequencies – Gray out

Mode Shapes and Frequencies – In this field, the User may place a check mark to instruct TRIFLEX to perform a modal analysis. Checks may be placed in the fields where check marks are shown.

- ❑ Perform Operating Case Analysis

- ❑ Perform Hydro-test Case Analysis – Gray out
- ❑ Perform Piping Code Compliance Analysis – Gray out
- ❑ Temperature – Gray out
- ❑ Pressure – Gray out
- ❑ Pipe Weight – Gray out
- ✓ Contents Weight Included; check mark can be removed by the User.
- ✓ Insulation Weight Included; check mark can be removed by the User.
- ❑ Anchor Movements Included; Gray out
- ❑ Restraint Movements Included; Gray out
- ✓ Restraint Loads Included; check mark can be removed by the User.
- ❑ Soil Interaction Included – Gray out.
- ❑ Wind included with Weight – Gray out – Wind cannot be considered if Soil is.
- ❑ Wind Loading as Occasional Load – Gray out – Wind cannot be considered if Soil is.
- ❑ Seismic Loads – Static Equivalent – Gray out
- ✓ Mode Shapes and Frequencies

3.3.5 Occasional Loading Data

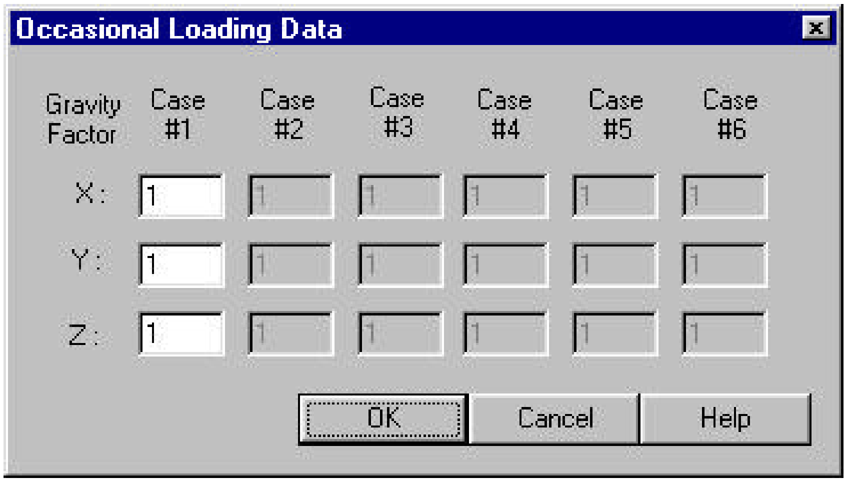


Figure 3.3.5 - Setup Occasional Loading Data

When the User wishes to simulate an occasional load as a percentage of gravity along one, two or three axes, the User should place a check in the Seismic Loads – Static Equivalent field on the Load Case dialog for the desired load case. When the check is properly placed in the Load Case dialog for a specific load case, the X, Y and Z Gravity Factor fields for that specific load case in the Occasional Loading Data dialog will be made active. In the X, Y and Z fields, the User may enter the desired magnitude of the gravity factor. Gravity factors may be positive or negative to indicate the application direction of the occasional loading. Note also that the User may enter different gravity factors for each load case, if desired. In order to obtain a piping code compliance analysis, the User must also request that TRIFLEX process this analysis on the Load Case dialog by placing a check in the Perform Piping Code Compliance Analysis check box.

3.3.6 Mode Shapes and Frequencies

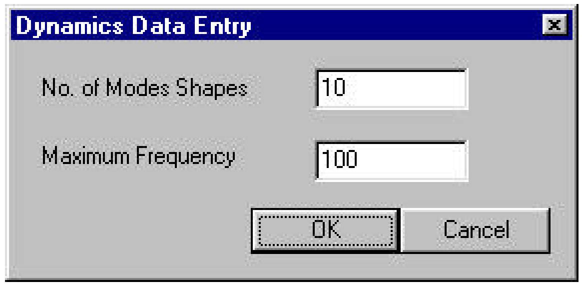


Figure 3.3.6 – Dynamic Data Entry

In order for TRIFLEX to perform a modal analysis of a piping system, the User must define several items of information in addition to the basic piping model. To enter the required data, the User must click on Setup on the Main Menu and then on Modal Analysis on the drop down list. Upon clicking Modal Analysis, a Modal Analysis Data dialog will be presented to the User. The User must accept the default values or enter the data desired as follows:

No. of Mode Shapes – In this field, the User is to enter the desired number of modes or frequencies to be calculated. The default is 10.

Maximum Frequency – In this field, the User is to specify the cut-off (maximum) frequency (default value = 100 Hz.) for the analysis in Hertz or cycles/sec. When TRIFLEX processes the analysis, it will stop determining frequencies when the frequency exceeds the cut-off frequency entered by the User.

3.3.7 Graphic Color Settings

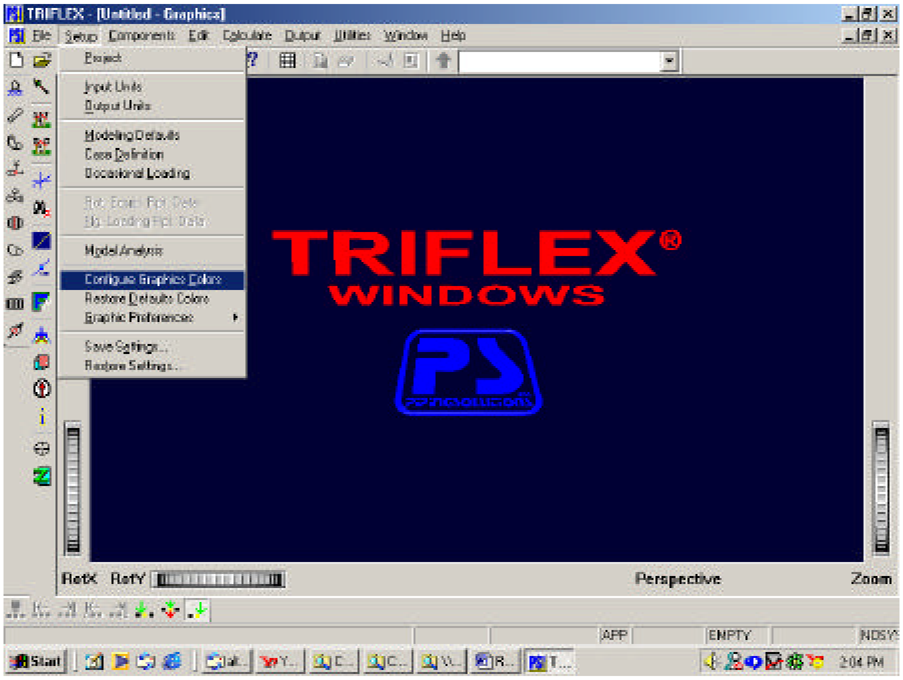


Figure 3.3.7 - Graphic Color Settings

3.3.7.1 Graphic Color Settings

Graphic Color Preferences can be selected for:

1. Background
2. Text
3. Any Graphic Component

Once one of the above items is selected then the user can select from a predetermined selection of “Basic Colors” or create a “custom color” for use.

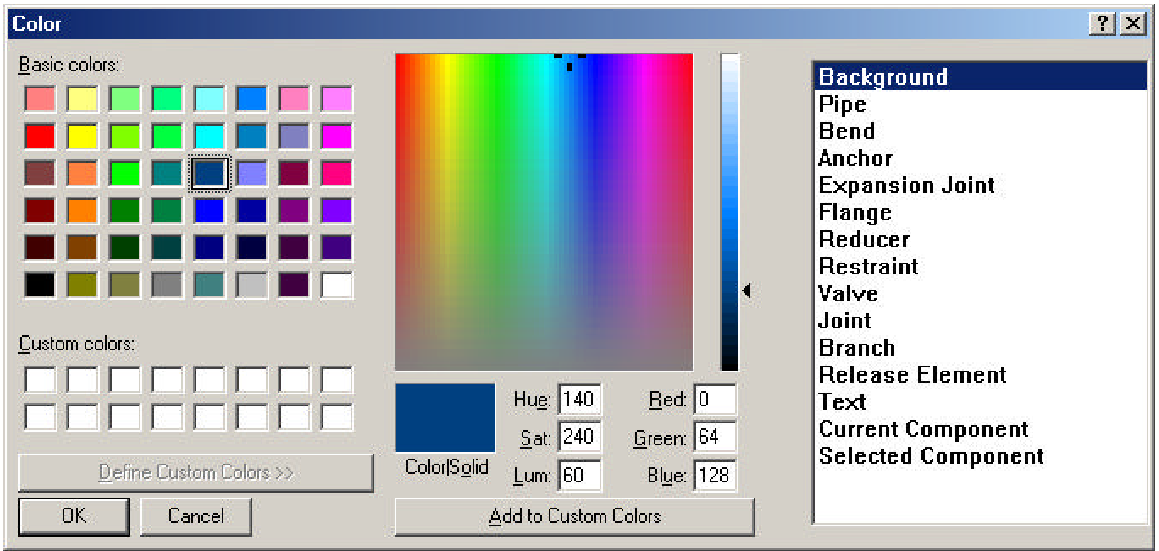


Figure 3.3.7.1 –Color Selection

To select a "**Basic Color**" click on the "**Basic Color**" and select **OK**.

To create a "Custom Color" depress the **Define Custom Color Bar** and select the color by clicking on the color in the box above the Hue, Sat., Lum, Red, Green, Blue boxes in the lower right hand side of the dialog box.

Note: When Cutting graphics to a final report it may be useful to change the background to white.

To make coarse adjustments use the sliding color scale on the right next to the color box. To make fine adjustments to the color use: **Hue, Sat., Lum, Red, and Green, Blue** input controls.

To save this color as a" custom color" depress the **Add to the Custom Colors** bar.

3.3.8 Graphic Preferences

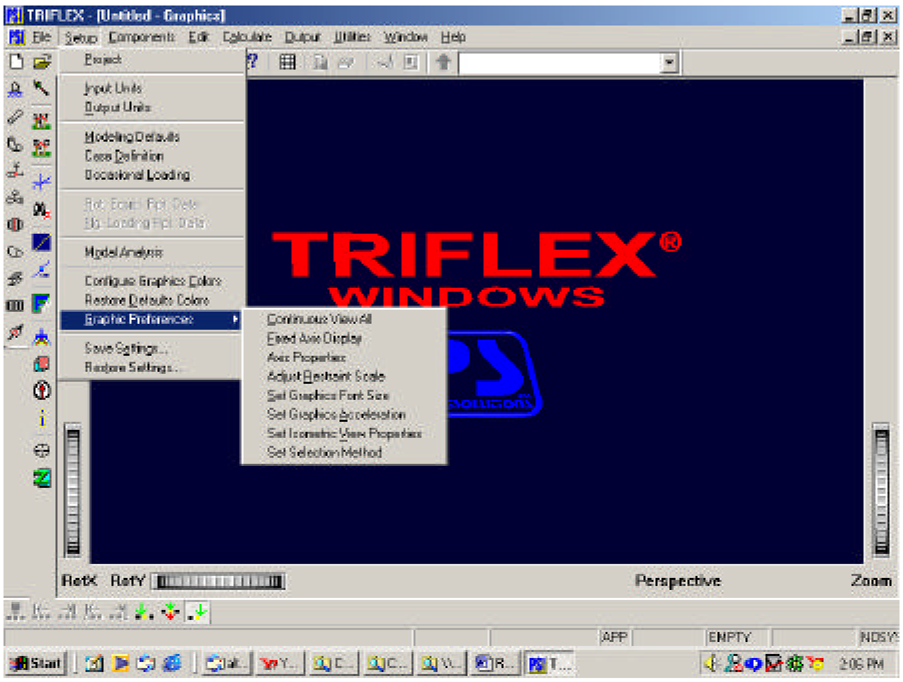


Figure 3.3.8 – Graphical Preferences

The user may select the following:

1. **Continuous All Views** - This allows the user to automatically have the graphics centered on the screen every time a new component is selected.

2. **Adjust Axis Scale** – The user can select an integer between 1 and 100 for the X, Y, and Z leg of the axis indicator.

3. **Adjust Restraint Scale** – This allows the user the ability to select an integer between 1 and 100 to adjust the relative size of the restraint indicator with respect to the dimension of the pipe or fitting to which they are attached. The options are to indicate the value in the **Nominal Restraint** field and the Spring field.

4. **Set Graphic Font Size** – The user can select the **Graphic Font Size**, using an integer from 4 to 72.

3.3.9 Save Graphic Setting

This allows the user to save the settings previous chosen from **Graphic Color Settings** and **Graphic Preferences**. This setting is stored as a *. ini file by the user, by a user determined path.

3.3.10 Restore Setting

This allows the user to restore the previously saved setting (*.ini) from a path determined by the user.

3.4 Coding Anchor Components

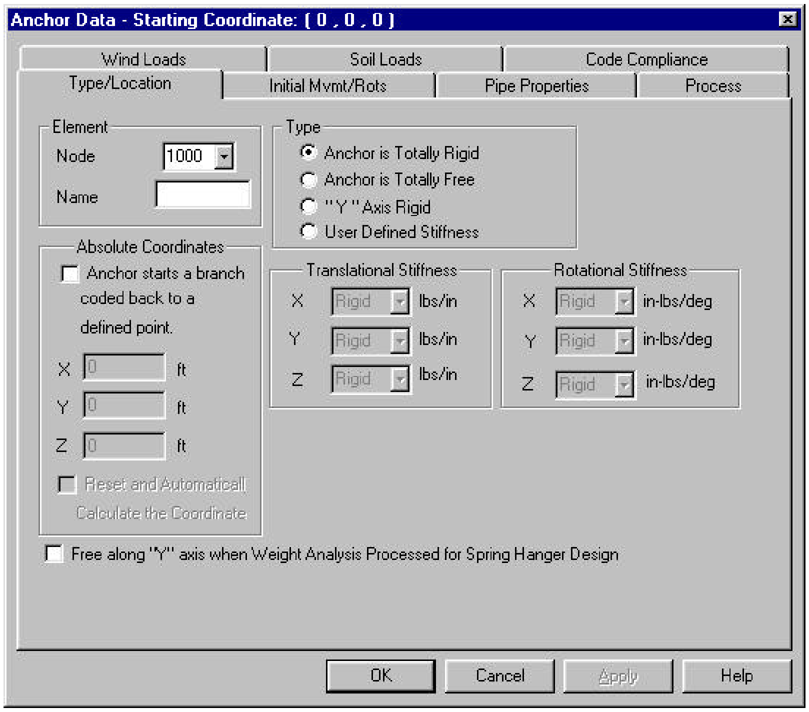


Figure 3.4-1 Anchor Components, All Tabs

3.4.1 Discussion of Anchor Component – Type/Location Tab

To enter an Anchor component, the User must click on the Anchor Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Anchor on the resulting pull down menu. Upon either of these sequences of actions, an Anchor dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Anchor dialog, a data group entitled “Element” is available for User data entry. The fields in which data can be entered in this data group are defined below:

Node – In this field, TRIFLEX will generate a Node Number equal to: 1.) The Initial Node Number entered by the User in the Modeling Defaults if this is the first node being entered, or 2.) The next available Node Number based upon the Last coded To Node number plus the node increment specified by the User in the Modeling Defaults. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the “Element” data, the User will find a data group entitled “Absolute Coordinates”. The default values will be all zeros. If the User wishes to enter coordinates for the first Anchor, TRIFLEX will use this entered coordinate as the starting coordinate. All of the subsequent components’ coordinates will be based upon this starting coordinate. If the User wishes to enter coordinates for a subsequent Anchor, TRIFLEX will use this entered coordinate to compare with the coordinate calculated by TRIFLEX. In the event that the coordinates are different, TRIFLEX will flag this as an error for the user to sort out and correct. The fields in which data can be entered in this data group are defined below:

X, Y and Z– The User can leave these fields blank or enter a numerical value in one, two or three of these fields. The default values will be 0,0,0.

To the immediate right of the “Element” data, the User will find a data group entitled “Type”. The fields in which data can be entered in this data group are defined below:

Anchor is Totally Rigid – If the User wishes to instruct TRIFLEX to consider the anchor to be totally rigid, then the User should accept the selection of this radio button. The default selection for all anchors will be with this radio button selected.

Anchor is Totally Free – If the User wishes to instruct TRIFLEX to consider the anchor to be totally free, then the User should click on the radio button just to the left of “Anchor is Totally Free”. TRIFLEX will then consider the anchor to be completely flexible along and about the X, Y and Z-axes.

<u>“Y” Axis Rigid</u> – If the User wishes to instruct TRIFLEX to consider the anchor to be totally free along the X and Z-axes and about all axes, then the User should click on the radio button just to the left of “Y” Axis Rigid. TRIFLEX will then consider the anchor to be completely flexible along the X and Z-axes and about the X, Y and Z-axes; but totally rigid along the Y-axis.

<u>User Defined Stiffness</u> – If the User wishes to instruct TRIFLEX to consider the anchor to have specific stiffness along one or more axes or about one or more axes, then the User should click on the radio button just to the left of “User Defined Stiffness”. When this radio button is selected, TRIFLEX will then activate the Translational Stiffness data group and the Rotational Stiffness data group to enable the User to enter the desired stiffness along and about the X, Y and Z-axes.

Immediately below the data group entitled “Type”, the User will find a data group entitled “Translational Stiffness”. The fields in which data can be entered in this data group are further defined below:

<u>X, Y and Z</u>– If the User has selected the User Defined Stiffness radio button, then the User can select Free or Rigid from the drop down combo list in the X, Y and/or Z fields or enter a numerical value in any of these fields to define the desired stiffness.

Immediately to the right of the data group entitled “Translational Stiffness”, the User will find a data group entitled “Rotational Stiffness”. The fields in which data can be entered in this data group are further defined below:

<u>X, Y and Z</u>– If the User has selected the User Defined Stiffness radio button, then the User can select Free or Rigid from the drop down combo list in the X, Y and/or Z fields or enter a numerical value in any of these fields to define the desired stiffness.

Immediately below the “Absolute Coordinates” data group, the User will find additional data fields for one additional anchor option as follows:

<u>Free along “Y” axis when Weight Analysis Processed for Spring Hanger Design</u> **-** If the User has elected to have TRIFLEX size and select spring hangers in this analysis and wishes to instruct TRIFLEX to consider this anchor to be free along the “Y” only during the Weight Analysis, then the User should place a check in the box immediately to the left of the label “Free along “Y” axis when Weight Analysis Processed for Spring Hanger Design”. The default for this option is that it is not selected. For a further discussion about the use of this option, see the Chapter 5 – Use of Restraints.

This Anchor starts a branch that is being coded back to a defined point in the piping system – If the User wishes to stop at some known point in the piping system and start modeling a new branch from an anchor point, the User should place a check in the box immediately to the left of this label. The default for this option is that it is not selected. When this check box is checked, the User will be required to enter values in the Absolute Coordinates data group. TRIFLEX will indicate that a geometry error exists until the branch is completed. Once the branch is connected to the piping system, TRIFLEX will calculate the coordinates of each point in the piping system. In the event that the Absolute Coordinates entered by the User are incorrect, TRIFLEX will automatically correct them.

3.4.2 Discussion of Anchor Component – Init. Mvmts and Rotations Tab

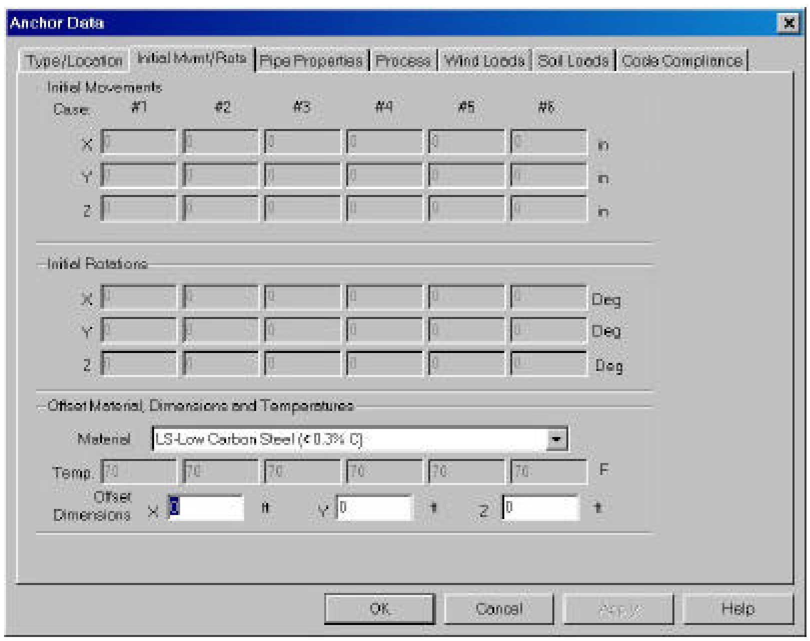


Figure 3.4. Node 1000, Anchor Dialog, Initial Mvmt/Rot Tab

For every anchor in a piping model, the User may enter initial movements and initial rotations, if desired. The initial movements and initial rotations are those that are caused by the growth or shrinkage of connected equipment or the physical movement and/or rotation of the connected equipment resulting from any cause. To enter the required data, the User must click on the Initial Mvmt/Rots tab at the top of the screen on each anchor component entered. Upon clicking on the tab, an Initial Movement / Rotations dialog will be presented to the User.

In the upper left corner of the Initial Mvmt/Rots dialog, a data group entitled "Initial Movements" is available for User data entry. The default value for each field will be zero. The data is to be entered by the User on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User may specify initial movements for each anchor for that particular load case. To activate a load case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired load case. If this has not been done, the fields for the initial movements will be grayed out. The fields in which data can be entered in this data group are defined below:

X Movement – The numerical value entered in this field for each case by the User represents the movement along the X Axis imposed on the anchor from the installed to operating position for that case.

Y Movement – The numerical value entered in this field for each case by the User represents the movement along the Y Axis imposed on the anchor from the installed to operating position for that case.

Z Movement – The numerical value entered in this field for each case by the User represents the movement along the Z Axis imposed on the anchor from the installed to operating position for that case.

Immediately below the data group entitled “Initial Movements”, the User will find a data group entitled “Initial Rotations”. The default value for each field will be zero. The data is to be entered by the User on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User may specify initial rotations for each anchor for that particular load case. To activate a load case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case. If this has not been done, the fields for the initial rotations will be grayed out. The fields in which data can be entered in this data group are defined below:

X Rotation – The numerical value entered in this field for each case by the User represents the rotation about the X Axis imposed on the anchor from the installed to operating position for that case.

Y Rotation – The numerical value entered in this field for each case by the User represents the rotation about the Y Axis imposed on the anchor from the installed to operating position for that case.

Z Rotation – The numerical value entered in this field for each case by the User represents the rotation about the Z Axis imposed on the anchor from the installed to operating position for that case.

Immediately below the “Initial Rotations” data, the User will find a data group entitled “Offset Material, Dimensions and Temperatures”. In the event that the User would prefer TRIFLEX to compute the anchor movements based upon the material of the anchor, the change in temperature of the material of the anchor and offset dimensions, then the User may enter this data and TRIFLEX will generate the anchor movements. The fields in which data can be entered in this data group are defined below:

Material – The User can click on the drop down combo list in this field and then select the desired material from the list of available materials for the anchor casing. In the event that the desired material is not available in the list, the User should enter the desired movements in the fields noted above.

In the event that the User wishes to enter the properties for one or more materials in the library of materials available on the drop down combo list, all such entries are to be made through "Utilities" then "Databases" then "Materials". This field is identical to the Material field that is found on the Pipe Properties dialog. The material selected by the User for the anchor casing may be different from the pipe material selected by the User on the Pipe Properties dialog. The material selected by the User on the Pipe Properties dialog will be the default material for the anchor casing material.

Temperature – In this field, the User may enter the temperature of the anchor casing for each of six cases. Data can only be entered in an active field (one that is not grayed out).

The default value is the value for base temperature for the system of units selected by the User.

Offset Dimensions – The offset dimensions are the actual dimensions of the anchor casing from the actual fixed point from which growth or shrinkage originates to the node location identified by the User as the anchor point in the piping model. Anchor movements for this data point will be calculated by TRIFLEX based the anchor casing material, the change in temperature experienced by the anchor casing from the installed to operating conditions and the offset dimensions entered by the User. The default values for the offset dimensions will be zero.

3.4.3 Discussion of Anchor Component, Pipe Properties Tab

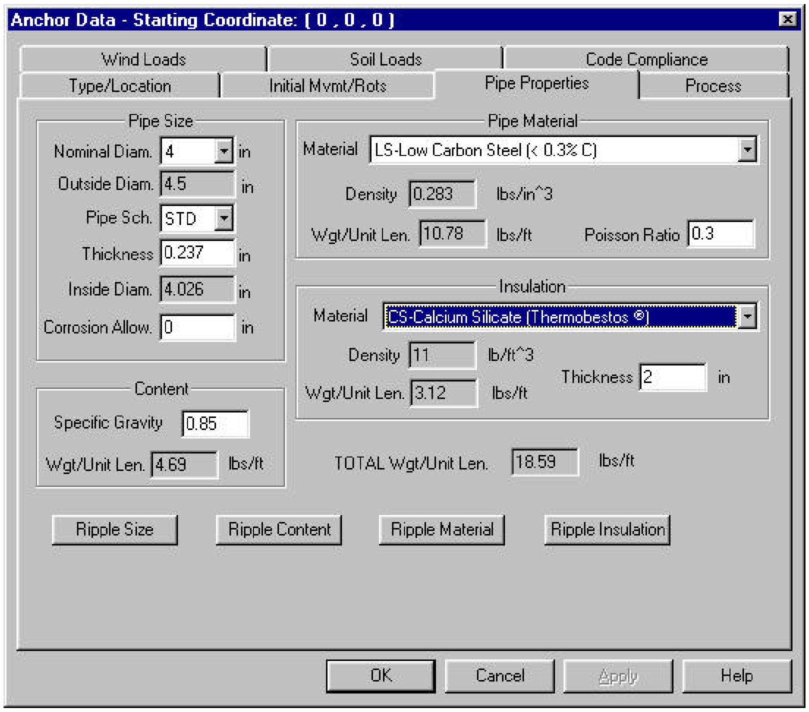


Figure 3.4.3 - Anchor Component, Pipe Properties Tab

For every piping system to be properly analyzed, the User must enter basic piping properties. To enter these required piping properties, the User must click on the Pipe Properties tab at the top of the screen on the first component entered. Upon clicking on the tab, a Pipe Properties dialog will be presented to the User.

The data is organized in related data groups on this dialog. In the upper left corner of the Pipe Properties dialog, a data group entitled "Pipe Size" is available for User data entry. The fields in which data can be entered in this data group are defined below:

<u>Nominal Diam.</u> – In this field, TRIFLEX will display the default pipe size of 6" for the first pipe component. If the desired nominal diameter is any size other than 6", the User may select a different nominal diameter from the drop down combo list in this field. In the event that the User does not find the desired nominal diameter in the list provided, the User may select "User Specified" and enter the exact outside diameter of the pipe for this particular pipe size and application or can add the desired nominal pipe diameter to the library of pipe diameters by clicking on "Utilities" then "Databases" then "Pipe".

<u>Outside Diam.</u> - When the User has entered a nominal diameter, TRIFLEX will display, in this field, the outside diameter of the pipe that TRIFLEX looked up in the internal database for the nominal diameter selected by the User. The field will be grayed out and inaccessible to the User, except through the Nominal Diameter field. When the User has selected "User Specified" in the Nominal Diameter field, TRIFLEX will activate the field and thereby enable the User to enter an actual outside diameter of the pipe.

Pipe Schedule – When the User has entered a nominal diameter; the User may select the desired pipe schedule from the drop down combo list in this field. In the event that the User does not find the desired pipe schedule in the list provided, the User may select "User Specified" and enter the exact pipe wall thickness or can add the desired custom pipe diameter and pipe schedule/wall thickness to the library of pipe diameters by clicking on "Utilities" then "Databases" then "Pipe".

Pipe Wall Thickness – When the User has entered a Nominal Diameter and a Pipe Schedule, TRIFLEX will display, in this field, the wall thickness of the pipe that TRIFLEX looked up in the internal database for the nominal diameter and schedule selected by the User. In the event that the User did not find the desired pipe schedule in the list provided and the User selected "User Specified", the User may then enter the exact pipe wall thickness in this field.

Inside Diam. - In this field, TRIFLEX will calculate and display the actual inside diameter of the pipe. From the User-entered outside diameter, TRIFLEX will subtract two times the pipe wall thickness. The resultant value will be displayed in this field. The field will be grayed out and inaccessible to the User, except through the appropriate pipe size fields.

Corrosion Allow. - If the user specifies a corrosion allowance in this field, TRIFLEX will perform a worst-case analysis (i.e., for calculating the forces and moments, the full un-corroded wall thickness will be assumed). For calculating stresses, the program will assume that the pipe is in the fully corroded state. The default value for this field is zero.

Immediately below the "Pipe Size" data, the User will find a data group entitled "Contents". The data entered in this data group define the weight of the contents of the component defined on the node dialog. The fields in which data can be entered in this data group are defined below:

Specific Gravity - If the user specifies a numeric value in this field, TRIFLEX will consider the value to be the liquid specific gravity of the contents of the component defined on the node dialog. TRIFLEX will calculate the weight per unit length for the contents based upon the specific gravity entered by the User and the inside diameter of the piping component. This value will then be shown in the **Weight/Unit Len.** Field located immediately below this field. The default value for this field is zero. For water filled piping, the User should enter a contents specific gravity of 1.0.

Weight/Unit Len. - In this field, TRIFLEX will calculate and display the actual weight per unit length of the contents of the pipe. The field will be grayed out and inaccessible to the User, except through the specific gravity field.

To the immediate right of the data group entitled "Pipe Size", the User will find a data group entitled "Pipe Material". The fields in which data can be entered in this data group are defined below:

Material – The User can click on the drop down combo list in this field and then select the desired pipe material from the list of available materials. In the event that the desired material is not available in the list, the User can select "User Specified" and enter the desired values for the pipe material density and Poisson's ratio on this dialog as well as the modulus of elasticity and coefficient of expansion on the Process tab.

In the event that the User wishes to enter these properties for one or more materials in the library of materials available on the drop down combo list, all such entries are to be made through "Utilities" then "Databases" then "Materials".

Density – When the User has selected a pipe material from the drop down combo list in the material field just above, TRIFLEX will select the proper value for the density from the data base in TRIFLEX and will display the value in this field. The field will be grayed out and inaccessible to the User, except through the material field. combo list in

When the User has selected "User Specified" from the drop down the material field just above, TRIFLEX will activate this field and thereby enable the User to enter the desired density of the pipe.

The current database supplied with TRIFLEX contains the following piping density properties. The following table shows the material codes that are within the TRIFLEX database. The column marked Post '90 reflect the material codes that use the latest modulus of elasticity as found in the post '90 ANSI B31.1 and ANSI B31.3 piping codes. The older, Pre '90, codes are maintained within the TRIFLEX program so users may continue to run older jobs. Users may change the material codes of older piping models to reflect the new materials.

Material	Code Post '90	Code Pre '90	Eng	SI	MET	IU1
Units			lb/in^3	$N/m^3 \times 10^4$	g/cm^3	kg/m^3
Aluminum	AL	AL	0.098	0.266	2.71	2713
Austenitic Stainless	AS	AU	0.290	0.787	8.03	8027
Brass (Yellow)	BR	BR	0.306	0.831	8.47	8470
Bronze (Phosphor 9%)	BZ	BZ	0.318	0.863	8.80	8802
Gray Cast Iron	CI	CI	0.265	0.719	7.34	7335
Carbon Molybdenum Steels	MS	CM	0.283	0.768	7.83	7833
Copper Nickel (70 Cu-30 Ni)	CN	CN	0.323	0.877	8.94	8941
Straight Chrome Steels (12 Cr, 17 Cr, 27 Cr)	SC	CR	0.281	0.763	7.78	7778
Low Carbon Steels (< 0.3% C)	LS	CS	0.283	0.768	7.83	7833
Copper Nickel (Navy Specs)	CU	CU	0.323	0.877	8.94	8941
High Carbon Steels (> 0.3% C)	HS	HC	0.283	0.768	7.83	7833
Intermediate Chrome Moly Steel 5 to 9 CrMo	IM	IC	0.283	0.768	7.83	7833
K-Monel	MK	KM	0.306	0.831	8.47	8470
Low Chrome Moly Steels through 3 CrMo	LM	LC	0.283	0.768	7.83	7833
Monel	ML	MO	0.319	0.866	8.83	8830
Nickel Iron Chrome (Ni-Fe-Cr)	NC	NC	0.300	0.814	8.30	8304
Nickel (3.5% Ni)	NK	NI	0.322	0.874	8.91	8913
Type 310 Stainless (25 Cr 20 Ni)	SL	ST	0.290	0.787	8.03	8027
Wrought Iron	WI	WI	0.280	0.760	7.75	7750

Weight/Unit Len. - In this field, TRIFLEX will calculate and display the actual weight per unit length of the pipe. The field will be grayed out and inaccessible to the User, except through the material field or density field if the User selected User Specified from the drop down combo list in the material field.

Poisson's Ratio – In this field, TRIFLEX will display a value of 0.3 for all materials. If the User wishes to alter this value, the User can click on this field and edit it.

Immediately below the data group entitled "Pipe Material", the User will find a data group entitled "Insulation". The fields in which data can be entered in this data group are further defined below:

Material – The User can click on the drop down combo list in this field and then select the desired insulation material from the list of available materials. In the event that the desired material is not available in the list, the User can select "User Specified" and enter the desired values for the insulation material density on this dialog.

In the event that the User wishes to enter these properties for one or more materials in the library of insulation materials available on the drop down combo list, all such entries are to be made through “Utilities” then “Databases” then “Insulation Matls”.

Density – When the User has selected an insulation material from the drop down combo list in the material field just above, TRIFLEX will select the proper value for the density from the data base in TRIFLEX and will display the value in this field. The field will be grayed out and inaccessible to the User, except through the material field.

When the User has selected “User Specified” from the drop down combo list in the material field just above, TRIFLEX will activate this field and thereby enable the User to enter the desired density of the insulation.

The current database of insulation materials supplied with TRIFLEX has the following insulation density properties.

Insulation Material	Code	Eng	S I	MET	IU1
Units		Lb/ft^3	N/m^3	Kg/m^3	Kg/m^3
Amosite Asbestos	AS	16.0	2513	256	256
Calcium Silicate Thermobestos 7 85% Magnesia Calcium Silicate	CS	11.0	1728	176	176
Careytemp 7	CT	10.0	1571	160	160
Foam-glass Cellular	FG	9.0	1414	144	144

Glass					
Fiberglass Owens/Corning 25 ASJ	FS	7.0	1100	112	112
High Temp	HT	24.0	3770	384	384
Kaylo 10 J	KA	12.5	1964	200	200
Mineral Wool	MW	8.5	1335	136	136
Perlite Celo-temp J 1500	PE	13.0	2042	208	208
Styro-Foam	ST	1.8	283	29	29
Super-X	SX	25.0	3927	400	400
Poly-Urethane 5 to 9 CrMo	UR	2.2	346	35	35

Weight/Unit Len. - In this field, TRIFLEX will calculate and display the actual weight per unit length of the insulation covering the pipe. The field will be grayed out and inaccessible to the User, except through the material field or density field if the User selected User Specified from the drop down combo list in the material field.

Thickness – If the User has selected an insulation material, then the User must also specify an insulation thickness in this field. The default value for this field is zero.

Immediately below the data group entitled "Insulation", the User will find a miscellaneous data item as further defined below:

Total Weight/Unit Len. - In this field, TRIFLEX will display the actual weight per unit length of the pipe itself, the contents and the insulation covering the pipe.

The field will be grayed out and inaccessible to the User, except through entry of the contents data, the pipe data and the insulation data fields.

Note: Inheritance and Rippling Property Changes

In TRIFLEX, physical properties such as those entered on the Pipe Properties dialog, the Process dialog, the Wind Loads dialog, the Soil Loads dialog and the Code Compliance dialog are all inherited from one piping component to the next as the piping model is built. This eliminates the necessity for the User to enter these physical properties more than once as the piping model is built. Whenever the User wants to change any of these physical properties as the model is being built, the User simply enters the new value in the appropriate field. From that component on, the piping model will use the new value as last entered.

Rippling, on the other hand, applies to piping models that have already been built. In other words, rippling applies where physical properties have already been defined and where the User wants to change the original value to a different value on one piping component as well as a number of subsequent components. In an existing TRIFLEX piping model, a User can elect to change a specific piping property on any component and then instruct TRIFLEX to change all subsequent occurrences of the originally specified property that are in an unbroken series from that point forward in the piping model. See the following example for clarification:

From Node No.	To Node No.	Nominal Diam.	Pipe Sched.
5	10	6	40
10	15	6	40
15	20	6	40
20	25	6	40
25	30	6	40
30	35	8	20
40	45	8	20
45	50	8	20
50	55	8	20
55	60	6	40

60	65	6	40

The User has modeled 6" schedule 40 pipe from Node No. 5 to Node No. 30. In the event that the User wishes to change the 6" schedule 40 pipe from Node No. 15 to Node No. 30 to 12" schedule 30 pipe, then the User should replace the 6" schedule 40 pipe defined on the component From 15 - To 20 with a 12" schedule 30 pipe and should then click on the Ripple Size button at the bottom of the Pipe Property dialog. TRIFLEX will change the 6" schedule 40 pipe on Node No. 15 – 20, 20 - 25 and 25 – 30 to be 12" schedule 30 pipe. TRIFLEX will not change the 6" schedule 40 pipe on Node No. 55 – 60, and 60 – 65. It is not in an unbroken series of components from the original Node No. 30.

Ripple Size – When the User has modified one or more properties in the Pipe Size data group, the User can instruct TRIFLEX to modify all subsequent occurrences of these properties that are in an unbroken series from the original revision forward by pressing the Ripple Size button.

Ripple Contents – When the User has modified one or more properties in the Contents data group, the User can instruct TRIFLEX to modify all subsequent occurrences of these properties that are in an unbroken series from the original revision forward by pressing the Ripple Contents button.

Ripple Material – When the User has modified one or more properties in the Pipe Material data group, the User can instruct TRIFLEX to modify all subsequent occurrences of these properties that are in an unbroken series from the original revision forward by pressing the Ripple Material button.

Ripple Insulation – When the User has modified one or more properties in the Pipe Insulation data group, the User can instruct TRIFLEX to modify all subsequent occurrences of these properties that are in an unbroken series from the original revision forward by pressing the Ripple Insulation button.

3.4.4 Discussion of Anchor Component, Process Tab

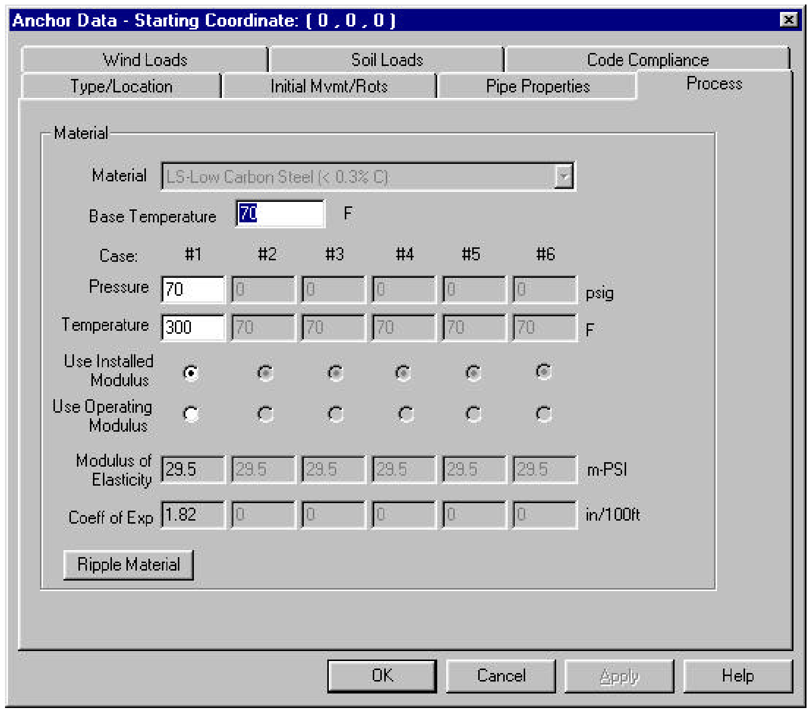


Figure 3.4.4 – Anchor Component, Process Tab

For every piping system to be properly analyzed, the User must enter basic piping properties and process data. To enter the required data, the User must click on the Process tab at the top of the screen on the first component entered. Upon clicking on the tab, a Process dialog will be presented to the User.

In the upper left corner of the Process dialog, a data group entitled "Material" is available for User data entry. The fields in which data can be entered in this data group are defined below:

Material – This field is identical to the Material field that is found on the Pipe Properties dialog, except that is not accessible. The field is a display field only and is grayed out. The information is provided for the User to see the material that the User selected on the Pipe Properties dialog. In the event that the User wishes to change this material, the User must return to the Pipe Properties dialog.

Base Temperature – In this field, the User can specify the base or ambient temperature at time of fabrication / installation. The default value assumed by TRIFLEX is 70 F or 21 C.

The remaining data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When a piping material is selected by the User from the database contained in TRIFLEX, the User may specify, on this dialog, the internal pressure, the temperature and whether the installed or operating modulus of elasticity is to be used. When the User selects "User Specified" for the piping material, the User may specify the modulus of elasticity and the coefficient of expansion to be used in the analysis. The User can also enter the temperature when "User Specified" is selected for material; however, the temperature entry has no effect on the analysis of User Specified materials.

Pressure – In this field, the User may enter the internal pressure for each of six cases. Data can only be entered in an active field (one that is not grayed out). To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case. The default value is zero.

Temperature – In this field, the User may enter the temperature for each of six cases. Data can only be entered in an active field (one that is not grayed out). The default value is the value for ambient temperature for the system of units selected by the User.

Use Installed Modulus / Use Operating Modulus – The selection of one or the other of these options is accomplished by the provision of two radio buttons. The default selection is "Use Installed Modulus". The User should select the Installed Modulus when performing piping code compliance studies. The User can select Use Operating Modulus in order to calculate loads on equipment if allowed by the equipment vendor. Use of the operating temperature modulus will also more accurately calculate distributed loads for the internal weight effect in an automated spring hanger sizing analyses. The resultant deflections using the operating modulus will also be more accurately calculated.

Modulus of Elasticity – When the User has selected a material from the material database contained within TRIFLEX; this field will be grayed out or inactive. In this field, the value of Modulus of Elasticity that TRIFLEX has selected from the TRIFLEX database will be displayed. When the User has selected "User Specified" for the piping material, the User may enter in this field the value of the modulus of elasticity to be used by TRIFLEX in the analysis.

Coeff. Of Expansion – When the User has selected a material from the material database contained within TRIFLEX; this field will be grayed out or inactive. In this field, the value of coefficient of expansion that TRIFLEX has selected from the TRIFLEX database will be displayed. When the User has selected "User Specified" for the piping material, the User may enter in this field the value of the coefficient of expansion to be used by TRIFLEX in the analysis.

Ripple Material – When the User has modified one or more properties in the Pipe Material data group, the User can instruct TRIFLEX to modify all subsequent occurrences of these properties that are in an unbroken series from the original revision forward by pressing the Ripple Material button.

3.4.5 Discussion of Anchor Component, Code Compliance

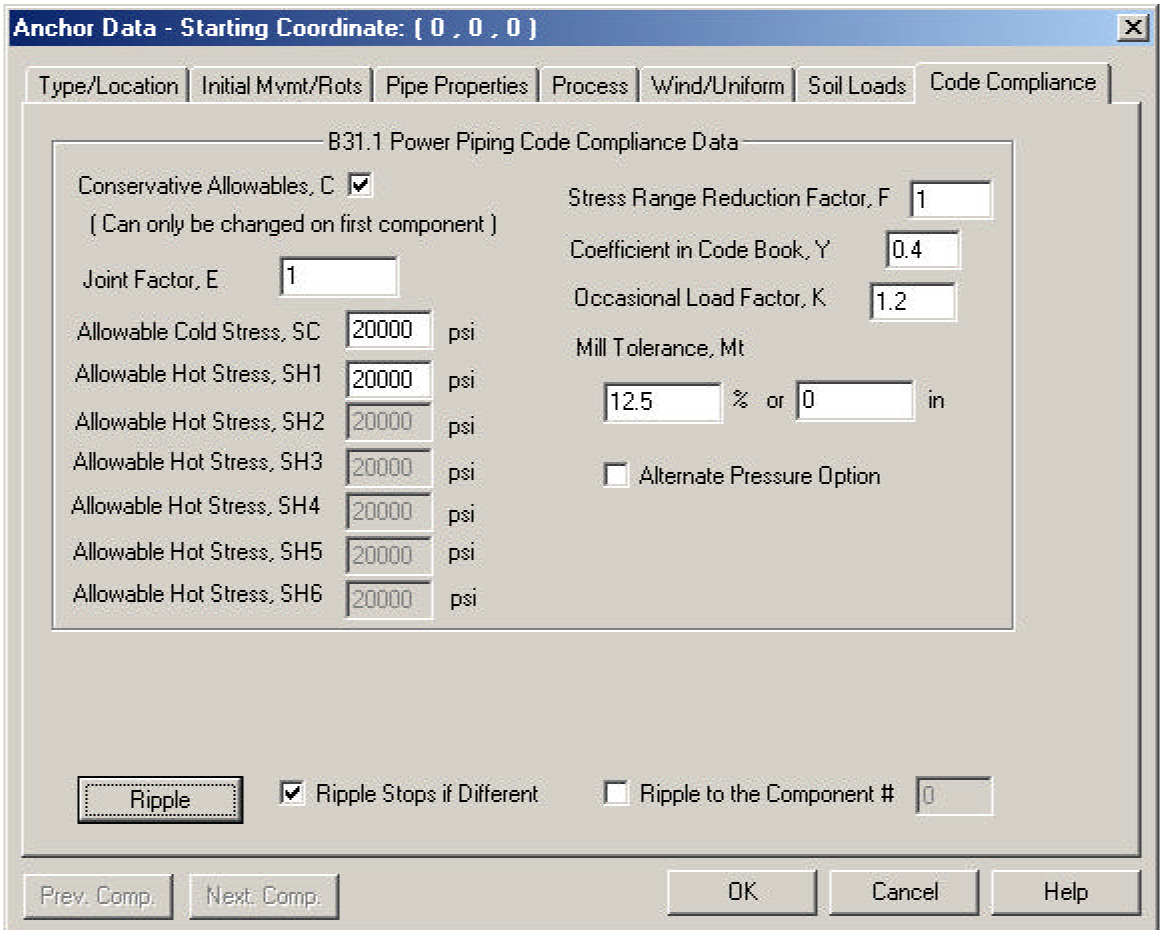


Figure 3.4.5 - Anchor Component, Code Compliance Tab

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "B31.1 Power Piping Compliance Data". The fields in which data can be entered in this data group are defined below:

Conservative Allowable, C – This field is a check box field. The default is with this check box checked – in other words, TRIFLEX defaults to the application of conservative stress allowable. If the User wishes TRIFLEX add the unused portion of the primary stress allowable to the allowable value for the secondary stress allowable value, then the User should eliminate the check in the check box.

Please note that this election can only be made on the first component in the piping system.

Joint Factor, E – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. The default value is 1.0.

Allowable Cold Stress, S_c – In this field, the User must enter a value for the allowable cold stress based upon the piping materials selected by the User. The default value is 20,000 psi.

The remaining allowable stress data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the hot allowable stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case. The default value is 20,000 psi.

Stress Range Reduction Factor, F – The User should enter the desired stress range reduction factor, based upon the number of cycles the piping system is expected to be subjected to. A default value of 1.0 will be assumed if the User does not enter this factor.

Coefficient in Code Book, Y - The User should enter the desired "Y" factor, based upon the codebook. A default value of 0.4 will be assumed if the User does not enter this factor. See Table 304-1.1 of the B31.1 Power Piping Code Book for reference.

Occasional Load Factor, K – The User can enter a value for the occasional load factor, if desired. A value of 1.15 should be entered for occasional loads acting less than 10 percent of the operating period and a value of 1.2 should be entered for loads acting for less than one percent of the operating period. A value of 1.2 will be assumed, if the User does not specify the factor.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent or in inches / mm. The default is 12 1/2 percent.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.1 B31.1 Code Compliance

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "B31.1 Power Piping Compliance Data". The fields in which data can be entered in this data group are defined below:

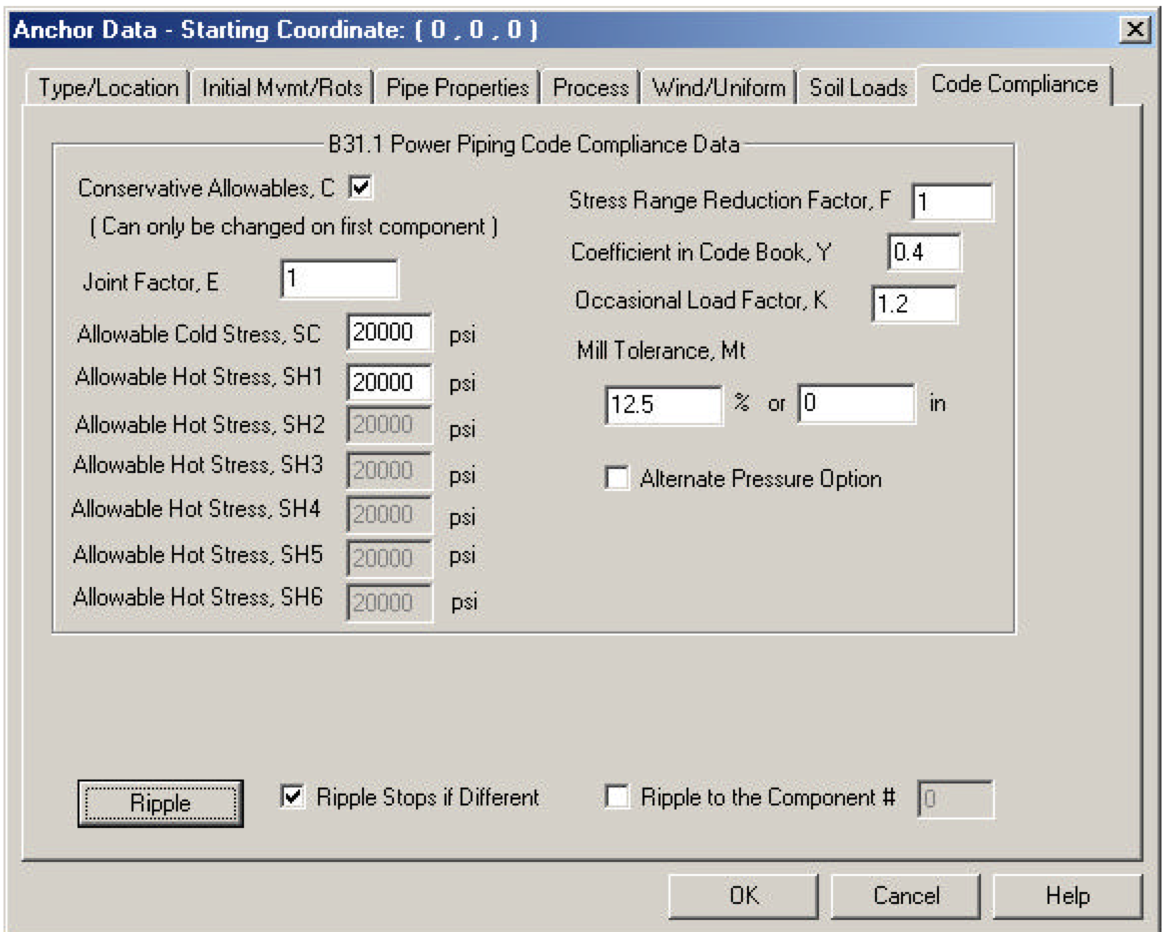


Figure 3.4.5.1 - Anchor Component, Code Compliance Tab, B31.1

<u>Conservative Allowable, C</u> – This field is a check box field. The default is with this check box checked – in other words, TRIFLEX defaults to the application of conservative stress allowable. If the User wishes TRIFLEX add the unused portion of the primary stress allowable to the allowable value for the secondary stress allowable value, then the User should eliminate the check in the check box. Please note that this election can only be made on the first component in the piping system.

Joint Factor, E – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. The default value is 1.0.

Allowable Cold Stress, S_c – In this field, the User must enter a value for the allowable cold stress based upon the piping materials selected by the User. The default value is 20,000 psi.

The remaining allowable stress data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the hot allowable stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case. The default value is 20,000 psi.

Stress Range Reduction Factor, F – The User should enter the desired stress range reduction factor, based upon the number of cycles the piping system is expected to be subjected to. A default value of 1.0 will be assumed if the User does not enter this factor.

Coefficient in Code Book, Y - The User should enter the desired "Y" factor, based upon the codebook. A default value of 0.4 will be assumed if the User does not enter this factor. See Table 304-1.1 of the B31.1 Power Piping Code Book for reference.

Occasional Load Factor, K – The User can enter a value for the occasional load factor, if desired. A value of 1.15 should be entered for occasional loads acting less than 10 percent of the operating period and a value of 1.2 should be entered for loads acting for less than one percent of the operating period. A value of 1.2 will be assumed, if the User does not specify the factor.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent or in inches / mm. The default is 12 1/2 percent.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.2 B31.3 Code Compliance and DIN 2413

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled “B31.3 Process Plant and Refinery Piping Compliance Data”. The fields in which data can be entered in this data group are defined below:

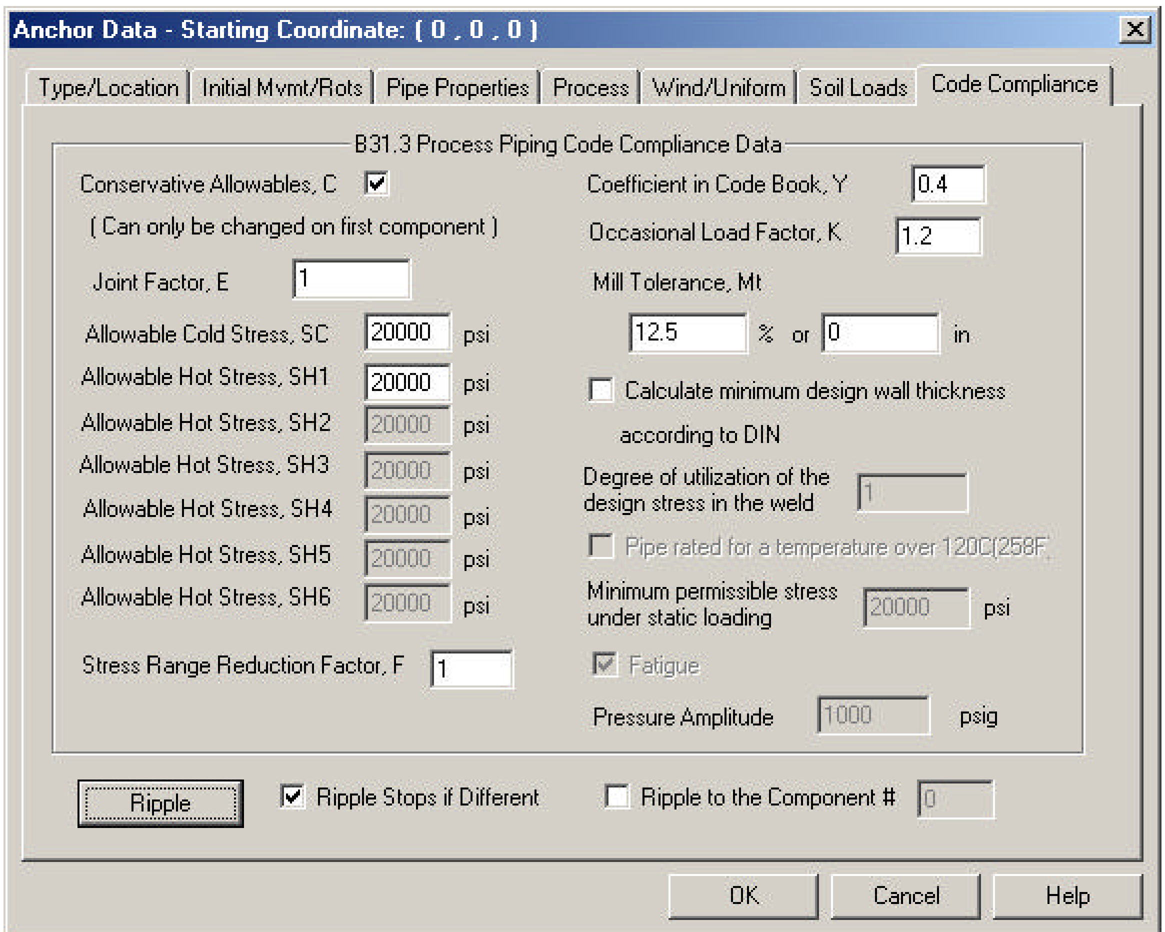


Figure 3.4.5.2 - Anchor Component, Code Compliance Tab,B31.3

<u>**Conservative Allowable, C**</u> – This field is a check box field. The default is with this check box checked – in other words, TRIFLEX defaults to the application of conservative stress allowable. If the User wishes TRIFLEX add the unused portion of the primary stress allowable to the allowable value for the secondary stress allowable value, then the User should eliminate the check in the check box.

Please note that this election can only be made on the first component in the piping system.

Joint Factor, E – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. The default value is 1.0.

Allowable Cold Stress, S_c – In this field, the User must enter a value for the allowable cold stress based upon the piping materials selected by the User. The default value is 20,000 psi.

The remaining allowable stress data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the hot allowable stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case. The default value is 20,000 psi.

Stress Range Reduction Factor, F – The User should enter the desired stress range reduction factor, based upon the number of cycles the piping system is expected to be subjected to. A default value of 1.0 will be assumed if the User does not enter this factor.

Coefficient in Code Book, Y - The User should enter the desired “Y” factor, based upon the codebook. A default value of 0.4 will be assumed if the User does not enter this factor. See Table 304-1.1 of the B31.3 Process Plant and Refinery Piping Code Book for reference.

Occasional Load Factor, K – The User can enter a value for the occasional load factor, if desired. A value of 1.15 should be entered for occasional loads acting less than 10 percent of the operating period and a value of 1.2 should be entered for loads acting for less than one percent of the operating period. A value of 1.2 will be assumed, if the User does not specify the factor.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent or in inches / mm. The default is 12 1/2 percent.

For minimum wall thickness calculation according DIN 2413 Code - to activate the DIN calculations the user need to check the box “Calculate the minimum design wall thickness according DIN”

Degree of utilization of design stress in the weld - In this field, the User should enter the degree of utilization of design stress in the weld used in the manufacture of the pipe. The default value is 1.0.

Pipe rated for a temperature over 120°C (258°F) – In this check box the USER can enter the service condition for piping system. By default the box is unchecked. In this case the calculations will be done according the formulae for load case I. If the check box is checked the calculation will be done according the formulae for load case II. (DIN 2413 – Part 1 – Table 3)

Maximum permissible stress under static loading s_{zul} – In this field for all activated load case the User must enter a value for the maximum permissible stress based upon the piping materials selected by the User. The default value is 20,000 psi.

Fatigue - In this check box the USER can enter the service condition for piping system. If the check box is checked the calculations will be done according the formulae for load case I and III. The higher value for minimum thickness will be displayed. (DIN 2413 – Part 1 – Table 3)

Pressure Amplitude - In this field for all activated load case the User must enter a value for the stress amplitude. Equation (4) (DIN 2413 – Part 1 – Table 3) shall be used to account for fatigue failure at constant stress amplitude. The default value is 1000 psi.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.3 B31.4 Code Compliance

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "B31.4 Liquid Petroleum Piping Compliance Data". The fields in which data can be entered in this data group are defined below:

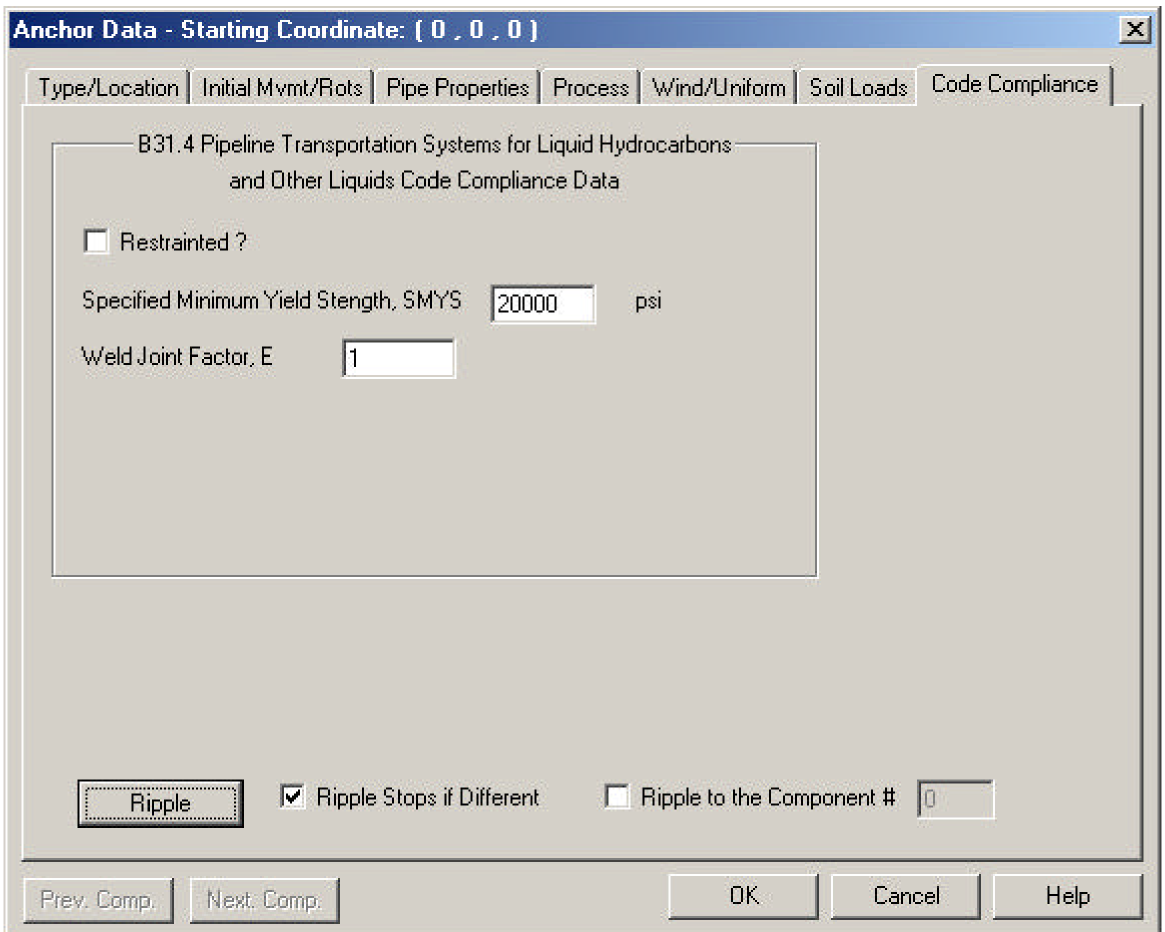


Figure 3.4.5.3 - Anchor Component, Code Compliance Tab, B31.4

Restrained – This field is a check box field. The default is with this check box not checked – in other words, TRIFLEX defaults to unrestrained piping. In general, restrained piping is underground piping with the soil restraining free movement of the piping. If the User wishes to define a portion of a piping system as "restrained piping", then the User should place a check in the check box by clicking on the box. For restrained piping, TRIFLEX will compute the longitudinal expansion stress from the equation given in B31.4 Section 419.6.4(b).

SMYS – In this field, the User must enter a value for the specified Minimum Yield Strength of the pipe to be covered by the Code Compliance. The default value is 20,000 psi.

Joint Factor, E – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. The default value is 1.0.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.4 B31.8 Code compliance

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "B31.8 or DOT Guidelines – Gas Transmission & Distribution Systems Code Compliance Data". The fields in which data can be entered in this data group are defined below:

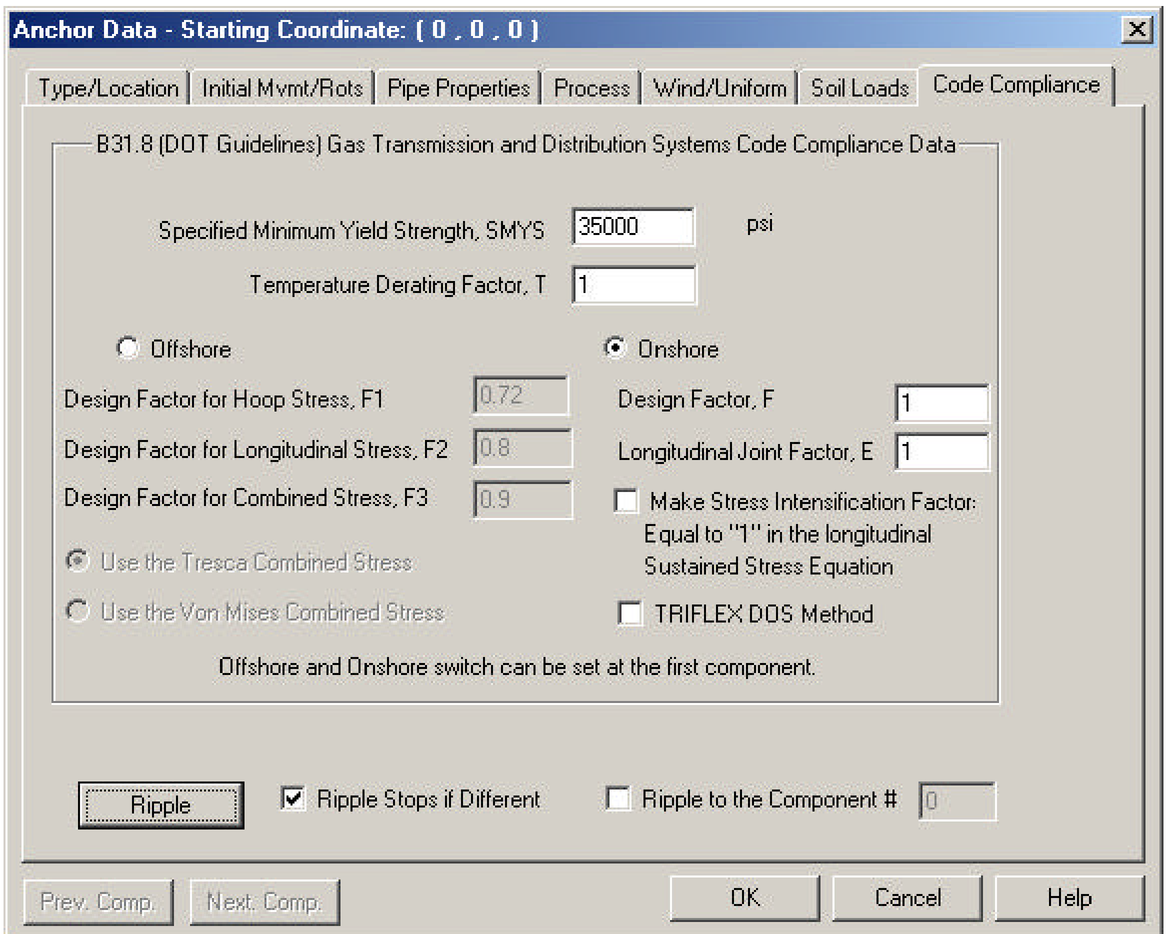


Figure 3.4.5.4 - Anchor Component, Code Compliance Tab, B31.8

<u>SMYS</u> – In this field, the User must enter a value for the specified Minimum Yield Strength of the pipe to be covered by the Code Compliance. The default value is 20,000 psi.

Design Factor, F – In this field, the User should enter the design factor for steel pipe as described in DOT Section 192.111. The default value is 1.0.

Joint Factor, E – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. The default value is 1.0.

Temperature De-rating Factor, T – In this field, the User should enter the temperature de-rating factor as described in DOT Section 192.115. The default value is 1.0.

OFFSHORE – This field is a check box field. The default is with this check box not checked – in other words, TRIFLEX defaults to onshore piping. If the User wishes TRIFLEX to analyze a portion of a piping system using the offshore criteria, then a check should be placed in the check box for the first component to be analyzed using the offshore criteria.

Design Factor for Hoop Stress, F1 – The User should enter the desired design factor for Hoop Stress as defined in the piping code. A default value of 0.72 will be assumed if the User does not enter a numerical value in this field.

Design Factor for Longitudinal Stress, F2 – The User should enter the desired design factor for Longitudinal Stress as defined in the piping code. A default value of 0.8 will be assumed if the User does not enter a numerical value in this field.

Design Factor for Combined Stress, F3 – The User should enter the desired design factor for Combined Stress as defined in the piping code. A default value of 0.9 will be assumed if the User does not enter a numerical value in this field.

The last two lines of this dialog are radio buttons that provide the User with the ability to select between the two stress equations that are available in TRIFLEX for calculating combined stresses. The first radio button is “Use the Tresca Combined Stress - Offshore”. When the User selects Offshore, TRIFLEX will default to “Use the Tresca Combined Stress - Offshore” radio button being selected. The second radio button is “Use the Von Mises Combined Stress - Offshore”. If the User desires the combined stresses to be calculated using the Von Mises equation, then the User should check the second radio button.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.5 U.S Navy General Specifications for Ships, Section 505

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "U.S. Navy General Specifications for Ships, Section 505". The fields in which data can be entered in this data group are defined below

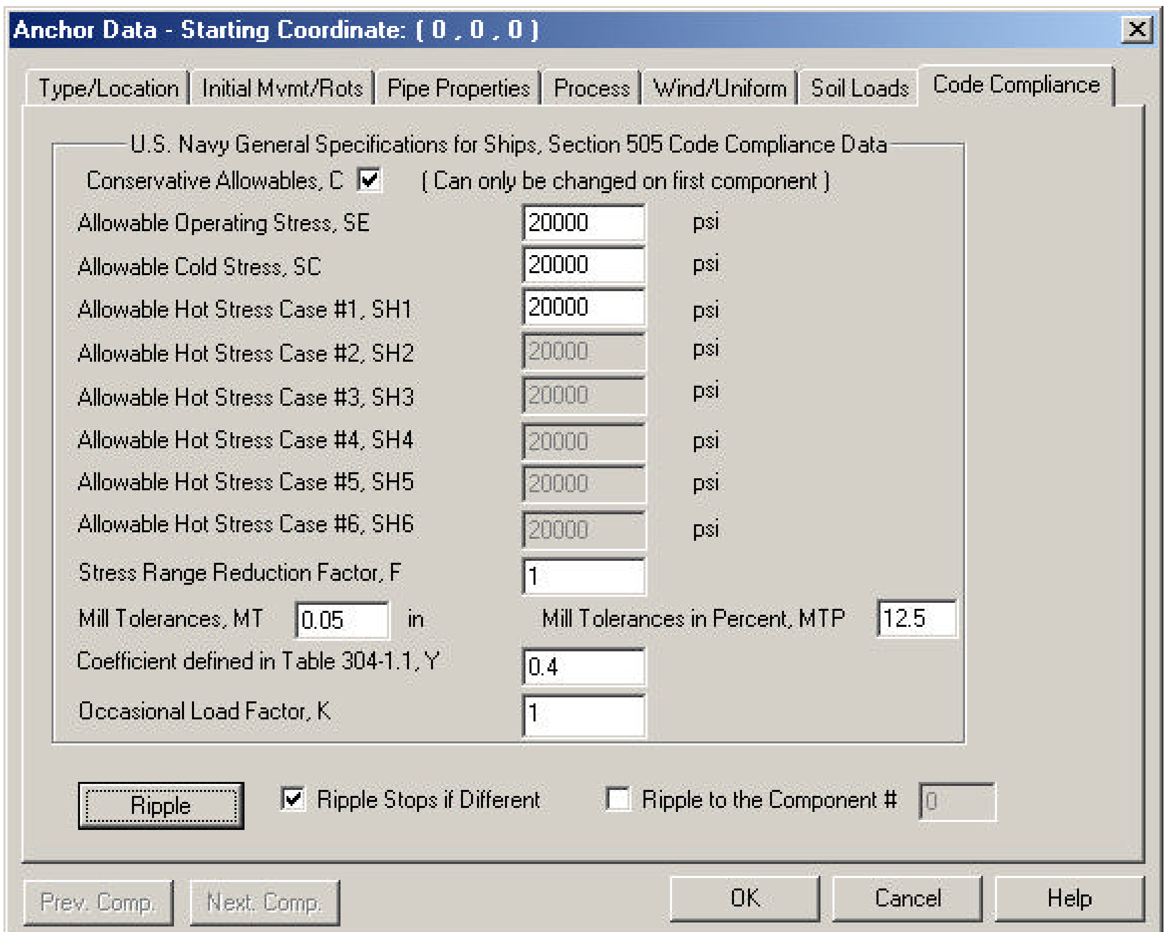


Figure 3.4.5.5 - Anchor Component, Code Compliance Tab, US Navy

Allowable Operating Stress, SE – In this field, the User must enter a value for the allowable operating stress based upon the piping materials selected by the User. The default value is 20,000 psi.

Allowable Cold Stress, S_c – In this field, the User must enter a value for the allowable cold stress based upon the piping materials selected by the User. The default value is 20,000 psi.

The remaining allowable stress data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the hot allowable stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case. The default value is 20,000 psi.

Stress Range Reduction Factor, F – The User should enter the desired stress range reduction factor, based upon the number of cycles the piping system is expected to be subjected to. A default value of 1.0 will be assumed if the User does not enter this factor.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent or in inches / mm. The default is 12 1/2 percent.

Coefficient in Code Book, Y - The User should enter the desired “Y” factor, based upon the codebook. A default value of 0.4 will be assumed if the User does not enter this factor. See Table 304-1.1 of Section 505 of the U.S. Navy Piping Code.

Occasional Load Factor, K – The User can enter a value for the occasional load factor, if desired. A value of 1.15 should be entered for occasional loads acting less than 10 percent of the operating period and a value of 1.2 should be entered for loads acting for less than one percent of the operating period. A default value of 1.2 will be assumed, if the User does not specify the factor.

Joint Factor, E – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. The default value is 1.0.

3.4.5.6 ASME Section III, Division I (Subsection NC) – Class 2

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled “ASME Class 2, Section III, Subsection NC Compliance Data”. The fields in which data can be entered in this data group are defined below:

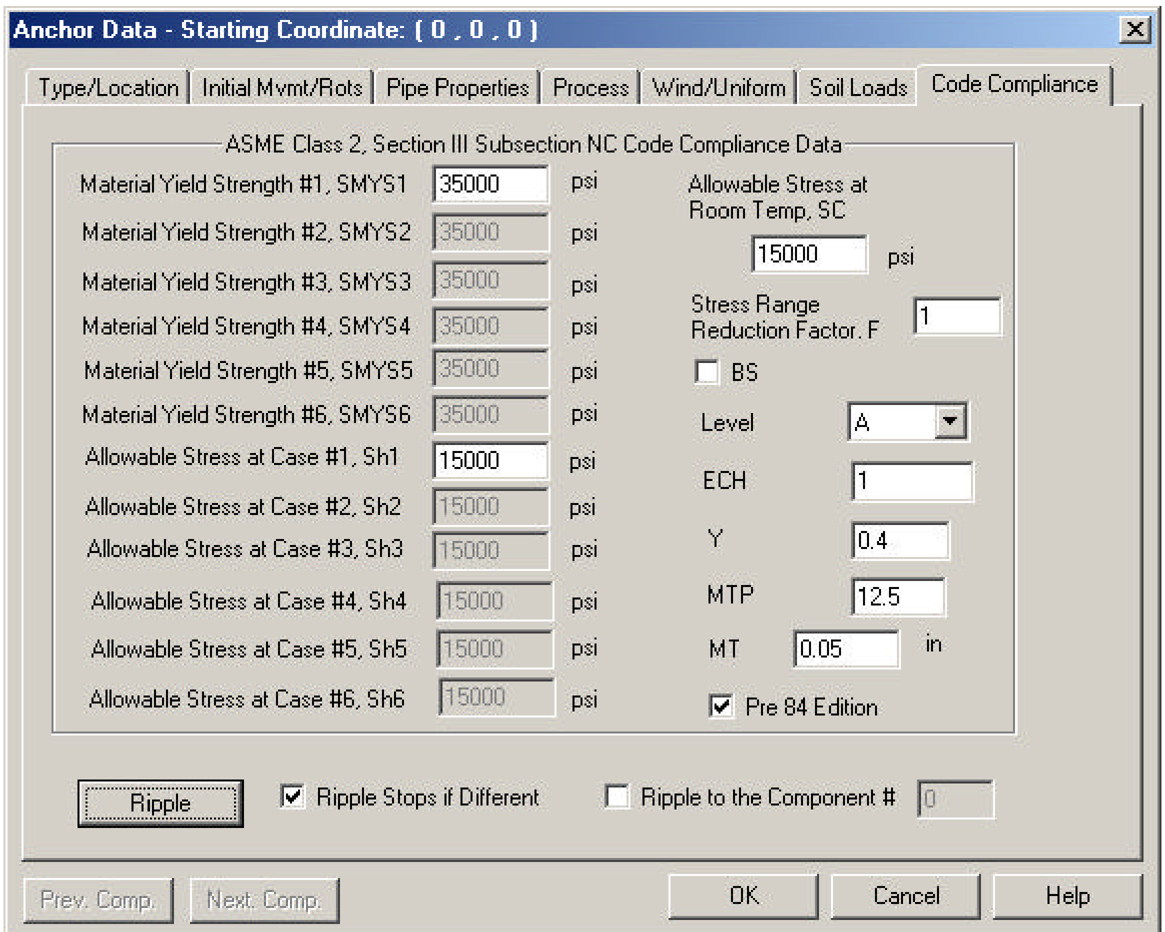


Figure 3.4.5.6 - Anchor Component, Code Compliance Tab, Class 2

SMYS – In each of the fields for each of the active cases, the User must enter a value for the specified Minimum Yield Strength of the pipe to be covered by the Code Compliance. The default value is 35,000 psi to activate a case; the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). The default value is 15,000 psi

Allowable Stress at Room Temperature, S_c – In this field, the User must enter a value for the allowable stress at room temperature based upon the piping materials selected by the User. The default value is 15,000 psi

Stress Range Reduction Factor, F – The User should enter the desired stress range reduction factor, based upon the total number N of full temperature cycles over total number of years during which system is expected to be in service, from Table NC-3611.2 (e)-1. A default value of 1.0 will be assumed if the User does not enter this factor.

Building Settlement - When the User places a check in the check box entitled "BS"; TRIFLEX will perform the stress analysis considering the specified anchor movements to be non-repeated anchor movements. In other words, TRIFLEX will treat the entered anchor movements as predicted building settlement and TRIFLEX will apply equation (10a) rather than equation (10).

Level – In accordance with NC-3611.1, the User may select one of the levels of service from the drop down combo list in this field. The choices for the Stress Limits are A, B, C, or D. The default is Stress Limit A.

ECH - ASME CLASS 1 NB-3672.5 allows the use of the operating modulus to determine the actual moments and forces. In this field, the User should enter the ratio of the installed modulus of elasticity over the operating modulus of elasticity. In order to generate the correct stress values, TRIFLEX will multiply the calculated expansion stresses by this ratio. A default value of 1.0 will be assumed if the User does not enter this factor.

Coefficient Y - The User should enter the desired "Y" factor, based upon the piping codebook. A default value of 0.4 will be assumed if the User does not enter this factor.

MTP - Mill Tolerance Percentage – The User may enter a value for the mill tolerance in percentage of the wall thickness. The default is 12 1/2 percent.

MT - Mill Tolerance – The User may enter a value for the mill tolerance in inches / mm. The default is 0.05 inches or 1.27 mm.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.7 ASME Section III, Division I (Subsection ND) – Class 3

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "ASME Class 3, Section III, Subsection ND Compliance Data". The fields in which data can be entered in this data group are defined below:

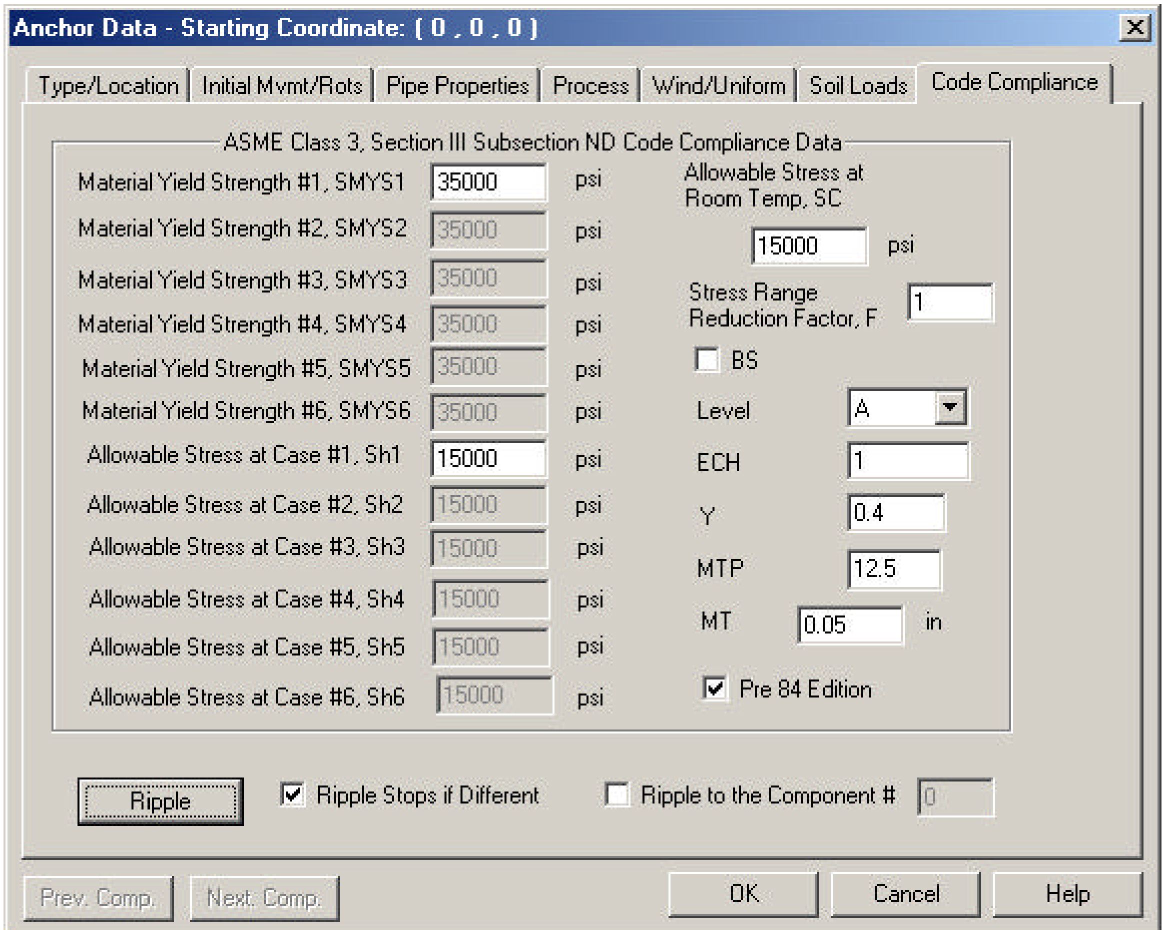


Figure 3.4.5.7 - Anchor Component, Code Compliance Tab, Class 3

SMYS – In each of the fields for each of the active cases, the User must enter a value for the specified Minimum Yield Strength of the pipe to be covered by the Code Compliance. The default value is 35,000 psi to activate a case; the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). The default value is 15,000 psi

Allowable Stress at Room Temperature, S_c – In this field, the User must enter a value for the allowable stress at room temperature based upon the piping materials selected by the User. The default value is 15,000 psi.

Stress Range Reduction Factor, F – The User should enter the desired stress range reduction factor, based upon the total number N of full temperature cycles over total number of years during which system is expected to be in service, from Table ND-3611.2 (e)-1. A default value of 1.0 will be assumed if the User does not enter this factor.

Building Settlement - When the User places a check in the check box entitled "BS"; TRIFLEX will perform the stress analysis considering the specified anchor movements to be non-repeated anchor movements. In other words, TRIFLEX will treat the entered anchor movements as predicted building settlement and TRIFLEX will apply equation (10a) rather than equation (10).

Level – In accordance with ND-3611.1, the User may select one of the levels of service from the drop down combo list in this field. The choices for the Stress Limits are A, B, C, or D. The default is Stress Limit A.

ECH - ASME CLASS 1 NB-3672.5 allows the use of the operating modulus to determine the actual moments and forces. In this field, the User should enter the ratio of the installed modulus of elasticity over the operating modulus of elasticity. In order to generate the correct stress values, TRIFLEX will multiply the calculated expansion stresses by this ratio. A default value of 1.0 will be assumed if the User does not enter this factor.

Coefficient Y - The User should enter the desired "Y" factor, based upon the piping codebook. A default value of 0.4 will be assumed if the User does not enter this factor.

MTP - Mill Tolerance Percentage – The User may enter a value for the mill tolerance in percentage of the wall thickness. The default is 12 1/2 percent.

MT - Mill Tolerance – The User may enter a value for the mill tolerance in inches / mm. The default is 0.05 inches or 1.27 mm.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.8 SPC1 - Swedish Piping Code (Method 1, Section 9.4)

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "Swedish Piping Code (Method 1 Section 9.4) Compliance Data". The fields in which data can be entered in this data group are defined below:

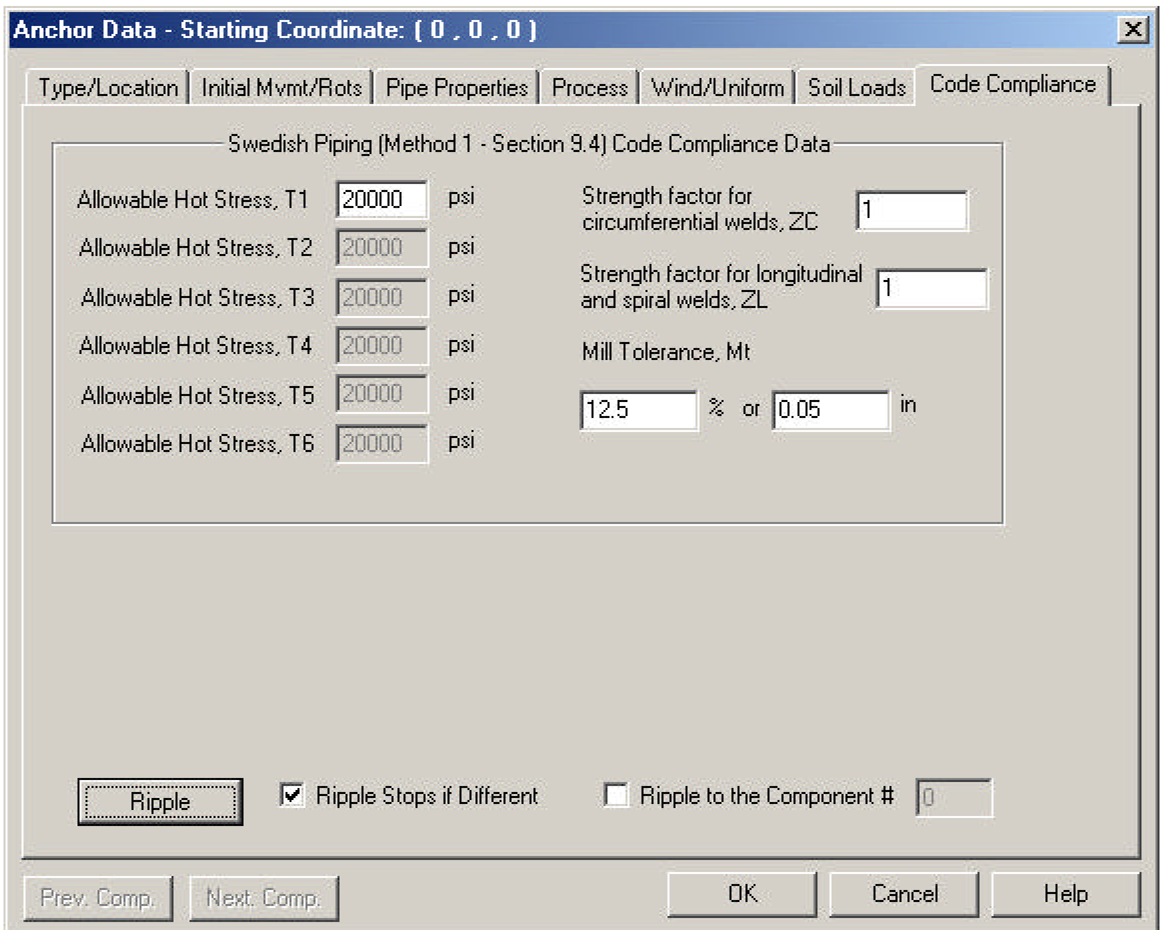


Figure 3.4.5.8 - Anchor Component, Code Compliance Tab, SPC1

The allowable stress data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the hot allowable stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). The default value is 20,000 psi.

Strength Factor for Circumferential Welds, Z_c – The User should enter the desired strength factor for Circumferential Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Strength Factor for Longitudinal Welds, Z_l – The User should enter the desired strength factor for Longitudinal and Spiral Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent in the field provided on the left or in inches / mm in the field provided on the right. A numerical value may only be entered one of the two fields. The default value is 12 1/2 percent.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.9 SPC2 - Swedish Piping Code (Method 2, Section 9.5)

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled “Swedish Piping Code (Method 2 Section 9.5) Compliance Data”. The fields in which data can be entered in this data group are defined below:

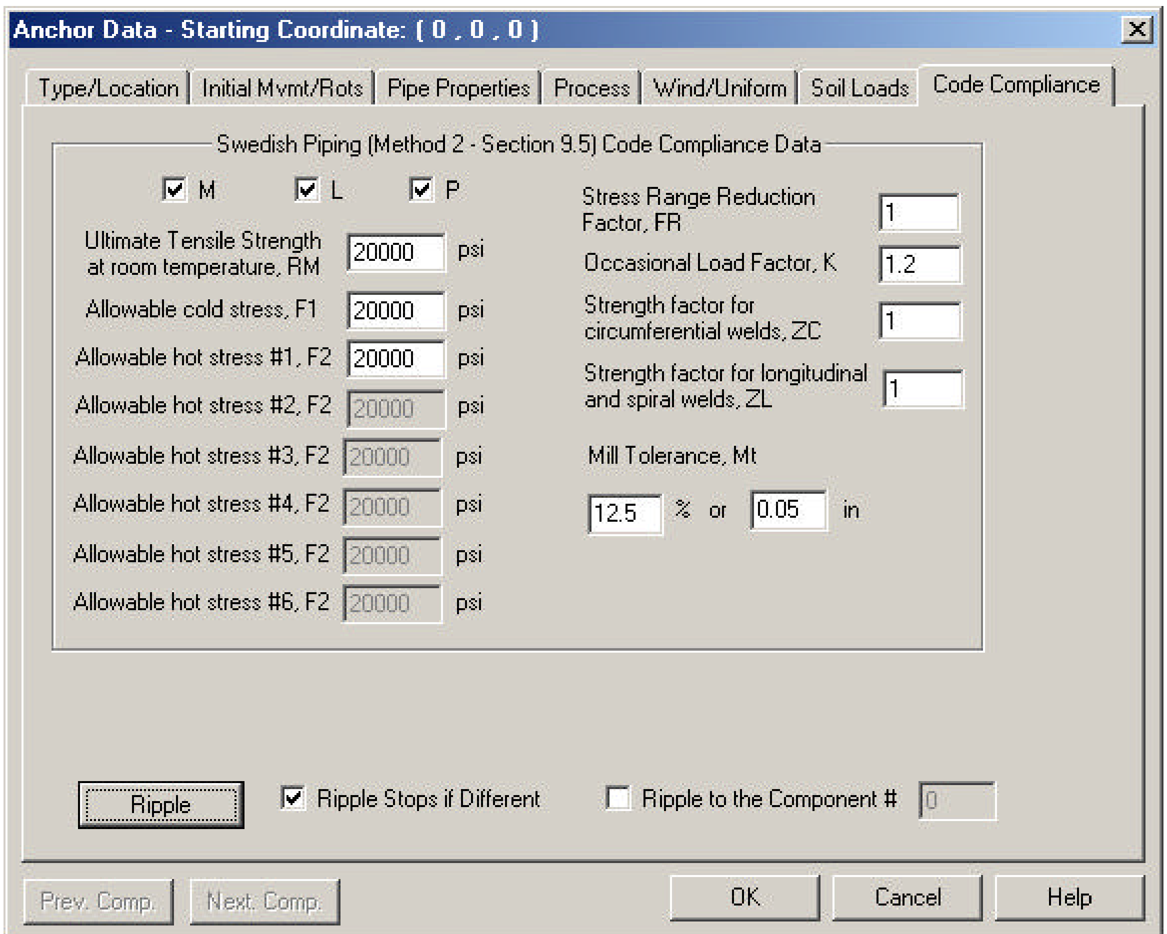


Figure 3.4.5.9 - Anchor Component, Code Compliance Tab, SPC2

M - When the User places a check in the check box entitled “M”; TRIFLEX will perform the stress analysis using the alternate method of determining S_r (allowable range of stress). The program will select the smaller of S_r and S_r as calculated by equations 9:43 and 9:44 respectively.

L - When the User places a check in the check box entitled "L"; TRIFLEX will perform the stress analysis using the liberal equation in determining the Allowable for Loads related to displacement [equation (9:40)].

P - When the User places a check in the check box entitled "P", TRIFLEX will perform the stress analysis using the alternate pressure term, as shown in paragraph 9.5.3.2, in equations 9:37, 9:38, and 9:40.

RM – In this field, the User must enter a value for the Ultimate Tensile Strength of the pipe at room temperature to be covered by the Code Compliance. The default value is 35,000 psi.

Allowable Cold Stress, F1 – In this field, the User must enter a value for the allowable stress at room temperature based upon the piping materials selected by the User. The default value is 20,000 psi.

The allowable hot stress data that can be entered by the User on this dialog is required on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the allowable hot stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, F2 – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). The default value is 20,000 psi.

Stress Range Reduction Factor, FR – The User should enter the desired stress range reduction factor, based upon the total number N of full temperature cycles over total number of years during which the piping system is expected to be in service. A default value of 1.0 will be assumed if the User does not enter this factor.

Occasional Load Factor, K – The User can enter a value for the occasional load factor, if desired. A value of 1.15 should be entered for occasional loads acting less than 10 percent of the operating period and a value of 1.2 should be entered for loads acting for less than one percent of the operating period. A value of 1.2 will be assumed, if the User does not specify the factor.

Strength Factor for Circumferential Welds, Z_e – The User should enter the desired strength factor for Circumferential Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Strength Factor for Longitudinal Welds, Zl – The User should enter the desired strength factor for Longitudinal and Spiral Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent in the field provided on the left or in inches / mm in the field provided on the right. A numerical value may only be entered one of the two fields. The default value is 12 1/2 percent.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.10 TBK5-6 - Norwegian General Rules for Piping System (Annex D-Alternative Method)

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "Norwegian Piping Code TBK 5 – 6 Alternative Method". The fields in which data can be entered in this data group are defined below:

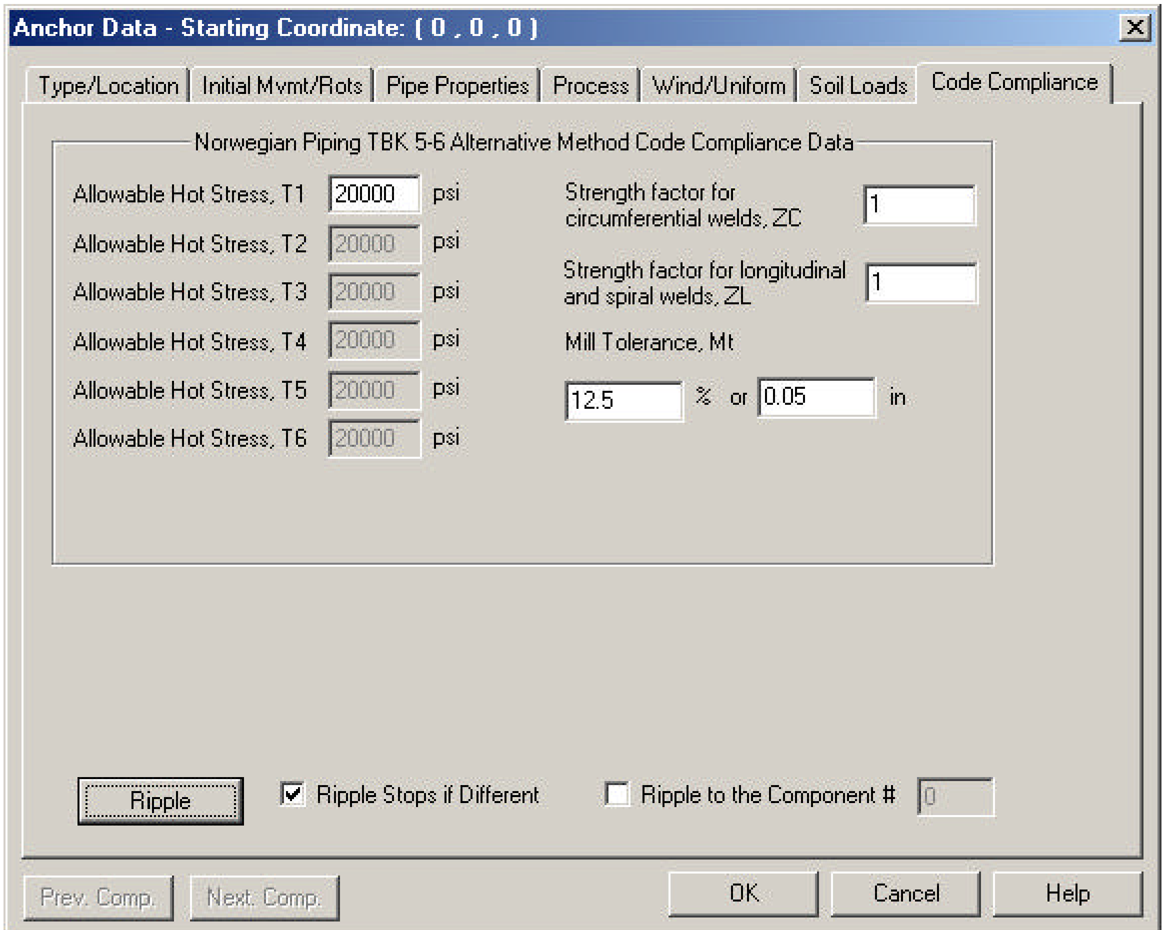


Figure 3.4.5.10 - Anchor Component, Code Compliance Tab, TBK 56

The allowable stress data that can be entered by the User on this dialog is requested on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the hot allowable stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, S_h – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). The default value is 20,000 psi.

Strength Factor for Circumferential Welds, Z_c – The User should enter the desired strength factor for Circumferential Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Strength Factor for Longitudinal Welds, Z_l – The User should enter the desired strength factor for Longitudinal and Spiral Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent in the field provided on the left or in inches / mm in the field provided on the right. A numerical value may only be entered one of the two fields. The default value is 12 1/2 percent.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.11 TBK5-6 - Norwegian General Rules for Piping System (Section 10.5)

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "Norwegian Piping TBK 5 – 6 (Method 2) Code Compliance Data". The fields in which data can be entered in this data group are defined below:

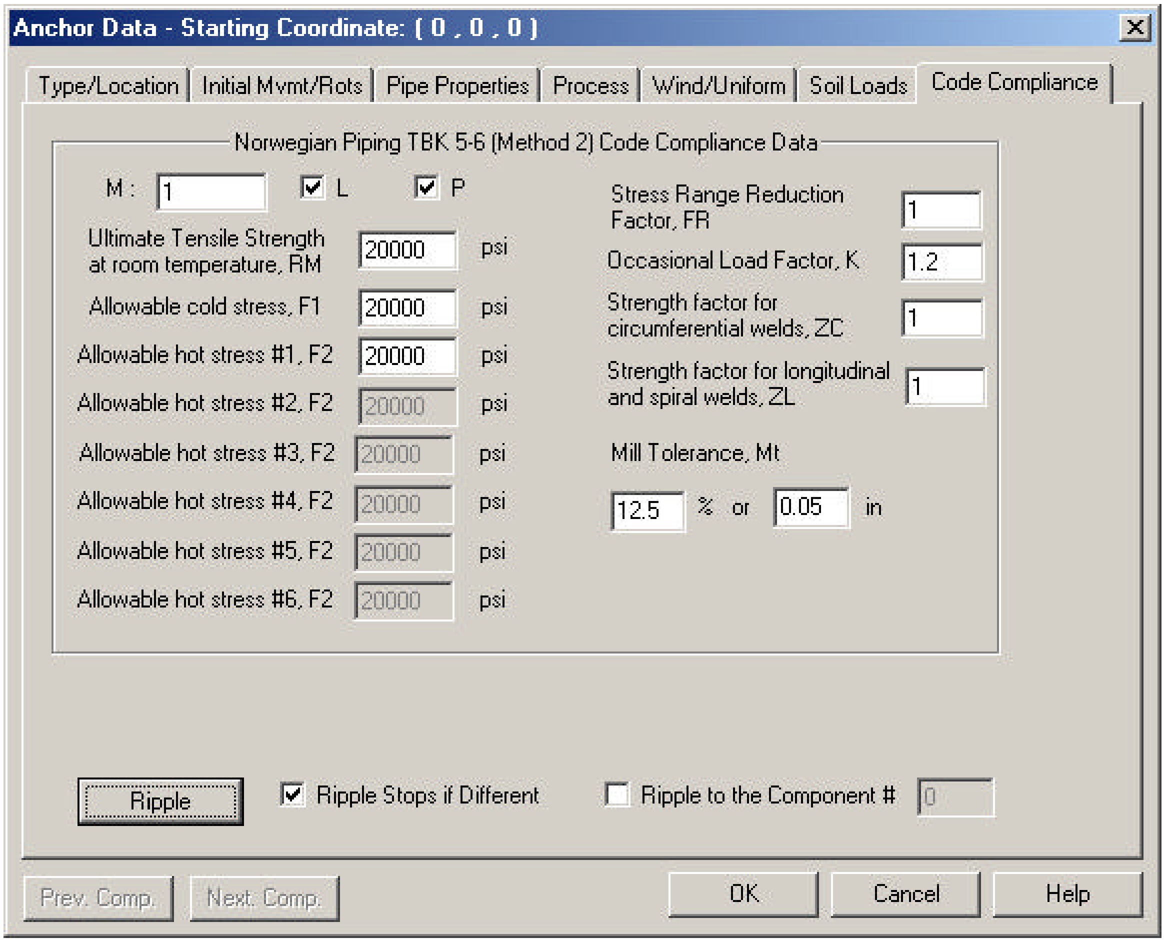


Figure 3.4.5.11 - Anchor Component, Code Compliance Tab, TBK 5-6 Method 2

M - When the User places a check in the check box entitled "M"; TRIFLEX will perform the stress analysis using the lower temperature equations for Sr

Lesser of:

$S_r = 1{,}25\ f_1 + 0{,}25\ f_2$ **or** $S_r = f_r\ R_s - f_2$

Based on the corresponding information in Table 2 from the TBK5-6, 1990 Codebook.

R_s

1	=	Carbon Steel	- 290 N/mm^2
2	=	Austenitic Stainless Steel	- 400 N/mm^2
3	=	Copper alloys, annealed	- 150 N/mm^2
4	=	Copper alloys, cold worked	- 100 N/mm^2
5	=	Aluminum	- 130 N/mm^2
6	=	Titanium	- 300 N/mm^2

When the User leaves the check box entitled "M" unchecked or blank, TRIFLEX will perform the stress analysis using the following equation for high temperatures.

$S_r = f_r\ (1{,}25\ R_1 + 0{,}25\ R_2)$

The default is with this check box checked – in other words, TRIFLEX uses the lower temperature equations for S_r.

L - When the User places a check in the check box entitled "L"; TRIFLEX will perform the stress analysis using the liberal equation in determining the Allowable for Loads related to displacement [equation (9:35)]. The default is with this check box checked – in other words, TRIFLEX defaults to using the liberal equation

P - When the User places a check in the check box entitled "P", TRIFLEX will perform the stress analysis using the alternate pressure term, as shown in paragraph 9.5.3.2, in equations 9:32, 9:33, and 9:35. The default is with this check box checked – in other words, TRIFLEX defaults to using the alternate pressure term.

RM – In this field, the User must enter a value for the Ultimate Tensile Strength of the pipe at room temperature to be covered by the Code Compliance. The default value is 35,000 psi

Allowable Cold Stress, F1 – In this field, the User must enter a value for the allowable stress at room temperature based upon the piping materials selected by the User. The default value is 20,000 psi.

The allowable hot stress data that can be entered by the User on this dialog is required on a case-by-case basis. In this release of TRIFLEX, six cases can be specified and processed one at a time or in combination. When the User has elected to activate a load case, the User must specify the allowable hot stress value for that particular load case. To activate a case, the User must go to Setup on the main menu, then Case Definition and then must place a check mark in the Process this Case check box for the desired case.

Allowable Hot Stress, F2 – In this field for each activated load case, the User must enter a value for the allowable hot stress based upon the piping materials selected by the User. Data can only be entered in an active field (one that is not grayed out). The default value is 20,000 psi.

Stress Range Reduction Factor, FR – The User should enter the desired stress range reduction factor, based upon the total number N of full temperature cycles over total number of years during which the piping system is expected to be in service. A default value of 1.0 will be assumed if the User does not enter this factor.

Occasional Load Factor, K – The User can enter a value for the occasional load factor, if desired. A value of 1.15 should be entered for occasional loads acting less than 10 percent of the operating period and a value of 1.2 should be entered for loads acting for less than one percent of the operating period. A value of 1.2 will be assumed, if the User does not specify the factor.

Strength Factor for Circumferential Welds, Z_c – The User should enter the desired strength factor for Circumferential Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Strength Factor for Longitudinal Welds, Z_l – The User should enter the desired strength factor for Longitudinal and Spiral Welds as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance, Mt – The User may enter a value for the mill tolerance in percent in the field provided on the left or in inches / mm in the field provided on the right. A numerical value may only be entered one of the two fields. The default value is 12 1/2 percent.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.12 DNV –Submarine Pipeline System -DnV, 1981 Edition

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled “DnV Rules for Submarine Pipeline Systems Compliance Data, 1981 Edition”. The fields in which data can be entered in this data group are defined below:

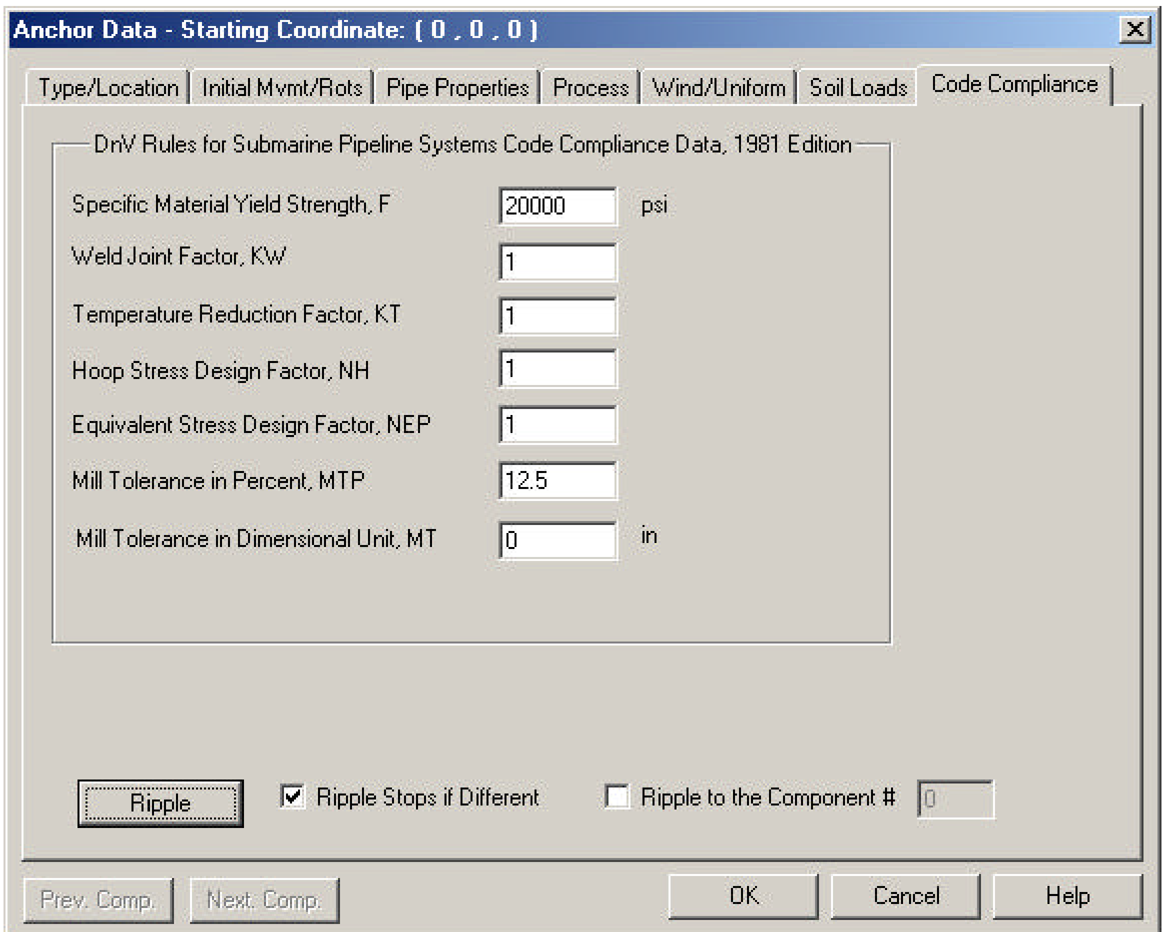


Figure 3.4.5.12 - Anchor Component, Code Compliance Tab, DNV 1981

<u>Specific Material Yield Strength, F</u> – In this field, the User must enter a value for the Specific Material Yield Strength of the pipe to be covered by the Code Compliance. A default value of 35,000 psi will be assumed if the User does not enter a value.

Weld Joint Factor, KW – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. A default value of 1.0 will be assumed if the User does not enter a value.

Temperature Reduction Factor, KT – In these fields, the User should enter the temperature reduction factor(s) applicable for this piping component. A default value of 1.0 will be assumed if the User does not enter a value.

Hoop Stress Design Factor, NH – The User should enter the desired hoop stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Equivalent Stress Design Factor, NEP – The User should enter the desired equivalent stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance in Percent, MTP – The User may enter a value for the mill tolerance in percent in this field. The default value is zero percent. A numerical value may only be entered in this field or in the following mill tolerance field, but not both.

Mill Tolerance in Dimensional Unit, MT – The User may enter a value for the mill tolerance in inches, mm or cm in this field. The default value is zero. A numerical value may only be entered in this field or in the previous mill tolerance field, but not both.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.13 DNV –Submarine Pipeline System -DnV, 1996 Edition

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "DnV Rules for Submarine Pipeline Systems Compliance Data, 1996 Edition". The fields in which data can be entered in this data group are defined below:

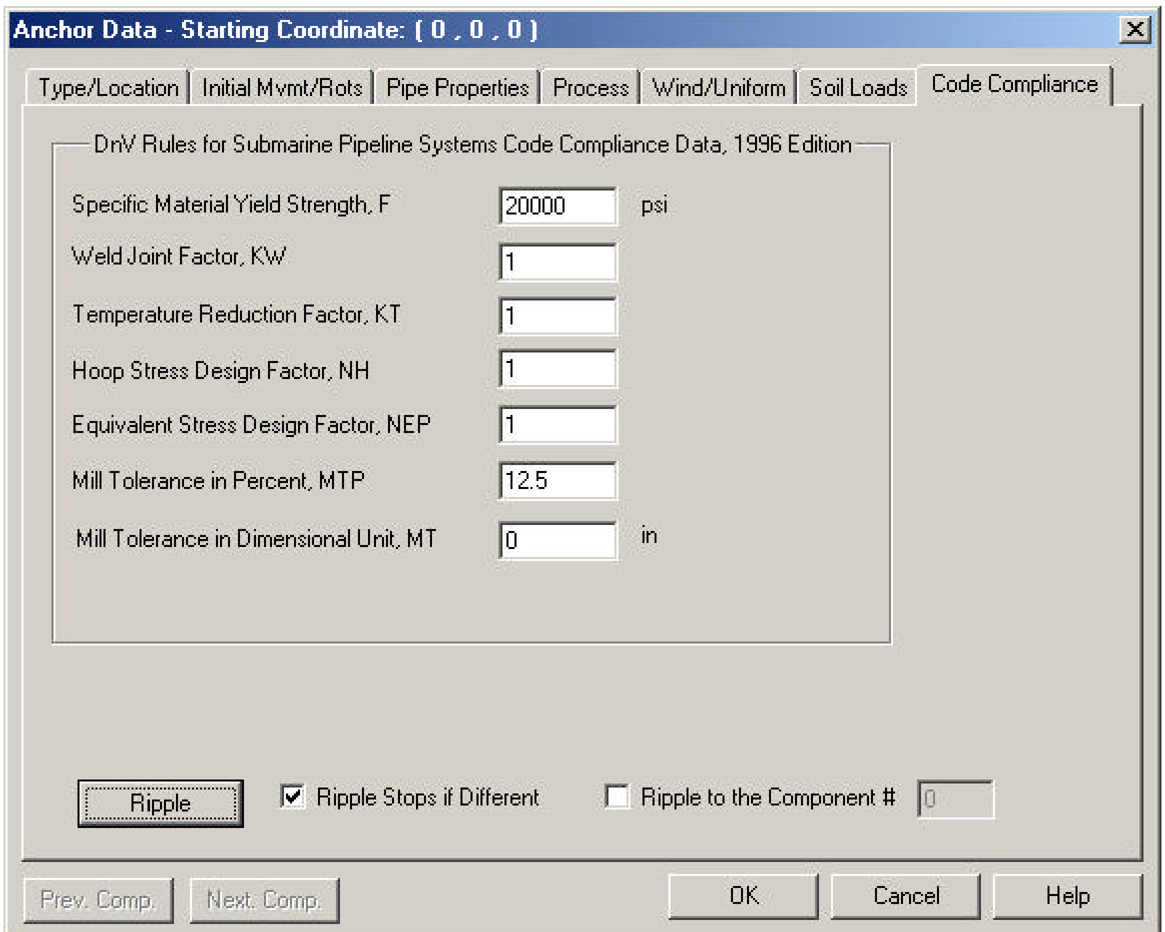


Figure 3.4.5.13 - Anchor Component, Code Compliance Tab, DNV 1996

<u>Specific Material Yield Strength, F</u> – In this field, the User must enter a value for the Specific Material Yield Strength of the pipe to be covered by the Code Compliance. A default value of 35,000 psi will be assumed if the User does not enter a value.

Weld Joint Factor, KW – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. A default value of 1.0 will be assumed if the User does not enter a value.

Temperature Reduction Factor, KT – In these fields, the User should enter the temperature reduction factor(s) applicable for this piping component. A default value of 1.0 will be assumed if the User does not enter a value.

Hoop Stress Design Factor, NH – The User should enter the desired hoop stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Equivalent Stress Design Factor, NEP – The User should enter the desired equivalent stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance in Percent, MTP – The User may enter a value for the mill tolerance in percent in this field. The default value is zero percent. A numerical value may only be entered in this field or in the following mill tolerance field, but not both.

Mill Tolerance in Dimensional Unit, MT – The User may enter a value for the mill tolerance in inches, mm or cm in this field. The default value is zero. A numerical value may only be entered in this field or in the previous mill tolerance field, but not both.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.14 DNV – Offshore Standard OS-F101 Submarine Pipeline System - DnV, 2000 Edition

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled “OSF 101”. The fields in which data can be entered in this data group are defined below:

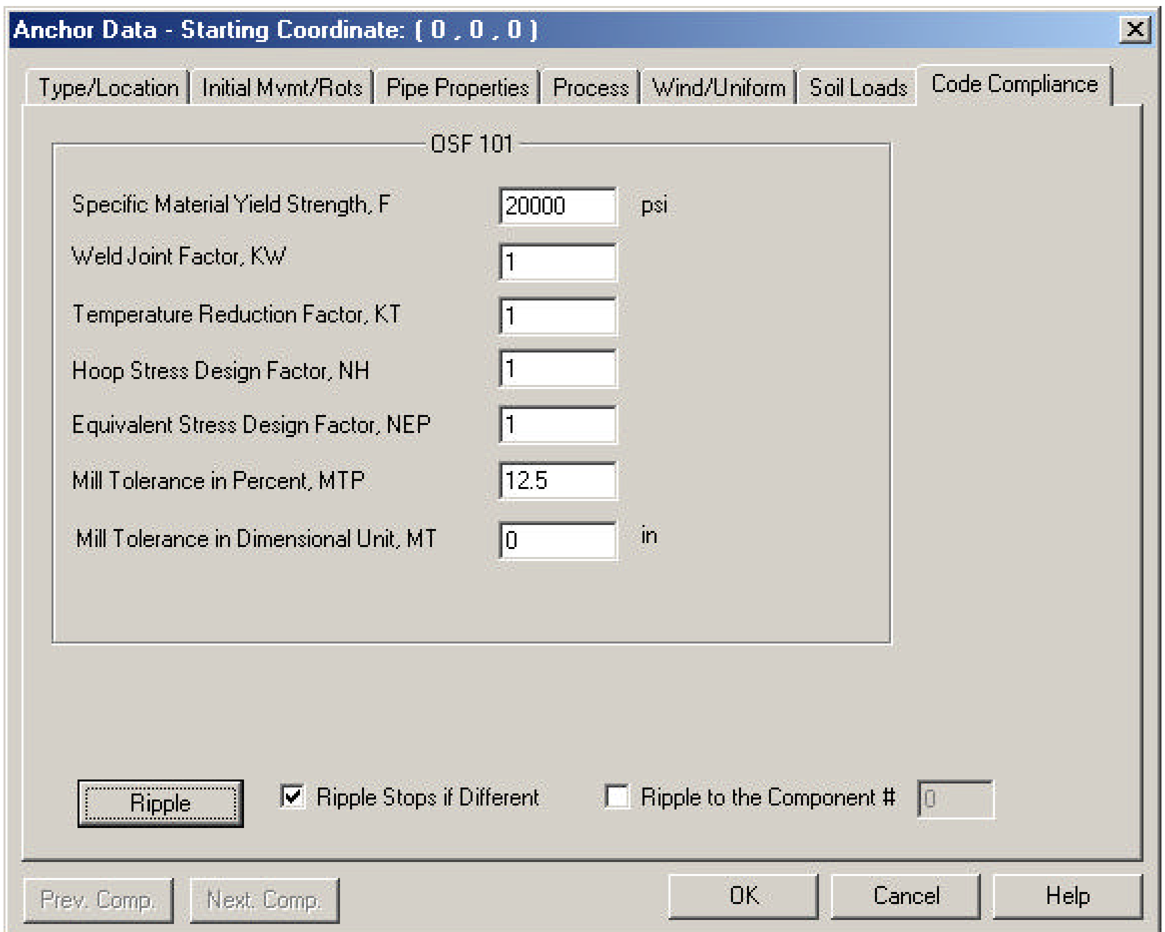


Figure 3.4.5.14 - Anchor Component, Code Compliance Tab, DNV 2000

<u>Specific Material Yield Strength, F</u> – In this field, the User must enter a value for the Specific Material Yield Strength of the pipe to be covered by the Code Compliance. A default value of 35,000 psi will be assumed if the User does not enter a value.

Weld Joint Factor, KW – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. A default value of 1.0 will be assumed if the User does not enter a value.

Temperature Reduction Factor, KT – In these fields, the User should enter the temperature reduction factor(s) applicable for this piping component. A default value of 1.0 will be assumed if the User does not enter a value.

Hoop Stress Design Factor, NH – The User should enter the desired hoop stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Equivalent Stress Design Factor, NEP – The User should enter the desired equivalent stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance in Percent, MTP – The User may enter a value for the mill tolerance in percent in this field. The default value is zero percent. A numerical value may only be entered in this field or in the following mill tolerance field, but not both.

Mill Tolerance in Dimensional Unit, MT – The User may enter a value for the mill tolerance in inches, mm or cm in this field. The default value is zero. A numerical value may only be entered in this field or in the previous mill tolerance field, but not both.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.15 Polska Norma PN-79 / M-34033

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

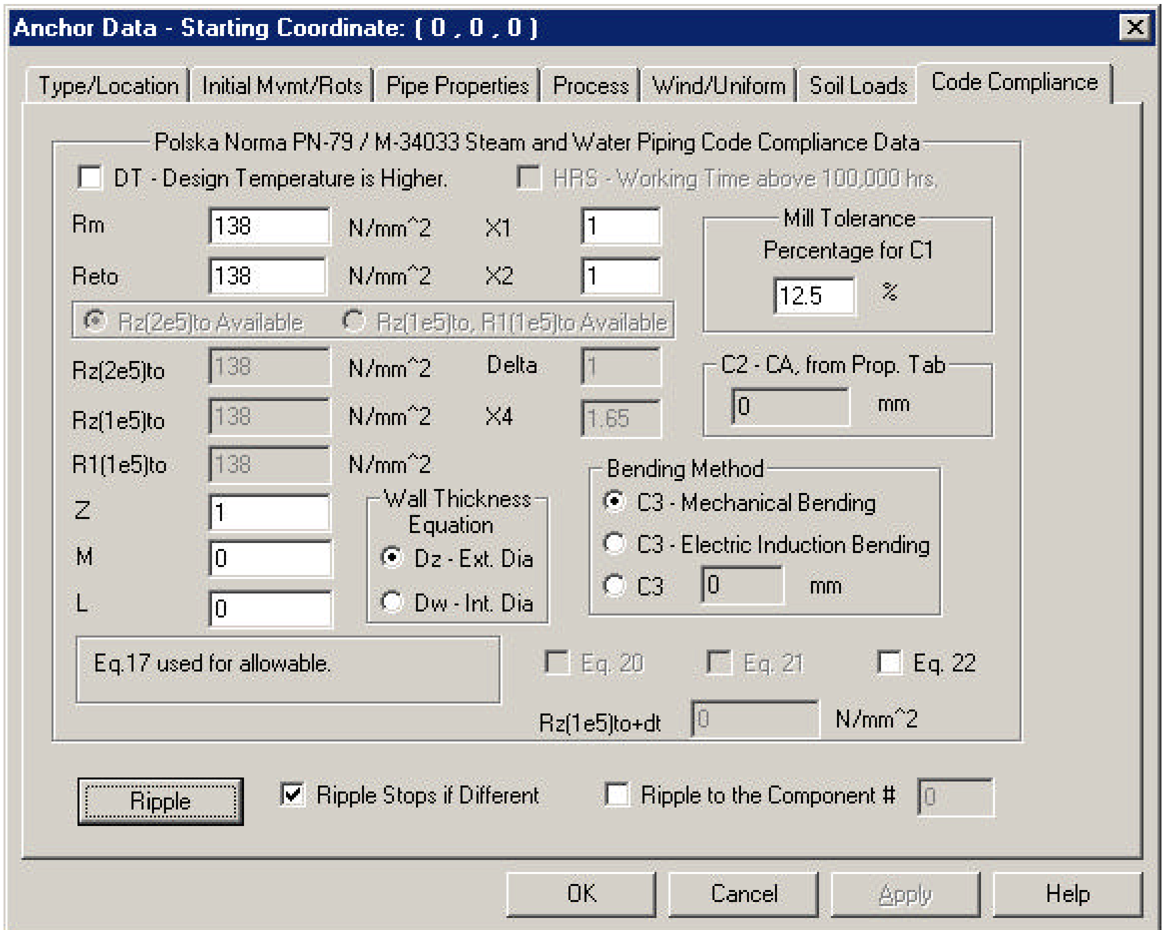


Figure 3.4.5.15 - Anchor Component, Code Compliance Tab, Polska Norma

DT - Design Temperature is Higher

This field is used to indicate that the design temperature is higher (**H**) if the check box is checked or lower (**L**) than the limit temperature for the pipe material if the check box is unchecked.

HRS – Working Time above 100,000 hrs

This check box will be checked when the user should specify that the piping system will have a working time above 100,000 hrs.

RM (psi, N/mm^2, kg/cm^2, N/mm^2)

RM is the Specified Minimum Tensile strength (minimal value) at room temperature (R_m).

Reto (psi, N/mm^2, kg/cm^2, N/mm^2)

Reto is Real Yield point (minimal value) at design temperature (R_{et_o}).

Rz(2e5)to (psi, N/mm^2, kg/cm^2, N/mm^2)

Rz(2e5)to is the Specified Temporary Creep Strength (average value) at 2*10^5 hours in

Design temperature t_o ($R_{z(2*10^5)t_o}$).

Rz(1e5)to (psi, N/mm^2, kg/cm^2, N/mm^2)

Rz(1e5)to is the Specified Temporary Creep Strength (average value) at 10^5 hours in design

temperature t_o ($R_{z(10^5)t_o}$).

R1(1e5)to (psi, N/mm^2, kg/cm^2, N/mm^2)

R1(1e5)to is the Specified Creep Strength limit (average value) with 1% permanent

elongation, at 10^5 hours and in design temperature t_o ($R_{1(10^5)t_o}$).

Z (Default: 1)

Strength factor of weld connection

1.0 - for seamless pipe

0.9 - for pipes with longitudinal double-sided wall

0.8 - for pipes with longitudinal one side weld as well as for pressure welded

M (Default: 0)

Pipe material (for reference, see table 2) shall be specified as:

0 - Boiler steel pipes

1 - Quality pipes made from C.S. with specified impact strength

2 - Other C.S. pipes

L (Default: 0)

Pressure level (for reference, see table 2) shall be specified as:

0 - pipes destined for pipelines where internal pressure and additional external loads occur.

1 - pipes destined for pipelines where only internal pressure occurs

X1 and X2

X1and X2 is coefficients – accordingly to Table 2 (Chapter 8-Polish Code) – depending on material grade (quality) and working conditions.

Mill Tolerance Percentage for C1 (Default: 12.5%)

Mill tolerance specified as a percent.

Delta **%**

The maximum minus allowance for creep stress value in time of $2x10^5$ hrs at design temperature $t_{o.}$ (Rz(2e5)to)

C2 – CA, from Prop. Tab (in, mm, cm, mm)

> $\mathbf{C_2}$ - Allowance taking account at corrosion influence. For nonagressive water and steam (with no solid particles, which can cause wall thickness abrasion - equals $\mathbf{C_2}$ = (0.3 up to 1.0 mm)

X4

> X_4 = see table 3 (Chapter 8 – Polish Code)
>
> If working time is less than or equal to 100,000 hours X_4 =1.65.

C3 (in, mm, cm, mm)

> Allowance for wall thickness taking into account because of thinning during bending process. There are 3 options:
>
> C3- Mechanical Bending
>
> C3- Electric Induction Bending
>
> C3- Input by User

Wall Thickness Equation

Dz –Ext. Dia – the thickness for wall pipe will be calculated using external diameter Dz

Dw – Int. Dia - the thickness for wall pipe will be calculated using internal diameter Dw

Special Allowable

The user can specify 20, 21, or 22 to indicate the equation to be used to calculate the allowable stress value.

Eq. 20

Equation (20) -For design conditions at indicated pipeline points for periodic material creep control.

Eq. 21

Equation (21) -For a case of maximum short-lived pressure or temperature increase.

Eq. 22

Equation (22) -For hydrostatic test.

Rz(1e5)to+dt

Rz(1e5)to+dt Creep strength (mean value) – when permanent elongation equals to 1% in time of 10^5 hrs at design temperature t_{o+dt} ($R_{z(10^5)\ to + \Delta t}$)

3.4.5.16 SNIP 2.05-06-95 FSU Transmission Piping Code

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

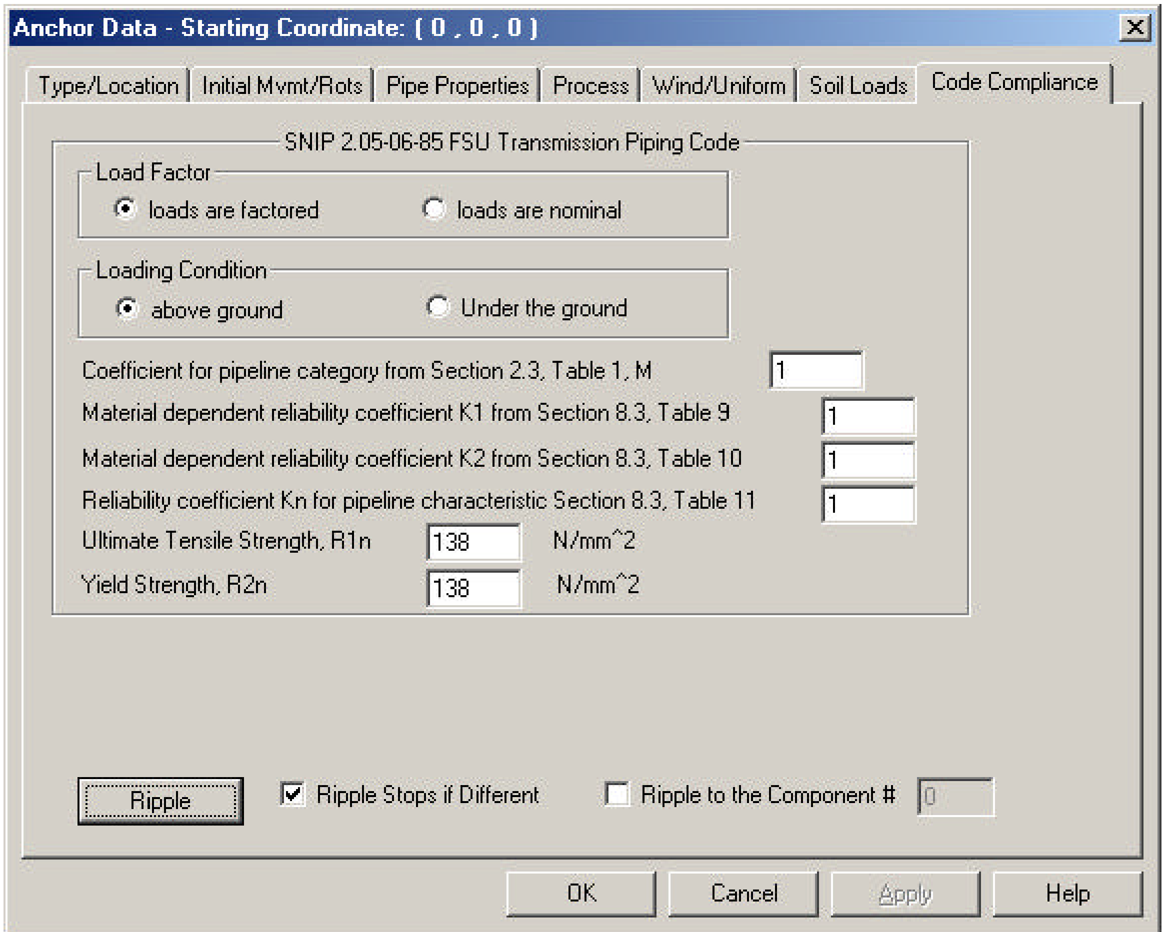


Figure 3.4.5.16 - Anchor Component, Code Compliance Tab, SNIP

LF

Loading Factor. There are two options:

Loads are factored

Loads are nominal

The default option is "Loads are factored"

LC

Loading Condition

There are two options:

- Above ground
- Under ground

The default option is “Above ground”.

M

Coefficient for pipeline category Sec 2.3 Table 1 - M

This filed is used to specify the Coefficient for pipeline category Sec 2.3 Table 1.

K1

Coefficient for pipeline category Sec 2.3 Table 1

This field is used to specify the material dependent reliability coefficient (K1) Sec 8.3 Table 9.

K2

Material dependent reliability coefficient Sec8.3 Table 10 - K2

This field is used to specify the material dependent reliability coefficient (K2) Sec 8.3 Table 10.

KN

Reliability coefficient for pipeline characteristic - KN

This field is used to specify the reliability coefficient for pipeline characteristic (KN) Sec 8.3 Table 11.

R1N

Ultimate Tensile Strength - $R_{(1,n)}$ psi, N/mm^2, kg/cm^2, N/mm^2

This field is used to specify the Ultimate Tensile Strength.

R2N

Yield Strength - $R_{(2,n)}$ psi, N/mm^2, kg/cm^2, N/mm^2

This field is used to specify the Yield Strength.

3.4.5.17 Fiberglass Reinforced Plastic Pipe

When creating a piping system using Fiberglass Reinforced Plastic Pipe the user needs to select the **Modeling Defaults** dialog under the **Setup** command under the **Main Menu**. Once the user has selected the Modeling Defaults screen, there are three changes that need to be implemented:

1) Change the Piping Code to read "BS7159- Fiberglass Reinforced Plastic Pipe";

2) Select the "Includes Translational Pressure deformations" box;

3) Select the "Includes Rotational Pressure Deformation" box. The later two action items tell TRIFLEX that the Bourdon Pressure Effect will be considered in the analysis.

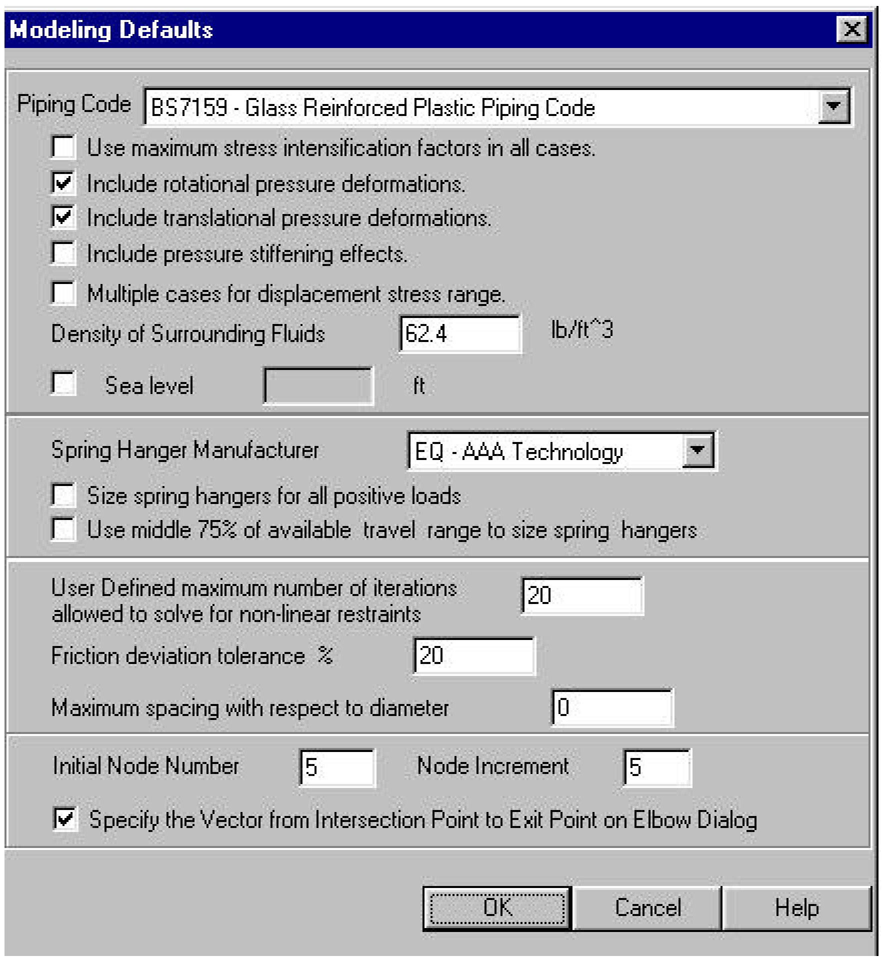


Figure 3.4.5.17.1 – Modeling Default, FRP

Note: Bourdon Pressure Effect-

The Bourdon pressure effect causes: straight pipes to displace along the x axis (elongate) and bends to elongate along the line that connects the bend near and far nodes. The Bourdon effect is always considered when plastic pipe is used. The impact of the Bourdon effect can be appreciable in long pipe runs or high pressures or large diameter bends (especially next to sensitive equipment). The two Bourdon options that are available are:

Option #1 Include rotational pressure deformations

If the elbows or bends are fabricated using hot or cold bending then this will cause a slight oval shape of the bend cross section. This will cause the bend to straighten out when pressurized. Fixed end moments are associated with the opening that do not exist when the original shape of the bend cross section is circular.

Option #2 Include translational pressure deformation

If bend or elbow has a circular cross section such as a system that has forged or welded fitting on the bend or elbow.

Bourdon Effect and TRIFLEX

TRIFLEX allows the user to control two effects of internal pressure. These are the translational deformation due to pressure, and INDEPENDENTLY, the rotational deformation due to pressure. The latter comes to play within the context of elbows. So, the user may control the translational or the rotational effects through the defaults screen.

With a bend, both in-plane directions participate because there is NO unique preferred direction. Actually, there are two, such as entry to the bend, and exit from it. Usually, programs take the entry and an in plane normal.

The next set is to build the piping model. As always TRIFLEX starts each model with an Anchor. However, under the **Piping Properties** of the Anchor dialog screen, **Reinforced Fiberglass** pipe material needs to be checked to do the BS-7159 analysis.

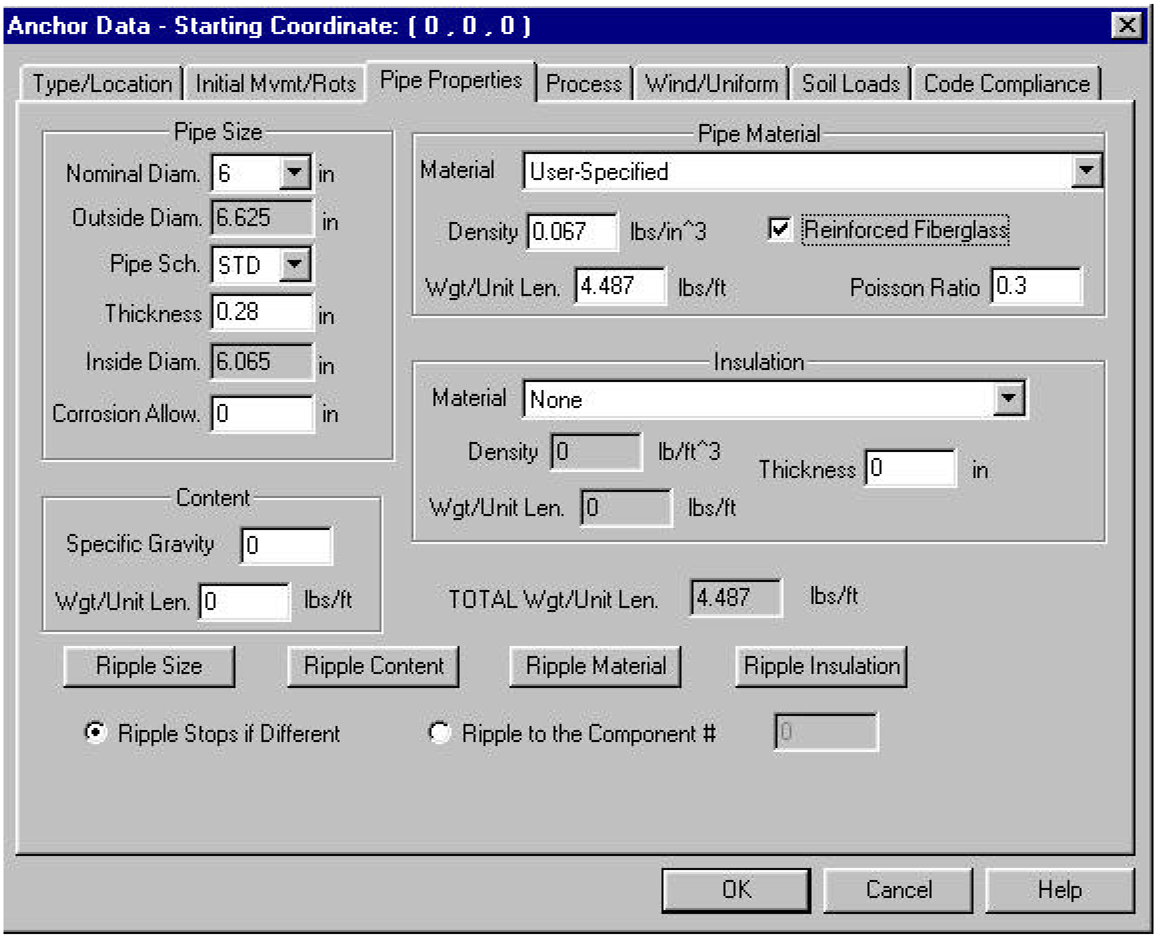


Figure 3.4.5.17.2 - Anchor Component, Pipe Properties Tab, FRP

Recent research has been conducted that shows that physical properties calculated by using the equations given in BS 7159 are likely to yield properties substantially different in magnitude from those you will obtain from the manufacturer of the FRP/GRP pipe you may be using. Therefore, it is highly recommended that each user obtain these properties from the appropriate FRP/GRP pipe manufacturer and use only these properties in the analysis of FRP/GRP piping systems.

Pipe Density (lbs/in^3, N/mm^3 10^4, g/cm^3, kg/m^3)

The user is to enter the density of the FRP/GRP material as obtained from the manufacturer.

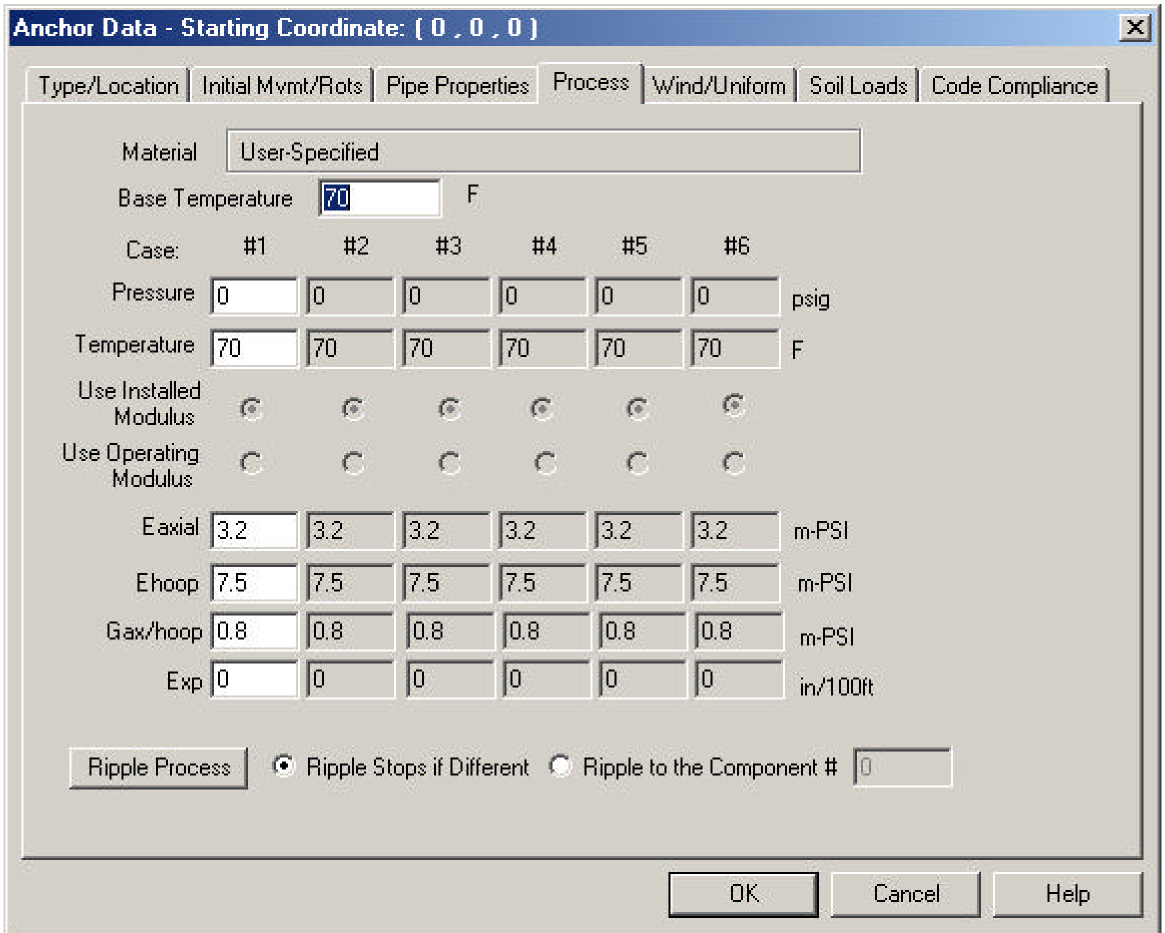


Figure 3.4.5.17.3 - Anchor Component, Process Tab, FRP

Pressure (psig, k-N/m^2, kg/cm^2, bars)

Modulus of Elasticity

Eaxial (m-PSI, k-N/m^2 x 10^{-6}, m-kg/cm^2, m-N/mm^2)

The user is to enter the modulus of elasticity in the axial direction of the Fiber Reinforced Plastic Pipe being modeled.

Ehoop (m-PSI, k-N/m^2 x 10^{-6}, m-kg/cm^2, m-N/mm^2)

The user is to enter the modulus of elasticity in the circumferential (hoop) direction of the Fiber Reinforced Plastic Pipe being modeled.

Gax/hoop (m-PSI, k-N/m^2 x 10^{-6}, m-kg/cm^2, m-N/mm^2)

The user is to enter the modulus of elastic shear between the radial and the hoop directions of the Fiber Reinforced Plastic Pipe being modeled, as it pertains to torsion.

Coefficient of Expansion

Exp (in/100 ft, mm/m, cm/100 m, mm/m)

The coefficient of expansion is to reflect the amount of growth per unit length of pipe. This value is available from the FRP/GRP pipe vendor's catalog. While modeling a fiber reinforced plastic piping system you may only specify T1 in the load case combinations field on the case data screen.

The following guidelines may assist you in properly analyzing your piping systems:

1. The design temperature change for non-insulated pipe systems containing liquids is generally recommended to be eighty-five (85) percent of the difference between the ambient temperature and the process temperature.

2. The design temperature change for non insulated pipe systems containing gases is generally recommended to be eighty (80) percent of the difference between the ambient temperature and the process temperature.

3. The design temperature change for insulated pipe systems containing liquids or gases is generally recommended to be one hundred (100) percent of the difference between the ambient temperature and the process temperature.

4. For piping systems operating at temperatures above the ambient temperature, your base temperature should be taken to be the lowest encountered ambient temperature.

5. For piping systems operating at temperatures below the ambient temperature, your base temperature should be taken to be the highest encountered ambient temperature.

6. The coefficient of expansion for unlined FRP/GRP piping varies between 1.7 and 2.5 times that of carbon steel depending on the type of reinforcement in

the pipe wall. Obtain the proper coefficient of expansion from the manufacturer of the FRP/GRP pipe you are using.

7. The coefficient of expansion for lined FRP/GRP piping systems operating at a relatively low temperature (40 degrees C) is found to be significantly higher than that for unlined FRP/GRP pipe. As the operating temperature increases to 60 degrees C, the effect of the PVC lining decreases. At 60 degrees C and above, the influence of the PVC lining can be ignored.

Poisson's ratio

This ratio is defined by the formula:

$$Axial\ strain = \frac{axial\ stress}{Eaxial} - Pois\ ratio \times \frac{hoop\ stress}{Ehoop}$$

For FRP/GRP pipe, the flexibility and stress intensification factors for smooth and mitered bends are calculated by TRIFLEX based upon the guidelines provided in BS 7159 as follows:

For FRP/GRP pipe, the flexibility and stress intensification factors for smooth and mitered bends are calculated by TRIFLEX based upon the guidelines provided in BS 7159 as follows:

For FRP/GRP pipe, the flexibility and stress intensification factors for molded and fabricated tees are calculated by TRIFLEX based upon the guidelines provided in BS 7159 as follows:

A piping system may consist of both fiberglass reinforced plastic pipes as well as metal (isotropic) pipes. In such cases, take care to change properties at the point where the transition occurs. The applicable stresses for the steel pipes will be found in the stress report following the forces and moments reports and the applicable stresses for the fiberglass reinforced plastic pipes will be found in the BS 7159 Code Compliance report following the standard output.

Remember that it is highly recommended that you contact the manufacturer of the fiberglass reinforced plastic pipes that you are using in your piping system for the exact values for the properties you must use to obtain an accurate piping analysis!

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "DnV Rules for Submarine Pipeline Systems Compliance Data, 1981 Edition". The fields in which data can be entered in this data group are defined below:

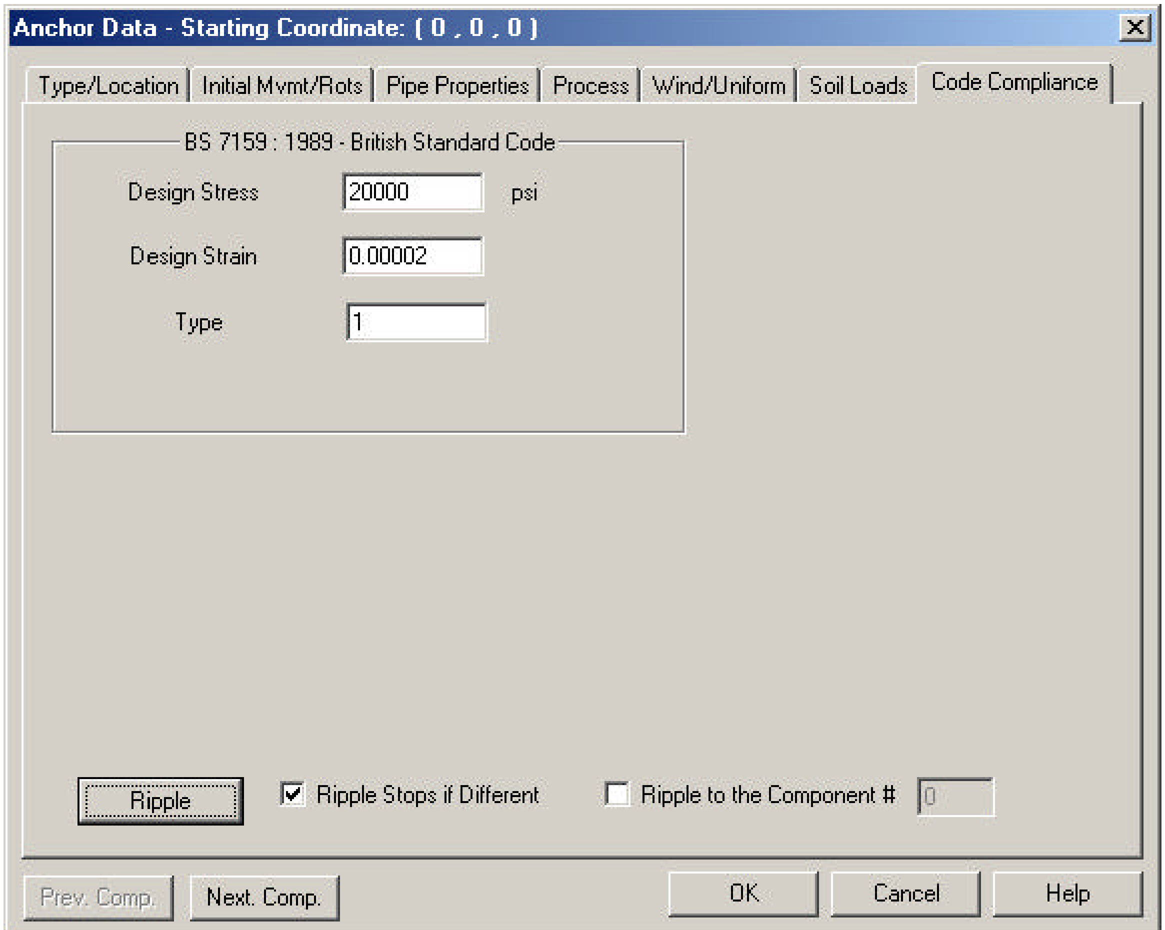


Figure 3.4.5.17.4 - Anchor Component, Code Compliance Tab, FRP

Design Stress (psi, k-N/m^2, kg/cm^2, N/mm^2)

The design stress to be entered by the user is the numeric value of the Maximum Combined Stress as obtained from the FRP/GRP pipe manufacturer.

Design Strain (Unit less)

The design strain (,$_N$) to be entered by the user is the numeric value of the maximum allowed strain as obtained from the FRP/GRP pipe manufacturer. The sum of the circumferential strain induced by pressure and the circumferential tensile strain resulting from the longitudinal compressive stress induced by temperature change shall not exceed the design strain.

Laminate Type (1, 2 or 3)

For specific details concerning the laminate types, please consult the BS 7159 Code for the Design and Construction of Glass Reinforced Plastics Piping Systems for Individual Plants or Sites. Section four of BS 7159 describes the three types of laminates and section seven of BS 7159 describes the flexibility factors and stress intensification factors for bends and branch connections for each laminate type.

Laminate Type 1 - All chopped strand mat (CSM) construction with an internal and an external surface tissue reinforced layer.

Laminate Type 2 - Chopped strand mat (CSM) and woven roving (WR) construction with an internal and an external surface tissue reinforced layer.

Laminate Type 3 - Chopped strand mat (CSM) and multi-filament roving construction with an internal and an external surface tissue reinforced layer.

3.4.5.18 UKOOA- UK Offshore Operator Association

When creating a piping system using Fiberglass Reinforced Plastic Pipe the user needs to select the **Modeling Defaults** dialog under the **Setup** command under the **Main Menu**. Once the user has selected the Modeling Defaults screen, there are three changes that need to be implemented:

4) Change the Piping Code to read "BS7159- Fiberglass Reinforced Plastic Pipe";

5) Select the "Includes Translational Pressure deformations" box;

6) Select the "Includes Rotational Pressure Deformation" box. The later two action items tell TRIFLEX that the Bourdon Pressure Effect will be considered in the analysis.

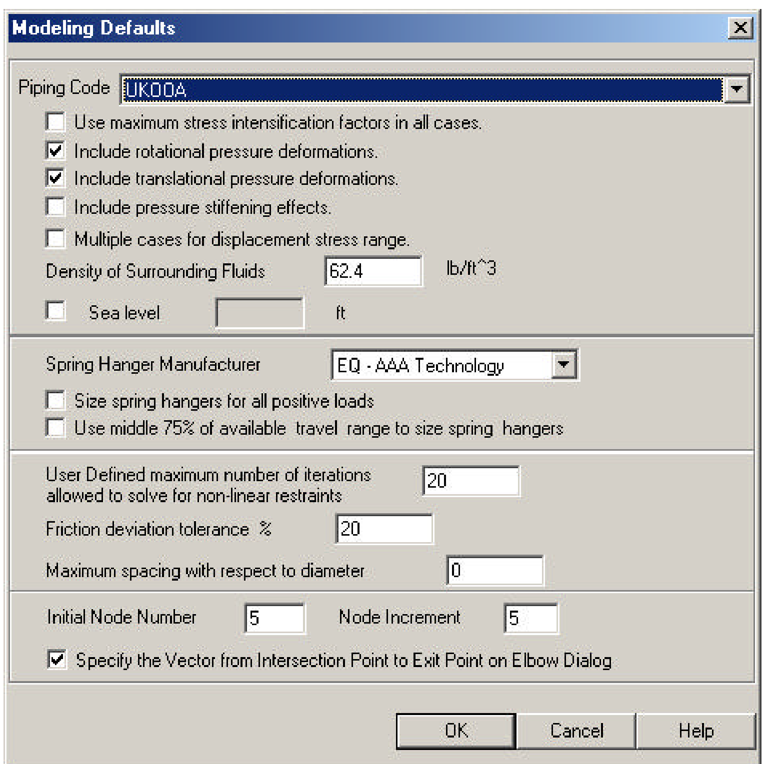


Figure 3.4.5.18.1 – Modeling Default, FRP

Note: Bourdon Pressure Effect-

The Bourdon pressure effect causes: straight pipes to displace along the x axis (elongate) and bends to elongate along the line that connects the bend near and far nodes. The Bourdon effect is always considered when plastic pipe is used. The impact of the Bourdon effect can be appreciable in long pipe runs or high pressures or large diameter bends (especially next to sensitive equipment). The two Bourdon options that are available are:

Option #1 Include rotational pressure deformations

If the elbows or bends are fabricated using hot or cold bending this will cause a slight ovalization of the bend cross section. This will cause the bend to straighten out when pressurized. Fixed end moments are associated with the opening that do not exist when the original shape of the bend cross section is circular.

Option #2 Include translational pressure deformation

If bend or elbow has a circular cross section such as a system that has forged or welded fitting on the bend or elbow.

Bourdon Effect and TRIFLEX

TRIFLEX allows the user to control two effects of internal pressure. These are the translational deformation due to pressure, and INDEPENDENTLY, the rotational deformation due to pressure. The latter comes to play within the context of elbows. So, the user may control the translational or the rotational effects through the defaults screen.

With a bend, both in-plane directions participate because there is NO unique preferred direction. Actually, there are two, such as entry to the bend, and exit from it. Usually, programs take the entry and an in-plane normal.

The next set is to build the piping model. As always TRIFLEX starts each model with an Anchor. However, under the **Piping Properties** of the Anchor dialog screen, **Reinforced Fiberglass** pipe material needs to be checked to do the BS-7159 analysis.

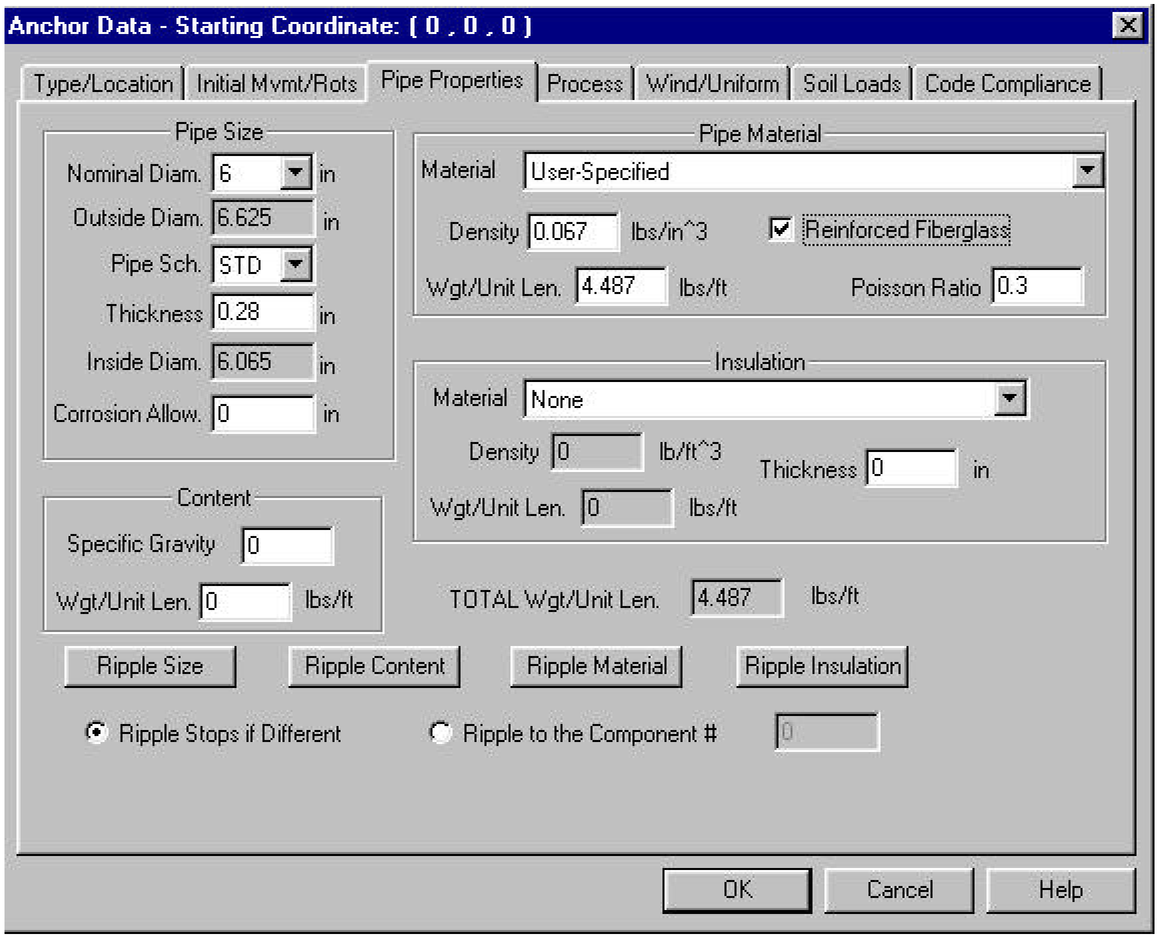


Figure 3.4.5.18.2 - Anchor Component, Pipe Properties Tab, FRP

Recent research has been conducted that shows that physical properties calculated by using the equations given in BS 7159 are likely to yield properties substantially different in magnitude from those you will obtain from the manufacturer of the FRP/GRP pipe you may be using. Therefore, it is highly recommended that each user obtain these properties from the appropriate FRP/GRP pipe manufacturer and use only these properties in the analysis of FRP/GRP piping systems.

Pipe Density (lbs/in^3, N/mm^3 10^4, g/cm^3, kg/m^3)

The user is to enter the density of the FRP/GRP material as obtained from the manufacturer.

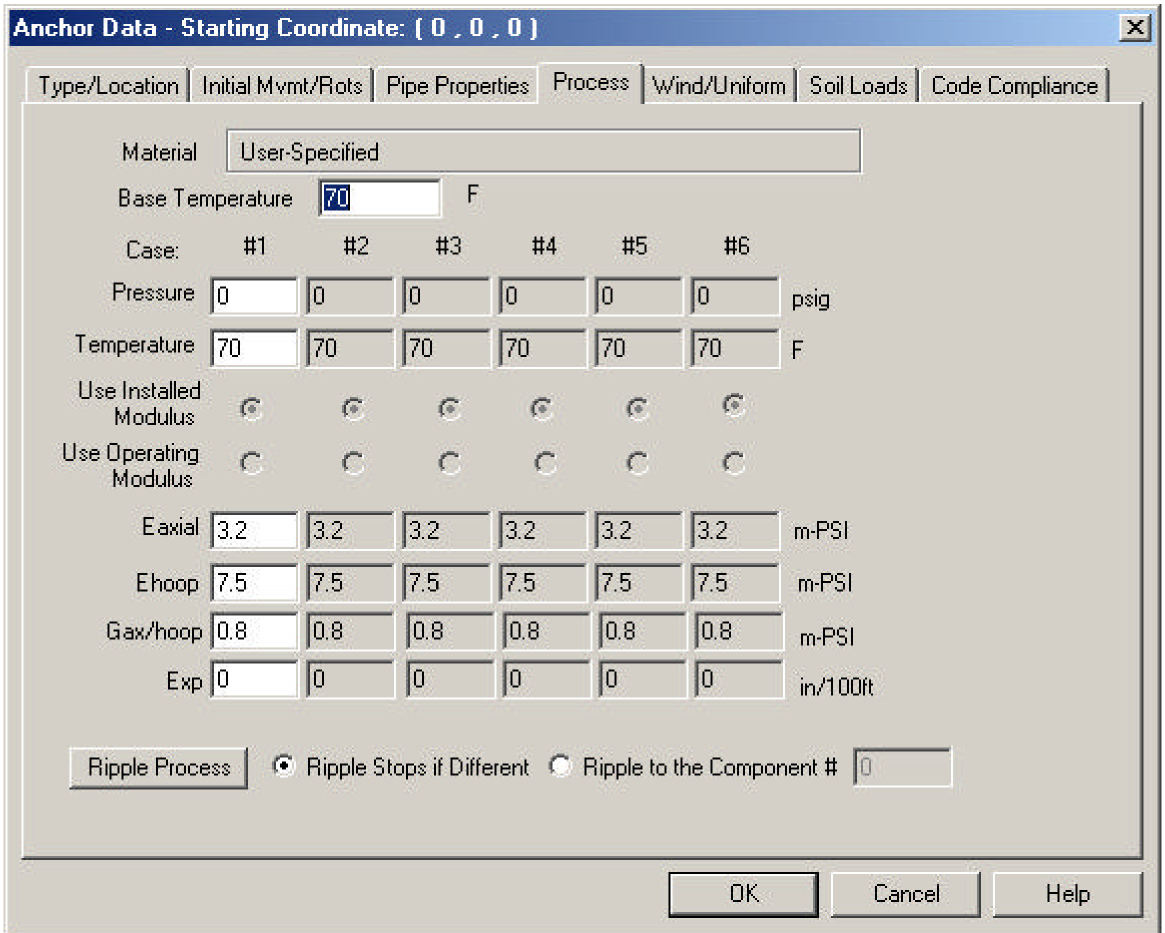


Figure 3.4.5.18.3 - Anchor Component, Process Tab, FRP

Pressure (psig, k-N/m^2, kg/cm^2, bars)

Modulus of Elasticity

Eaxial (m-PSI, k-N/m^2 x 10^{-6}, m-kg/cm^2, m-N/mm^2)

The user is to enter the modulus of elasticity in the axial direction of the Fiber Reinforced Plastic Pipe being modeled.

Ehoop (m-PSI, k-N/m^2 x 10^{-6}, m-kg/cm^2, m-N/mm^2)

The user is to enter the modulus of elasticity in the circumferential (hoop) direction of the Fiber Reinforced Plastic Pipe being modeled.

Gax/hoop (m-PSI, k-N/m^2 x 10^{-6}, m-kg/cm^2, m-N/mm^2)

The user is to enter the modulus of elastic shear between the radial and the hoop directions of the Fiber Reinforced Plastic Pipe being modeled, as it pertains to torsion.

Coefficient of Expansion

Exp (in/100 ft, mm/m, cm/100 m, mm/m)

The coefficient of expansion is to reflect the amount of growth per unit length of pipe. This value is available from the FRP/GRP pipe vendor's catalog. While modeling a fiber reinforced plastic piping system you may only specify T1 in the load case combinations field on the case data screen.

The following guidelines may assist you in properly analyzing your piping systems:

1. The design temperature change for non-insulated pipe systems containing liquids is generally recommended to be eighty-five (85) percent of the difference between the ambient temperature and the process temperature.

2. The design temperature change for non insulated pipe systems containing gases is generally recommended to be eighty (80) percent of the difference between the ambient temperature and the process temperature.

3. The design temperature change for insulated pipe systems containing liquids or gases is generally recommended to be one-hundred (100) percent of the difference between the ambient temperature and the process temperature.

4. For piping systems operating at temperatures above the ambient temperature, your base temperature should be taken to be the lowest encountered ambient temperature.

5. For piping systems operating at temperatures below the ambient temperature, your base temperature should be taken to be the highest encountered ambient temperature.

6. The coefficient of expansion for unlined FRP/GRP piping varies between 1.7 and 2.5 times that of carbon steel depending on the type of reinforcement in

the pipe wall. Obtain the proper coefficient of expansion from the manufacturer of the FRP/GRP pipe you are using.

7. The coefficient of expansion for lined FRP/GRP piping systems operating at a relatively low temperature (40 degrees C) is found to be significantly higher than that for unlined FRP/GRP pipe. As the operating temperature increases to 60 degrees C, the effect of the PVC lining decreases. At 60 degrees C and above, the influence of the PVC lining can be ignored.

Poisson's ratio

This ratio is defined by the formula:

$$Axial\ strain = \frac{axial\ stress}{Eaxial} - Pois\ ratio \cdot \frac{hoop\ stress}{Ehoop}$$

For FRP/GRP pipe, the flexibility and stress intensification factors for smooth and mitered bends are calculated by TRIFLEX based upon the guidelines provided in BS 7159 as follows:

For FRP/GRP pipe, the flexibility and stress intensification factors for smooth and mitered bends are calculated by TRIFLEX based upon the guidelines provided in BS 7159 as follows:

For FRP/GRP pipe, the flexibility and stress intensification factors for molded and fabricated tees are calculated by TRIFLEX based upon the guidelines provided in BS 7159 as follows:

A piping system may consist of both fiberglass reinforced plastic pipes as well as metal (isotropic) pipes. In such cases, take care to change properties at the point where the transition occurs. The applicable stresses for the steel pipes will be found in the stress report following the forces and moments reports and the applicable stresses for the fiberglass reinforced plastic pipes will be found in the BS 7159 Code Compliance report following the standard output.

Remember that it is highly recommended that you contact the manufacturer of the fiberglass reinforced plastic pipes that you are using in your piping system for the exact values for the properties you must use to obtain an accurate piping analysis!

For a piping system code compliance analysis to be processed, the User must enter the necessary pipe material data. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component entered. Upon clicking on the tab, a Code Compliance dialog will be presented to the User.

The data group where all of the required data is to be entered is entitled "DnV Rules for Submarine Pipeline Systems Compliance Data, 1981 Edition". The fields in which data can be entered in this data group are defined below:

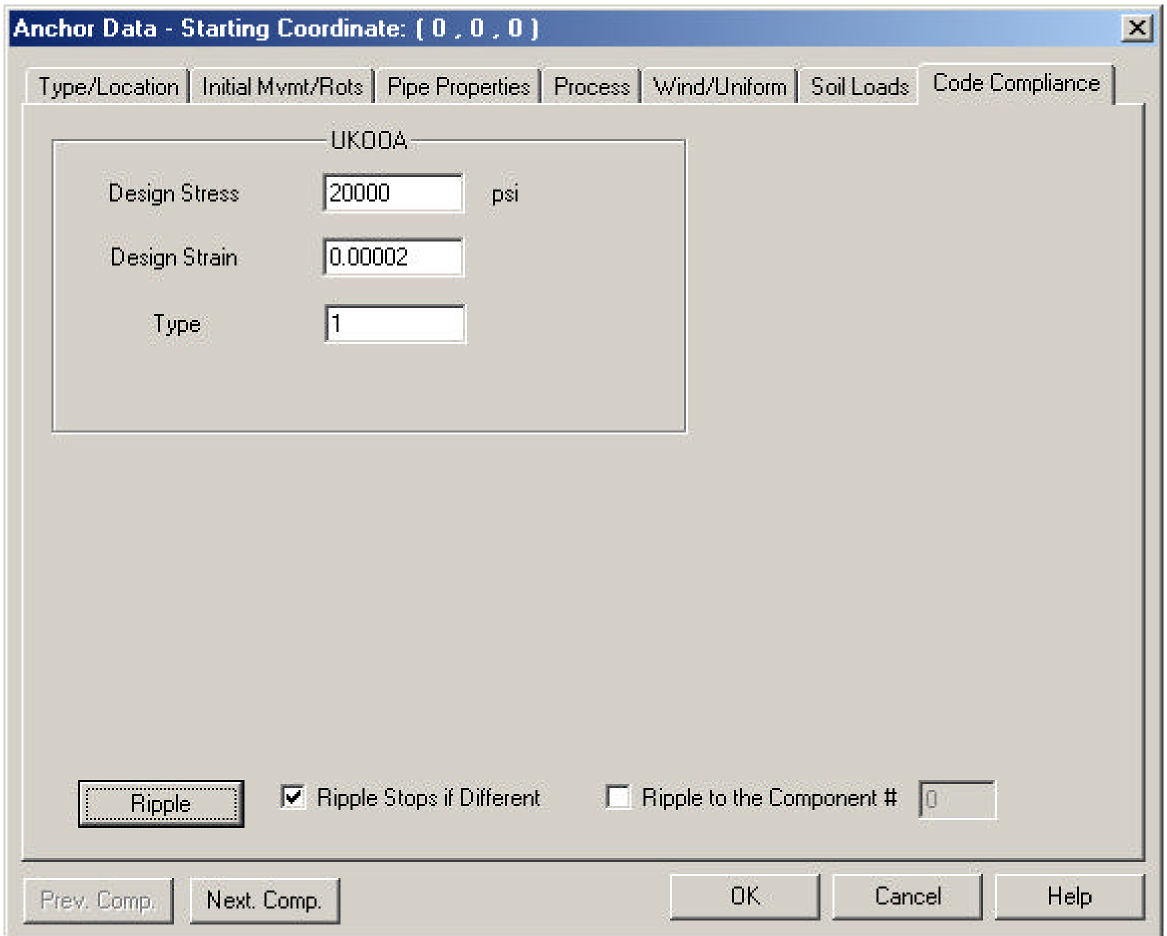


Figure 3.4.5.18.4 - Anchor Component, Code Compliance Tab, FRP

Design Stress (psi, k-N/m^2, kg/cm^2, N/mm^2)

The design stress to be entered by the user is the numeric value of the Maximum Combined Stress as obtained from the FRP/GRP pipe manufacturer.

Design Strain (Unit less)

The design strain (,$_N$) to be entered by the user is the numeric value of the maximum allowed strain as obtained from the FRP/GRP pipe manufacturer. The sum of the circumferential strain induced by pressure and the circumferential tensile strain resulting from the longitudinal compressive stress induced by temperature change shall not exceed the design strain.

Laminate Type (1, 2 or 3)

For specific details concerning the laminate types, please consult the BS 7159 Code for the Design and Construction of Glass Reinforced Plastics Piping Systems for Individual Plants or Sites. Section four of BS 7159 describes the three types of laminates and section seven of BS 7159 describes the flexibility factors and stress intensification factors for bends and branch connections for each laminate type.

Laminate Type 1 - All chopped strand mat (CSM) construction with an internal and an external surface tissue reinforced layer.

Laminate Type 2 - Chopped strand mat (CSM) and woven roving (WR) construction with an internal and an external surface tissue reinforced layer.

Laminate Type 3 - Chopped strand mat (CSM) and multi-filament roving construction with an internal and an external surface tissue reinforced layer.

3.4.5.19 BS8010 Code Compliance

For a piping system code compliance analysis to be processed, the User must enter the necessary data on the Code Compliance dialog. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component modeled in the piping system. Upon clicking on the tab, the BS8010 Piping Code Compliance dialog will be presented to the User.

The data group in which the required data is to be entered is entitled "BS8010 Code for Pipelines Compliance Data". The fields in which data can be entered in this dialog are defined below:

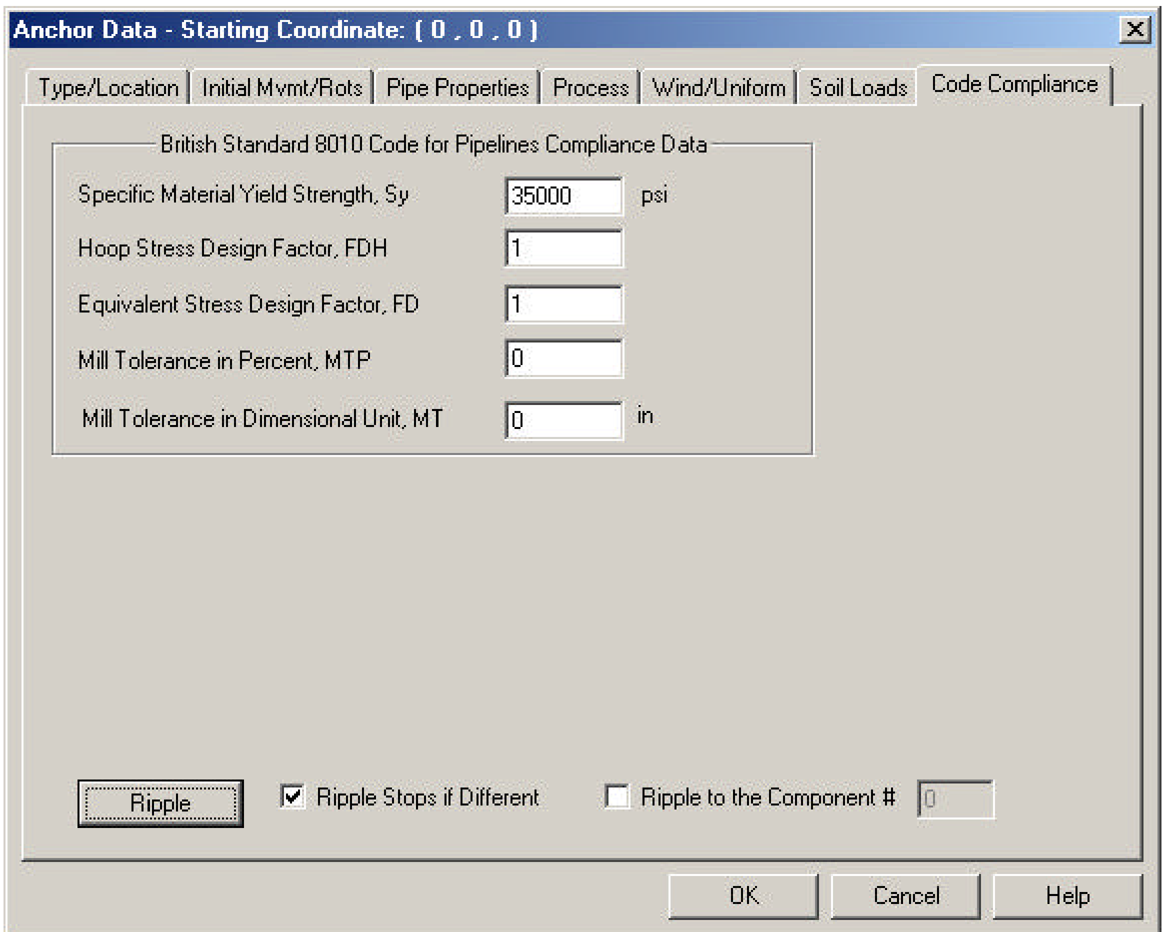


Figure 3.4.5.19 - Anchor Component, Code Compliance Tab, BS 8010

SMYS – In this field, the User must enter a value for the Specified Minimum Yield Strength of the pipe to be covered by the Code Compliance. If the User does not enter a value in this field, TRIFLEX will assume the default value of 35,000 psi for the SMYS.

Hoop Stress Design Factor, FDH – In this field, the User should enter the Hoop Stress Design Factor (FDH) as described in the BS8010 Code for Pipelines. If the User does not enter a value in this field, TRIFLEX will assume the default value of 1.0.

Equivalent Stress Design Factor, FD – In this field, the User should enter the Equivalent Stress Design Factor (FD) as described in the BS8010 Code for Pipelines. If the User does not enter a value in this field, TRIFLEX will assume the default value of 1.0.

Mill Tolerance in Percent, MTP – The User may enter a value for the mill tolerance in percent in this field. The default is 12 1/2 percent. This field will be grayed out if the User enters mill tolerance in a fixed amount in the following field.

Mill Tolerance in Dimensional Units, MT – The User may enter a value for the mill tolerance in inches / mm. There is no assumed default value in this field. This field will be grayed out if the User enters mill tolerance as a percentage of the entered wall thickness in the previous field.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.20 NPD Guidelines for Submarine Pipelines and Risers

For a piping system code compliance analysis to be processed, the User must enter the necessary data on the Code Compliance dialog. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component modeled in the piping system. Upon clicking on the tab, the **Submarine Pipelines and Risers Norvegian Petroleum Directorate** for Piping Code Compliance dialog will be presented to the User.

The data group in which the required data is to be entered is entitled "**Norvegian Petroleum Directorate for Submarine Pipelines and Risers**" Code for Pipelines Compliance Data". The fields in which data can be entered in this dialog are defined below:

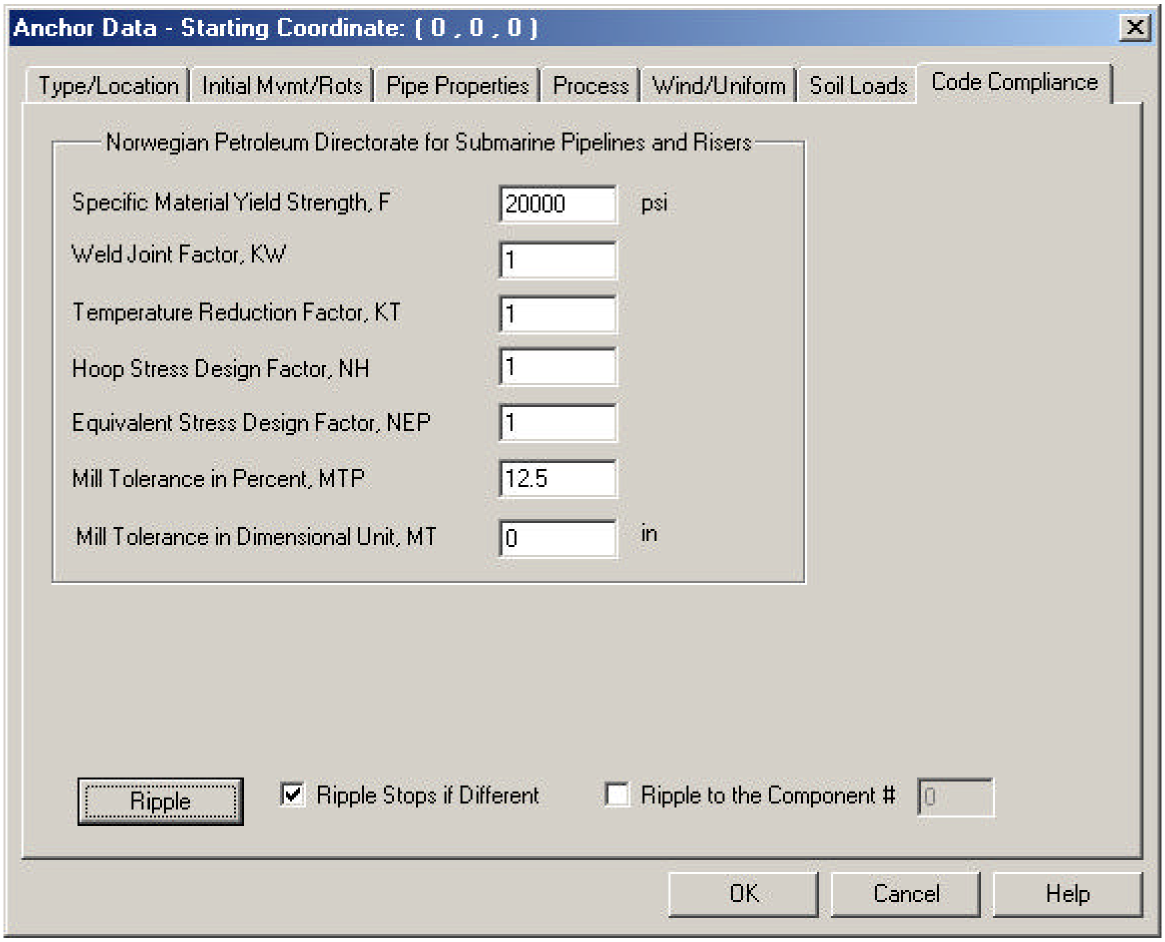


Figure 3.4.5.20 - Anchor Component, Code Compliance Tab, NPD

Specific Material Yield Strength, F – In this field, the User must enter a value for the Specific Material Yield Strength of the pipe to be covered by the Code Compliance. A default value of 20,000 psi will be assumed if the User does not enter a value.

Weld Joint Factor, KW – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. A default value of 1.0 will be assumed if the User does not enter a value.

Temperature Reduction Factor, KT – In these fields, the User should enter the temperature reduction factor(s) applicable for this piping component. A default value of 1.0 will be assumed if the User does not enter a value.

Hoop Stress Design Factor, NH – The User should enter the desired hoop stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Equivalent Stress Design Factor, NEP – The User should enter the desired equivalent stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

Mill Tolerance in Percent, MTP – The User may enter a value for the mill tolerance in percent in this field. The default value is zero percent. A numerical value may only be entered in this field or in the following mill tolerance field, but not both.

Mill Tolerance in Dimensional Unit, MT – The User may enter a value for the mill tolerance in inches, mm or cm in this field. The default value is zero. A numerical value may only be entered in this field or in the previous mill tolerance field, but not both.

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.21 Statoil Design, Specifications Offshore Pipeline Systems

For a piping system code compliance analysis to be processed, the User must enter the necessary data on the Code Compliance dialog. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component modeled in the piping system. Upon clicking on the tab, the "Design, Specification Offshore Installations-Offshore Pipeline Systems -F-SD-101", 1987 by Statoil for Piping Code Compliance dialog will be presented to the User.

The data group in which the required data is to be entered is entitled "Statoil Specification Offshore Pipeline Systems" Code for Pipelines Compliance Data". The fields in which data can be entered in this dialog are defined below:

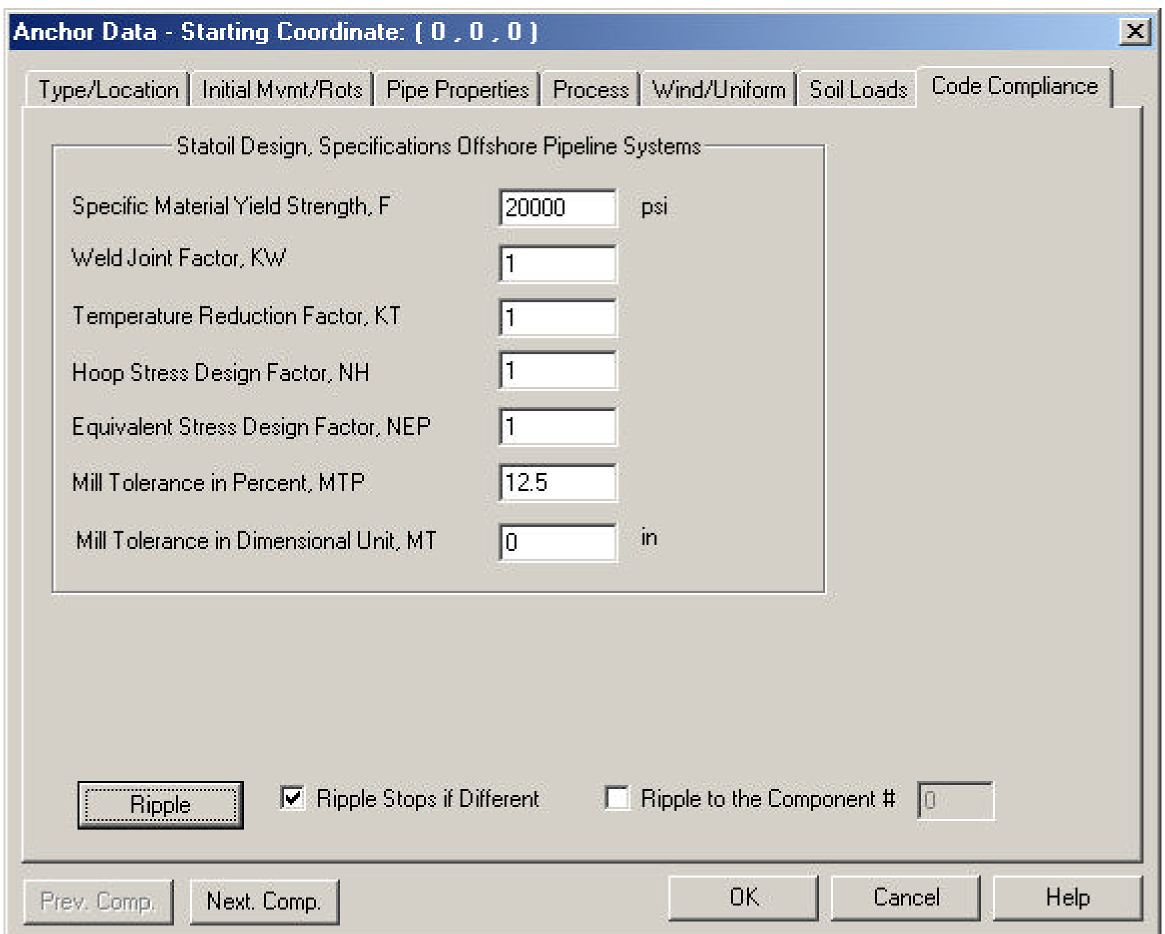


Figure 3.4.5.21 - Anchor Component, Code Compliance Tab, STOL

<u>Specific Material Yield Strength, F</u> – In this field, the User must enter a value for the Specific Material Yield Strength of the pipe to be covered by the Code Compliance. A default value of 20,000 psi will be assumed if the User does not enter a value.

<u>Weld Joint Factor, KW</u> – In this field, the User should enter the weld joint factor for the welding process used in the manufacture of the pipe. A default value of 1.0 will be assumed if the User does not enter a value.

<u>Temperature Reduction Factor, KT</u> – In these fields, the User should enter the temperature reduction factor(s) applicable for this piping component. A default value of 1.0 will be assumed if the User does not enter a value.

<u>Hoop Stress Design Factor, NH</u> – The User should enter the desired hoop stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

<u>Equivalent Stress Design Factor, NEP</u> – The User should enter the desired equivalent stress design factor as defined in the piping code. A default value of 1.0 will be assumed if the User does not enter a numerical value in this field.

<u>Mill Tolerance in Percent, MTP</u> – The User may enter a value for the mill tolerance in percent in this field. The default value is zero percent. A numerical value may only be entered in this field or in the following mill tolerance field, but not both.

<u>Mill Tolerance in Dimensional Unit, MT</u> – The User may enter a value for the mill tolerance in inches, mm or cm in this field. The default value is zero. A numerical value may only be entered in this field or in the previous mill tolerance field, but not both.

<u>Ripple</u> – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.5.22 European Standard prEN 13480-3

For a piping system code compliance analysis to be processed, the User must enter the necessary data on the Code Compliance dialog. To enter the required data, the User must click on the Code Compliance tab at the top of the screen on the first component modeled in the piping system. Upon clicking on the tab, the "European Code prEN 13480-3", the Piping Code Compliance dialog will be presented to the User.

The data group in which the required data is to be entered is entitled “Euro Piping Code Compliance Data”. The fields in which data can be entered in this dialog are defined below:

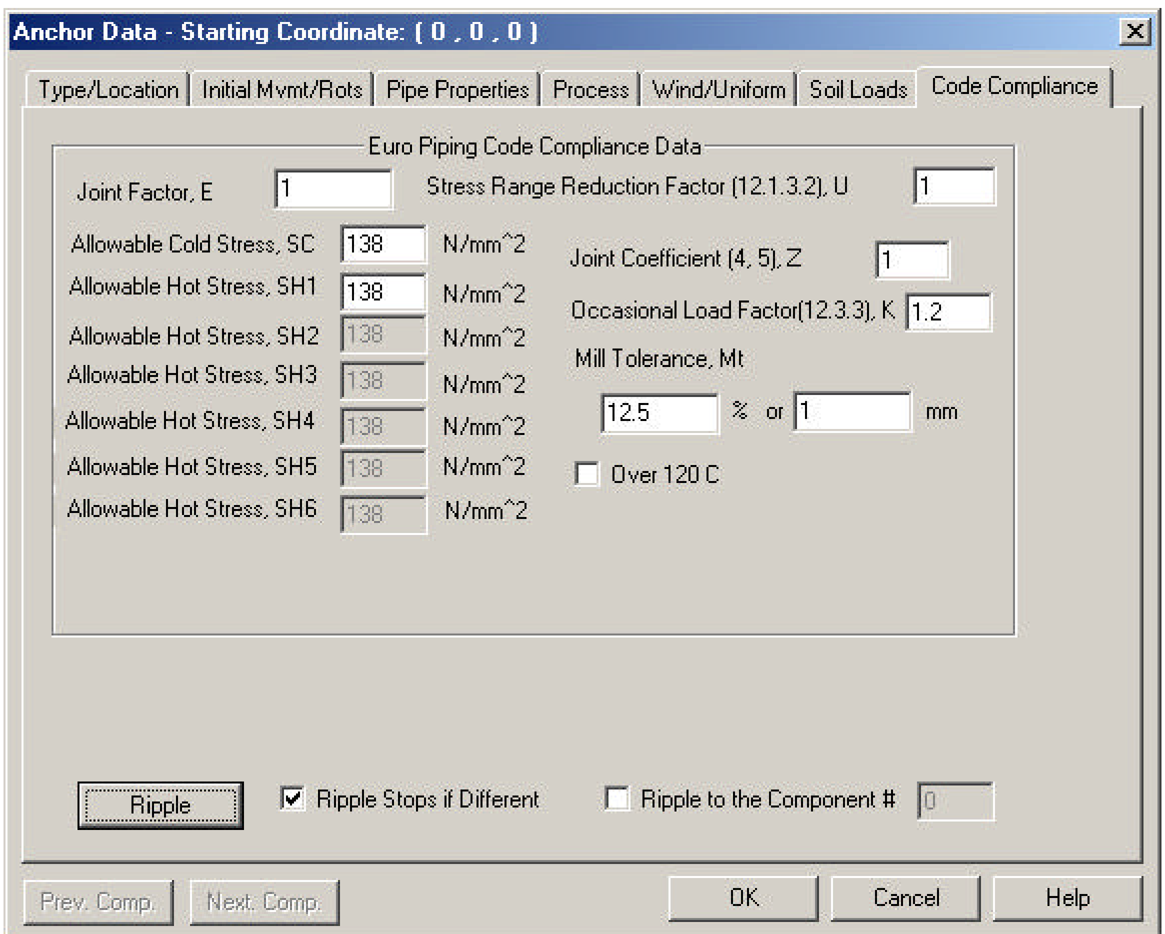


Figure 3.4.5.22 - Anchor Component, Code Compliance Tab, EUROCODE

Minimum Cold Stress (f_c)

The basic material allowable stress value at room temperature.

Maximum Hot Stress (f_h)

The material allowable stress at temperature consistent with the loading under consideration.

Stress Range Reduction Factor U

The stress range reduction factor for cyclic conditions for total number N of full temperature cycles over total number of years during which system is expected to be in service from Table 12.1.3-1

Occasional Load Factor k

Factor specified by the analyst, based upon the duration of the occasional loads (12.3-3)

Joint Coefficient Z

The joint coefficient z shall be used in the calculation of the thickness of components which include one of several butt welds, other than circumferential (4.5)

Mill Tolerance

Manufacturer mill tolerance in percent or millimeters.

Temp Over 120^o C

If the design temperature is above 120^o C the User must check this check box

Ripple – When the User has modified one or more data entries on the Code Compliance dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.6 Discussion of Anchor Component, Soil Load Tab

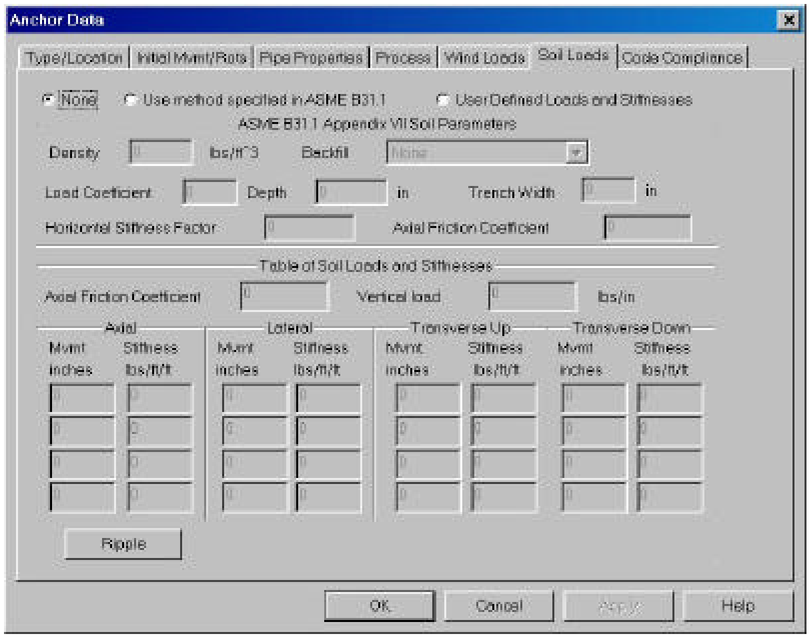


Figure 3.4.6 – Anchor Component, Soil Load Tab

For every piping system where soil loading is to be considered, the User must enter the soil related data. To enter the required data, the User must click on the Soil Loads tab at the top of the screen on the first component on which the soil loads are to be applied. Upon clicking on the tab, a Soil Loads dialog will be presented to the User.

The data is organized in related data groups on this dialog. On the first line of the Soil Loads dialog, TRIFLEX provides three radio buttons for the User to select from. The default is "None". If the User wishes to consider Soil Loads, the User can select between the middle radio button and the right radio button. Each radio button and resultant option is explained in the following text.

If the User selects the left most radio button ("None"), all fields on the dialog will be grayed out and inaccessible by the User.

If the User selects the middle radio button (Use method specified in ASME B31.1), all the fields in the ASME B31.1 Appendix VII Soil Parameters data group will be made active and available for data entry. Immediately below the three radio buttons, a data group entitled "ASME B31.1 Appendix VII Soil Parameters" will be available for User data entry. The data group entitled Table of Soil Loads and Stiffness' will be grayed out and inaccessible for data entry.

If the User selects the right most radio button (User Defined Loads and Stiffness'), all the fields in the Table of Soil Loads and Stiffness' data group will be made active and available for data entry. The data group entitled "ASME B31.1 Appendix VII Soil Parameters" will be grayed out and inaccessible for data entry.

The fields in which data can be entered in these data groups are defined below:

When the User selects the middle radio button (Use method specified in ASME B31.1), data can be entered by the User in the following fields in the ASME B31.1 Appendix VII Soil Parameters data group:

Density – In this field, the User may enter a numerical value for the density of the soil surrounding the pipe. This value is used to calculate vertical load and stiffness.

Backfill - In this field, TRIFLEX will display the default backfill – "NONE". The User must select the desired backfill from the drop down combo list in this field. The backfill type must be entered if the user wants TRIFLEX to use the data found in Table VII-3.2.3 in Marston's formula for pipes buried below three times the pipe diameter. The available selections are:

Damp Top Soil

Saturated Top Soil

Damp Yellow Clay

Saturated Yellow Clay

Dry Sand

Wet Sand

Depth – In this field, the User must enter the depth – the distance from grade to the centerline of the pipe. This value is needed for the calculation of the vertical load and the soil stiffness. The User may enter the **load coefficient**, undoing the effect of depth on the load calculation.

Trench Width - In this field, the User must enter the trench width – the width of the trench that was dug in which the pipe is buried. This value is needed if the User wishes TRIFLEX to apply Marston's formula for pipes buried below three times the pipe diameter.

As soon as the vertical load is calculated, it appears in the Vertical Load field in the Table of Soil Loads and Stiffness' data group. The User can modify any of the values displayed in the Table of Soil Loads and Stiffness' data group.

Load Coefficient – In this field, the load coefficient calculated by TRIFLEX according to table VII-3.2.3 of the ASME B31.1 Appendix VII will be displayed.

The User may override the value by entering a different numerical value in this field. The load coefficient is used for the calculation of vertical loads and applied to pipes buried more than three pipe diameters below grade.

Horizontal Stiffness Factor - In this field, the User must enter the horizontal stiffness factor. This value is needed in order to calculate the lateral stiffness. Recommended values are:

Loose soil	20
Medium soil	30
Dense or compact soil	80

Axial Friction Coefficient - In this field, the User may specify the upper limit for the axial frictional force as a fraction of the resultant normal forces acting on the pipe. The normal forces consist of the lateral and transverse components, each dependent on the respective stiffness and movement. The total weight of the soil above the pipe, the pipe, and the pipe contents are loads that would be considered in these forces acting on the pipe.

As the User enters the above listed data variables, TRIFLEX performs calculations in accordance with the procedures set forth in the ASME B31.1 Appendix VII Soil Parameters. As the data is entered and the calculations performed, TRIFLEX places the calculated values in the data fields in the Table of Soil Loads and Stiffness' data group located at the bottom of the dialog.

If the User selects the right most radio button (User Defined Loads and Stiffness'), all the fields in the Table of Soil Loads and Stiffness' data group will be made active and available for data entry. The data group entitled "ASME B31.1 Appendix VII Soil Parameters" will be grayed out and inaccessible for data entry.

Axial Friction Coefficient - In this field, the User may specify the upper limit for the axial frictional force as a fraction of the resultant normal forces acting on the pipe. The normal forces consist of the lateral and transverse components, each dependent on the respective stiffness and movement. The total weight of the soil above the pipe, the pipe, and the pipe contents are loads that would be considered in these forces acting on the pipe.

Vertical Load - In this field, the User may specify the desired vertical load. The vertical load is parallel to the pull of gravity (the direction the force of weight acts), regardless of the orientation of the pipe. The user may estimate the load (per unit length) from the weight of backfill (density, depth, width).

The remainder of the data that can be entered in the fields in the Table of Soil Loads and Stiffness' data group is divided into four additional sub-groups as described below. Each sub-group contains the stiff nesses for the specified range of movements. The soil resistance to pipe movement is specified by the User as a system of variable spring stiffnesses. As the pipe moves against the soil, the soil will offer a resistance to that movement. Since soil is non-linear in nature, the resistance may vary as the movement increases. TRIFLEX allows up to four pairs of movements and stiffnesses for each direction considered in the analysis. This enables the User to more accurately define the soil properties.

The significance of movement and stiffness pairs is best explained by considering the following example. Consider the following table (regardless of units).

Movement	Stiffness
1	1000
3	500
4	-200

All forces in the following are per unit length of the piping.

For movement 1 resistance is (1000)(1 unit of movement), up to 1000 force

For 1 < movement 3 resistance is (1000 + (500)(3 units - 1) up to 2000 force

For 3 < movement 4 resistance is (2000 - (200)(4 units - 3) up to 1800 force

For movement > 4 resistances is 1800 force (implied zero stiffness).

NOTE: The negative stiffness indicates soil loses strength.

Axial Direction - The axial direction is determined by the pipe direction, or tangent to the bend. When values are specified by the User in the movement and soil stiffness fields as well as the Axial Friction coefficient field, both will be used in the analysis.

Lateral Direction **-** The lateral direction for a run of pipe is perpendicular to the axis of the pipe and horizontal. For an elbow (bend), this direction is in the radial direction. The bend beginning and bend end points will be treated as bend points and, therefore, will use the radial direction.

Transverse Up **-** The transverse up direction for a run of pipe is the upward direction and perpendicular to the axis of the pipe. For an elbow (bend), this direction is perpendicular to the axial and radial direction.

Transverse Down **-** The transverse down direction for a run of pipe is the downward direction and perpendicular to the axis of the pipe. For an elbow (bend), this direction is perpendicular to the axial and radial direction.

The stiffness for the **Transverse Down** direction may be altered to reflect a well-packed condition. A value such as 1,000,000 pounds per foot per foot length of pipe may be used to indicate a well-packed condition. The coding of this number may be accomplished by entering the numerical value in the following fashion, **1E6**.

Ripple **–** When the User has modified one or more data entries on the Wind Loads dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.4.7 Discussion of Anchor Component, Wind Loads

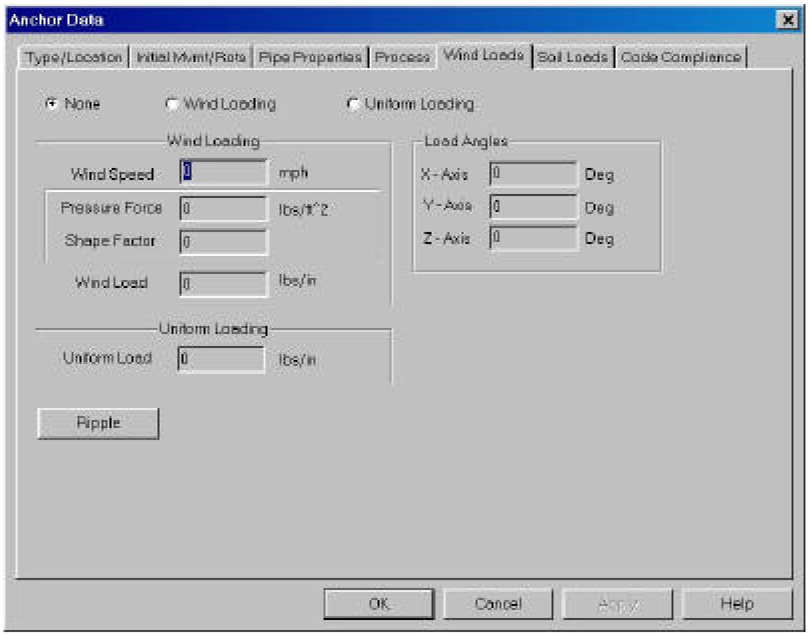


Figure 3.4.7 – Anchor Component, Wind Load Tab

For every piping system where wind or uniform loads are to be considered, the User must enter the wind load data or the uniform load data. To enter the required data, the User must click on the Wind Loads tab at the top of the screen on the first component on which the wind or uniform loads are to be applied. Upon clicking on the tab, a Wind Load dialog will be presented to the User.

The data is organized in related data groups on this dialog. On the first line of the Wind Load dialog, TRIFLEX provides three radio buttons for the User to select from. The default is “None”. If the User wishes to enter Wind Loads, the User should click on the Wind Loading radio button. If the User wishes to enter Uniform Loads, the User should click on the Uniform Loading radio button.

If the “None” radio button is selected, all fields on the dialog will be grayed out and inaccessible by the User.

If the User selects Wind Loading, the Wind Loading data group will be made active as will the Load Angles data group. Immediately below the three radio buttons and in the left column of the Wind Loads dialog, a data group entitled “Wind Loading” will be available for User data entry. The fields in which data can be entered in this data group are defined below:

Wind Speed – In this field, the User may enter a numerical value for the Wind Speed. When a value is entered in this field, TRIFLEX will calculate the Wind Load based upon the projected pipe shape. In the event that the pipe is insulated, the projected pipe shape will include the insulation.

From ANSI A58.1 (1982) Para. - 6.5 Velocity Pressure

Velocity Pressure (lbs/ft5) @ height z = 0.00256 x KHz (IV)5

Where:

KHz = Velocity pressure exposure (Table 6)

I = Importance Factor (Table 5)

V = Wind speed (MPH) (Fig 1 or Table 7)

Wind Load: (lbs per linear inch of pipe) calculated by TRIFLEX:

$$WindLoad = \frac{0.00256 \cdot VI^2 \cdot (OD + 2(insthk)) \cdot 0.6}{144}$$

Where:

V1 = Wind speed supplied by user in the Wspeed field to accommodate extraneous factors.

$$\text{Suggested Wspeed} = \text{V x I x } \sqrt{\text{Hz}}$$

When the User enters Wind Speed, TRIFLEX will automatically calculate additional loads and stresses that result from the wind loads. The true effect of the wind loads will be projected onto the piping system.

Wind Pressure – In this field, the User may enter a numerical value for the Wind Pressure. When a value is entered in this field, TRIFLEX will calculate the resulting Wind Load based upon the projected pipe area and the shape factor. In the event that the pipe is insulated, the projected pipe shape will include the insulation.

Shape Factor – The factor for a flat surface is 1.0. The factor for a cylinder is typically considered to be 0.6. See the latest version of the ANSI A58.1 Standard for further data.

Wind Load – In this field, the User may enter the actual numerical value for the Wind Load that is to be applied to each unit length of the pipe. When a value is entered in this field, TRIFLEX will simply apply the entered load. No calculations for projected area will be performed, no shape factor will be considered and entering pipe insulation will have no effect. Wind load must be entered as a positive number.

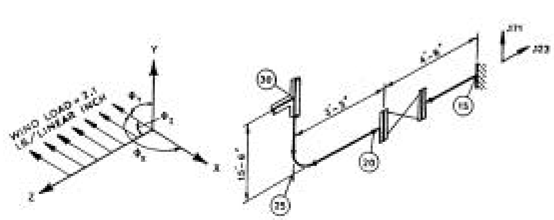
If the User selects Uniform Loading, the Uniform Loading data group will be made active as will the Load Angles data group. Immediately below the data group entitled "Wind Loading", a data group entitled "Uniform Loading" will be available for User data entry. The field in which data can be entered in this data group is defined below:

Uniform Load – In this field, the User may enter the actual numerical value for the Uniform Load that is to be applied to each unit length of the pipe. When a value is entered in this field, TRIFLEX will simply apply the entered load. No calculations for projected area will be performed, no shape factor will be considered and entering pipe insulation will have no effect. Uniform load must be entered as a positive number.

When the User selects Wind Loading or Uniform Loading, the Load Angles data group will be made active. Immediately to the right of the data group entitled "Wind Loading", a data group entitled "Load Angles" will be available for User data entry. The fields in which data can be entered in this data group are defined below:

Wind or Uniform Loading Angles – In this data group, the User may enter the angles between the wind or uniform load vector and the actual numerical value for the global X, Y, and Z-axes. These angles must be between 0 and 180 degrees.

L-Angles :X Y Z

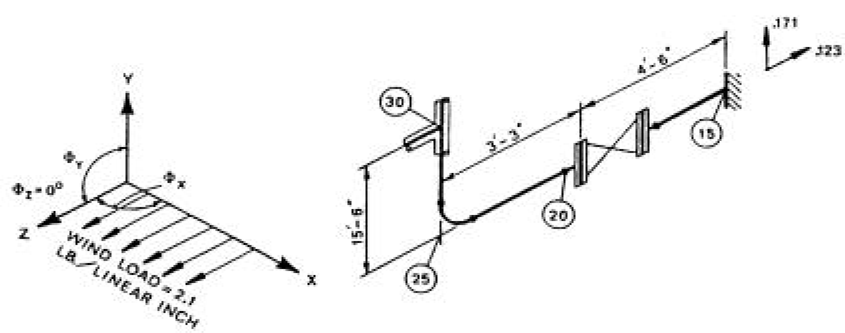


The load angles for this figure are:

L-angles-X=180.00

L-angles-Y=90.00

L-angles-Z=90.00



The load angles for this figure are:

L-angles-X=90.00

L-angles-Y=90.00

L-angles-Z=0.00

Ripple– When the User has modified one or more data entries on the Wind Loads dialog, the User can instruct TRIFLEX to modify all subsequent occurrences of these data entries that are in an unbroken series from the original revision forward by pressing the Ripple button.

3.5 Coding Piping Data

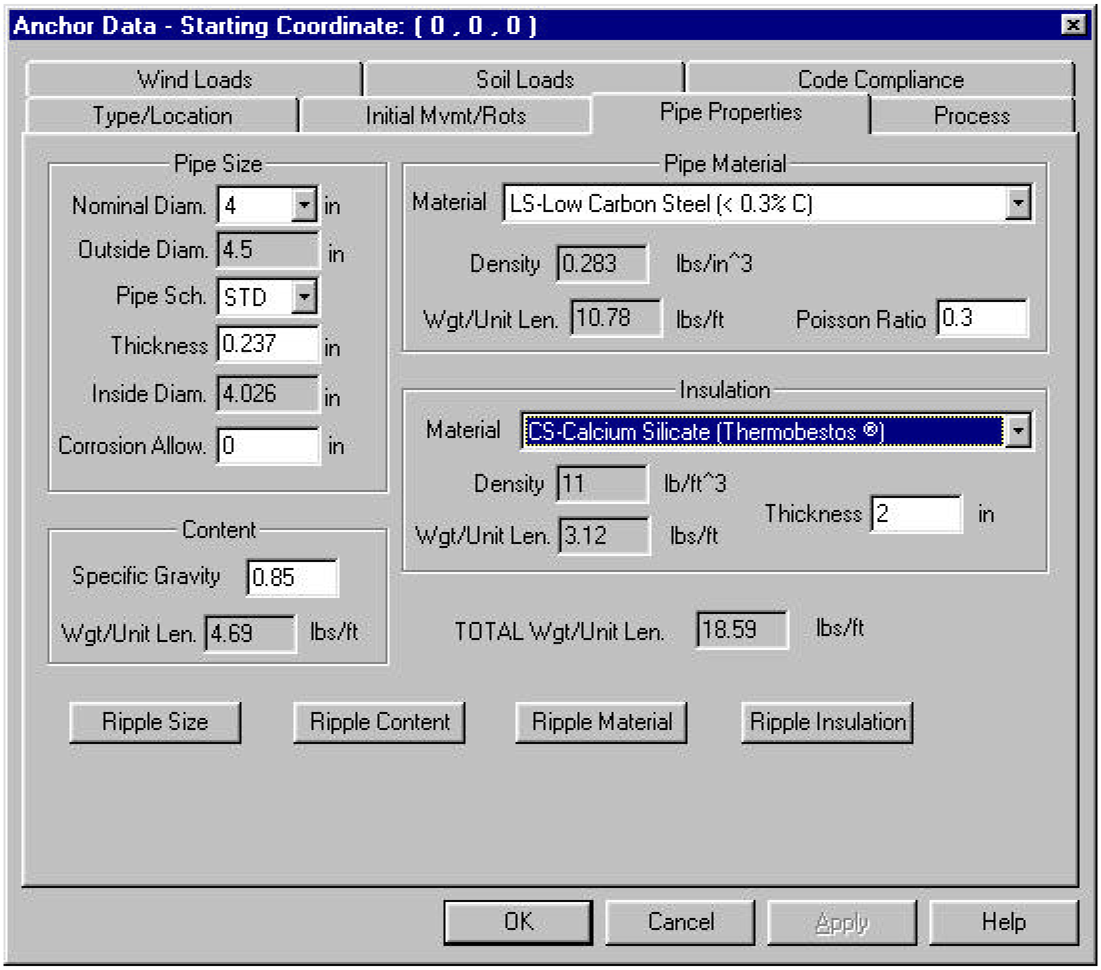


Figure 3.5 – Anchor Component, Pipe Properties Tab

3.5.1 Coding Piping Data, Piping Data

To enter a Pipe component, the User must click on the Pipe Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Pipe on the resulting pull down menu. Upon either of these sequences of actions, a Pipe dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Pipe dialog, a data group entitled "Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the “Element” data, the User will find a data group entitled “Dimension from “From Node” to “To Node”. The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the end point) of the pipe being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z– If the User is specifying the first component after an anchor, TRIFLEX will assume an “X” dimension equal to one foot if English units are specified or .35 Meters if metric units are specified. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or more axes, the User may simply enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will calculate the Delta X, Delta Y and Delta Z dimensions automatically and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of any preceding component, and then the User should place a check in the box immediately to the left of the label “Use the Minimum Length”. TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the Pipe being modeled follows a Bend or a Valve, Flange or Joint with the data point specified at a point other than the end point.

Minimum Length – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

Number of Intermediate Nodes – If the User enters a number in this field, TRIFLEX will break the pipe into one more segment that the number of intermediate nodes specified in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the to node and the from node – TRIFLEX will break the pipe into three (3) segments.

Maximum Spacing – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe components generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

To the immediate right of the "Element" data, the User will find a data group entitled "Cold Spring". The fields in which data can be entered in this data group are defined below:

Cut Short – If the User wishes to tell TRIFLEX to consider a "cut short", then the User should place a check in the box immediately to the left of the label "Cut Short" and then enter the amount of cut short in the field entitled "Cut Length".

Cut Long – If the User wishes to tell TRIFLEX to consider a "cut long", then the User should place a check in the box immediately to the left of the label "Cut Long" and then enter the amount of cut long in the field entitled "Cut Length".

Cut Length – If the User has placed a check in the Cut Short or the Cut Long check boxes, then the value entered in this field will be the amount of the cut short or long considered by TRIFLEX.

Immediately to the right of the data group entitled "Element Data" and below the data group entitled "Cold Spring", the User will find a data group entitled "Stress Intensification Factors". The fields in which data can be entered in this data group are further defined below:

For "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the pipe component, then the User should enter the desired numerical value in this field.

For "To Node" - If the User wishes to specify a numerical stress intensification factor on the end of the pipe component, then the User should enter the desired numerical value in this field.

Immediately below the "Stress Intensification Factors" data group, the User will find additional data fields for miscellaneous data defined as follows:

Weight Off - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label "Weight Off". The default is for weight to be considered.

Buoyancy - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label "Buoyancy Calculations". The density of the fluid surrounding the pipe should also be specified on the **Setup / Modeling Defaults dialog**. The default is for the effects of buoyancy not to be considered.

Immediately below the miscellaneous data fields, the User will find a data group entitled "Pipe Size". In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

3.5.2 Coding Pipe Data, Restraint Tab

Restraints may be entered on the following components: Pipe, Bend/Elbow, Branch Connection, Valve, Flange, Reducer and Joint. A Restraint is not a component in TRIFLEX; it is an attachment to a piping component that enables an external action to be applied on the piping system. To enter a restraint on the piping system, select any of the above noted components and the Restraint tab will be displayed as one of the tabs along the top edge of the component dialogs. To enter a restraint, click on the Restraint tab at the top of the dialog and the Restraint dialog will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on the Restraint dialog. In the upper left corner of the "Restraint" dialog, a data group entitled "Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

Load Case – This field is reserved for future use and is grayed out at this time.

Node Num. – In this field, TRIFLEX displays the To Node number for the component on which the restraints are to be applied. The field is grayed out since the node number may not be altered on this dialog.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Coordinate System". In this data group, the User defines the axis system that will be used to describe the restraint action. TRIFLEX provides three different axis systems to choose from – the standard X, Y, Z axis system, the L, N, G axis system which enables a User to enter restraints along the axis of the pipe and along the axes perpendicular to the pipe and the A, B, C axis system which enables a User to enter restraints along an axis system that may be skewed in relationship to the X, Y, Z axis system as well as the L, N, G axis system. The User clicks on a radio button to select the desired axis system. The options available for the User are defined below:

X, Y, Z Coord. System – The default axis system is the X, Y, and Z-axis system. The radio button to the left of this title will be selected as the default. When the User selects the X, Y, Z coordinate system, all restraint action specified by the User will be applied along the X, Y or Z-axes. No orientation angles are required when the X, Y, Z coordinate system is selected. When the X, Y, Z coordinate system is selected by the User, TRIFLEX will display the Translational Restraint Action data group and the Rotational Restraint Action data group with the X axis, Y axis and Z axis headings and orientation angles will not be accepted as input.

L, N, G Coord. System – When the User selects the L, N, G coordinate system; all restraint action specified by the User will be applied along the L, N or G axes. No orientation angles are required to be entered by the User. TRIFLEX will automatically compute the orientation angles internally when the User selects the L, N, and G coordinate system. The axis convention for the L, N, and G coordinate system is as follows:

L is along the axis of the pipe and positive in the direction of coding.

N is normal to the pipe and most vertical.

G is perpendicular to the pipe and horizontal (guide).

When the L, N, G coordinate system is selected by the User, TRIFLEX will display the Translational Restraint Action data group and the Rotational Restraint Action data group with the L axis, N axis and G axis headings.

A, B, C Coord. System – When the User selects the A, B, C coordinate system, all restraint action specified by the User will be applied along the A, B or C axes. Orientation angles must be entered by the User when the A, B, C coordinate system is selected. The A, B, C axis system is a standard right hand rule axis system that can be oriented as desired by the User. The User simply must orient the A, B, and C axis system with respect to the X, Y Z axis system. When the A, B, C coordinate system is selected by the User, TRIFLEX will display the Translational Restraint Action data group and the Rotational Restraint Action data group with the A axis, B axis and C axis headings.

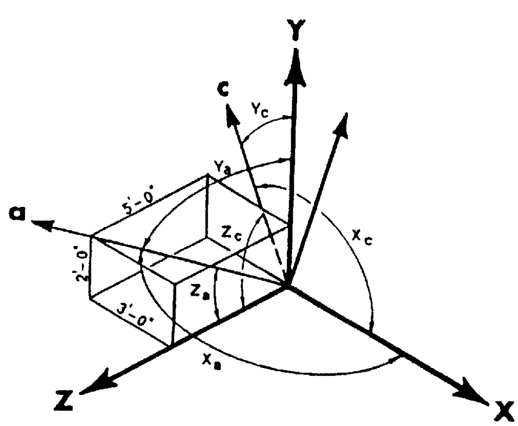
Use Directional Vectors – This option is the default when the User selects the A, B, and And C Coordinate System. When the User selects this option, the User must specify the vector direction of a skewed restraint. The length of the X, Y, and Z vectors must be coded from the point of the restraint attachment on the pipe to the point of restraint attachment on the external structure. The resultant of the X, Y, and Z vectors defines the orientation of the A axis along which the restraint action will be applied. When this option is selected, the User can only enter a restraint along or about the A axis.

The User must enter the directional vectors in decimal values in the three fields provided - the X-axis, the Y-axis and the Z-axis. Note: these vectors only provide the A axis orientation and the resultant specific length has no significance.

Use Action Angles – When the User selects the A, B, C coordinate system and wishes to enter restraints along or about more than one axis; the User should select this option. This enables the User to specify the angles between the X, Y, Z coordinate system and the A, B, and C coordinate system along and about which the User can enter restraints.

The User must define the orientation of the A axis and the C axis in order for TRIFLEX to completely orient the A, B, C coordinate system in relation to the X, Y, Z coordinate system. The User must specify the angles that the A-axis makes with the +X, +Y, and +Z-axes and the C-axis makes with the +X, +Y, and +Z-axes. The angles specified will be between 0 and 180 degrees.

Figure 3.5.2-2 XYZ and ABC Coordinate Systems



Immediately below the data group entitled "Coordinate System", the User will find an individual data item defined as follows:

Friction Coefficient – The default condition for this field is grayed out and inaccessible for data entry. In order for the User to be able to enter a coefficient of frictional resistance to movement, the User must have done one of the following:

Selected the X, Y, Z Coordinate System and have selected a +Y or a +/- Y or a – Y restraint

Selected the L, N, G Coordinate System and have selected a +N or a +/- N or a – N restraint

Selected the A, B, C Coordinate System and have selected a +B or a +/- B or a –B restraint

The coefficient of frictional resistance must be determined by the User and entered in this field. If this field is left blank, the frictional resistance will be considered to be zero. The actual frictional restraining force is iteratively calculated by TRIFLEX. Section 5 in the TRIFLEX User Manual contains an in-depth discussion of modeling with frictional restraints.

Immediately to the right of the data group entitled "Element", the User will find a data group entitled "Translational Restraint Action". When the X, Y, Z coordinate system is selected by the User, TRIFLEX will display the Translational Restraint Action data group with the X axis, Y axis and Z axis headings. When the L, N, G coordinate system is selected by the User, TRIFLEX will display the Translational Restraint Action data group with the L axis, N axis and G axis headings. When the A, B, C coordinate system is selected by the User, TRIFLEX will display the Translational Restraint Action data group with the A axis, B axis and C axis headings. The fields in which data can be entered in this data group are defined below:

X, Y, Z coordinate system

Note: X Axis check can only be placed in one check box for each translational axis action.

+ By placing a check in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + X direction. This restraint resist movement in the negative X direction and allows movement in the positive X direction.

+ And - By placing a check in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - X direction. This restraint resist movement in the positive and negative X directions. In other words, all movement along the X-axis will be prevented.

- By placing a check in this check box, the User instructs TRIFLEX to apply one directional restraint acting in the - X direction. This restraint resist movement in the positive X direction and allows movement in the negative X direction.

Limit Stops - By placing a check in this check box, the User instructs TRIFLEX to apply limit stop acting along the X-axis.

A limit stop is a device that will prevent further movement of a pipe after it has moved a specified allowed distance. This type of restraining action has also been referred to as a gap element. Through the use of limit stops and the limit fields, it is possible to code movement limits for a data point. It is also possible to code an initial movement of the pipe with the condition that if the pipe would tend to move away from this point, it may. By simply coding any one limit and zero (0) as the other limit, an one-directional limit stop may be coded. Users may code different gap spaces in each direction (positive and negative). In addition, both gaps can be specified with the same sign resulting in an initial movement being imposed and then a gap until the larger movement is encountered.

When a check is placed in the limit stop check box, the labels of the following three fields will be altered to allow the User to enter the following data:

Upper Limit – In this field, the User may specify the upper limit for the limit stop along the X-axis. The upper limit will be the most positive value for the limit stop.

Lower Limit – In this field, the User may specify the lower limit for the limit stop along the X-axis. The lower limit will be the least positive value for the limit stop.

Stiffness - In this field, the User may specify the stiffness of the limit stop along the X-axis. This stiffness will only come into effect after the pipe has deflected freely to the limit position of the limit stop. If the User specifies no value for stiffness, TRIFLEX will assume the limit stop restraint to be totally rigid. To enter a definable value for the stiffness, the User must type the desired numerical value in this field.

When no check mark is placed in any of the four check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative X direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the X-axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

Force - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative X direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive X direction, then the User need not enter any sign. When a User has entered a Force along the X-axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will imposed this force on the piping system and will continue to apply this Force no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

When no check mark is placed in any of the top four check boxes and no values are entered for movement, force and stiffness, the User still has the option to instruct TRIFLEX to apply a damper at the Node Location as follows:

Damper - By placing a check in this check box, the User instructs TRIFLEX to apply a damper acting in the + and - X direction. A damper is a two directional restraint that is considered to be totally rigid when an occasional loading case is being processed and totally free when an operating case is being processed. In other words, all movement along the X-axis will be allowed by the damper in the operating case but will be prevented in the occasional load case.

NOTE: Y-Axis check can only be placed in one check box for each translational axis action.

+ By placing a check in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + Y direction. These restraints resist movement in the negative Y direction and allow movement in the positive Y direction.

+ And - By placing a check in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - Y direction. These restraints resist movement in the positive and negative Y directions. In other words, all movement along the Y-axis will be prevented.

- By placing a check in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - Y direction. These restraints resist movement in the positive Y direction and allow movement in the negative Y direction.

Limit Stops - By placing a check in this check box, the User instructs TRIFLEX to apply limit stop acting along the Y-axis. For more details concerning the application of a typical limit stop, see the discussion for limit stop acting along the X-axis.

When a check is placed in the limit stop check box, the labels of the following three fields will be altered to allow the User to enter the following data:

Upper Limit – In this field, the User may specify the upper limit for the limit stop along the Y-axis. The upper limit will be the most positive value for the limit stop.

Lower Limit – In this field, the User may specify the lower limit for the limit stop along the Y-axis. The lower limit will be the least positive value for the limit stop.

Stiffness - In this field, the User may specify the stiffness of the limit stop along the Y-axis. This stiffness will only come into effect after the pipe has deflected freely to the limit position of the limit stop. If the User specifies no value for stiffness, TRIFLEX will assume the limit stop restraint to be totally rigid. To enter a definable value for the stiffness, the User must type the desired numerical value in this field.

When no check mark is placed in any of the four check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative Y direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the Y-axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

Force - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative Y direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive Y direction, then the User need not enter any sign. When a User has entered a Force along the Y-axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will imposed this force on the piping system and will continue to apply this Force no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

When no check mark is placed in any of the top four check boxes and no values are entered for movement, force and stiffness, the User still has the option to instruct TRIFLEX to apply a damper at the Node Location as follows:

Damper - By placing a check in this check box, the User instructs TRIFLEX to apply a damper acting in the + and - Y direction. A damper is a two directional restraint that is considered to be totally rigid when an occasional loading case is being processed and totally free when an operating case is being processed. In other words, all movement along the Y-axis will be allowed by the damper in the operating case but will be prevented in the occasional load case.

Note: Z Axis check can only be placed in one check box for each translational axis action.

+ By placing a check in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + Z direction. This restraint resists movement in the negative Z direction and allows movement in the positive Z direction.

+ And - By placing a check in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - Z direction. These restraints resist movement in the positive and negative Z directions. In other words, all movement along the Z-axis will be prevented.

- By placing a check in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - Z direction. These restraints resist movement in the positive Z direction and allow movement in the negative Z direction.

Limit Stops - By placing a check in this check box, the User instructs TRIFLEX to apply limit stop acting along the Z-axis. For more details concerning the application of a typical limit stop, see the discussion for limit stop acting along the X-axis

When a check is placed in the limit stop check box, the labels of the following three fields will be altered to allow the User to enter the following data:

Upper Limit – In this field, the User may specify the upper limit for the limit stop along the Z-axis. The upper limit will be the most positive value for the limit stop.

Lower Limit – In this field, the User may specify the lower limit for the limit stop along the Z-axis. The lower limit will be the least positive value for the limit stop.

Stiffness - In this field, the User may specify the stiffness of the limit stop along the Z-axis. This stiffness will only come into effect after the pipe has deflected freely to the limit position of the limit stop. If the User specifies no value for stiffness, TRIFLEX will assume the limit stop restraint to be totally rigid. To enter a definable value for the stiffness, the User must type the desired numerical value in this field.

When no check mark is placed in any of the four check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative Z direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the Z-axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

Force - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative Z direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive Z direction, then the User need not enter any sign. When a User has entered a Force along the Z-axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will imposed this force on the piping system and will continue to apply this Force no matter where the piping system moves.

Stiffness - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

When no check mark is placed in any of the top four check boxes and no values are entered for movement, force and stiffness, the User still has the option to instruct TRIFLEX to apply a damper at the Node Location as follows:

Damper -By placing a check in this check box, the User instructs TRIFLEX to apply a damper acting in the + and - Z direction. A damper is a two directional restraint that is considered to be totally rigid when an occasional loading case is being processed and totally free when an operating case is being processed. In other words, all movement along the Z-axis will be allowed by the damper in the operating case but will be prevented in the occasional load case.

Immediately below the data group entitled "Translational Restraint Action", the User will find a data group entitled "Rotational Restraint Action". When the X, Y, Z coordinate system is selected by the User, TRIFLEX will display the Rotational Restraint Action data group with the X axis, Y axis and Z axis headings. When the L, N, G coordinate system is selected by the User, TRIFLEX will display the Rotational Restraint Action data group with the L axis, N axis and G axis headings. When the A, B, C coordinate system is selected by the User, TRIFLEX will display the Rotational Restraint Action data group with the A axis, B axis and C axis headings. The fields in which data can be entered in this data group are defined below:

X, Y, Z coordinate system

X Axis

+ And - By placing a check in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the X axis. These restraints resist rotation about the X-axis in the positive and negative directions. In other words, all rotations about the X-axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

Rotation - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the X-axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the X-axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the X-axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the X-axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the X-axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Y Axis

+ And - By placing a check in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the Y axis. These restraints resist rotation about the Y-axis in the positive and negative directions. In other words, all rotations about the Y-axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

Rotation - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the Y-axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the Y-axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the Y-axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the Y-axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the Y-axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Z Axis

+ And - By placing a check in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the Z axis. These restraints resist rotation about the Z-axis in the positive and negative directions. In other words, all rotations about the Z-axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

Rotation - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the Z-axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the Z-axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the Z-axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the Z-axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the Z-axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

L, N, G coordinate system

Note: L Axis check can only be placed in one check box for each translational axis action. The L axis is coincident with the axis of the pipe.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - L direction. This restraint will resist movement in the positive and negative L directions. In other words, all movement along the axis of the pipe will be prevented.

Limit Stops - By placing a check mark in this check box, the User instructs TRIFLEX to apply limit stop acting along the L axis.

A limit stop is a device that will prevent further movement of a pipe after it has moved a specified allowed distance. This type of restraining action has also been referred to as a gap element. Through the use of limit stops and the limit fields, it is possible to code movement limits for a data point. It is also possible to code an initial movement of the pipe with the condition that if the pipe would tend to move away from this point, it may. By simply coding any one limit and zero (0) as the other limit, an one-directional limit stop may be coded. Users may code different gap spaces in each direction (positive and negative). In addition, both gaps can be specified with the same sign resulting in an initial movement being imposed and then a gap until the larger movement is encountered.

When a check is placed in the limit stop check box, the labels of the following three fields will be altered to allow the User to enter the following data:

Upper Limit – In this field, the User may specify the upper limit for the limit stop along the L axis. The upper limit will be the most positive value for the limit stop.

Lower Limit – In this field, the User may specify the lower limit for the limit stop along the L axis. The lower limit will be the least positive value for the limit stop.

Stiffness - In this field, the User may specify the stiffness of the limit stop along the L axis. This stiffness will only come into effect after the pipe has deflected freely to the limit position of the limit stop. If the User specifies no value for stiffness, TRIFLEX will assume the limit stop restraint to be totally rigid. To enter a definable value for the stiffness, the User must type the desired numerical value in this field.

Note: N Axis check mark can only be placed in one check box for each translational axis action). The N axis is normal to the pipe and the +N direction is the most vertical.

+ By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + N direction. These restraints resist movement in the negative N direction and allow movement in the positive N direction.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - N direction. These restraints resist movement in the positive and negative N directions. In other words, all movement along the N axis will be prevented.

- By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - Y direction. These restraints resist movement in the positive N direction and allow movement in the negative N direction.

Limit Stops - By placing a check mark in this check box, the User instructs TRIFLEX to apply limit stop acting along the N axis. For more details concerning the application of a typical limit stop, see the discussion for limit stop acting along the L axis.

When a check mark is placed in the limit stop check box, the labels of the following three fields will be altered to allow the User to enter the following data:

Upper Limit – In this field, the User may specify the upper limit for the limit stop along the N axis. The upper limit will be the most positive value for the limit stop.

Lower Limit – In this field, the User may specify the lower limit for the limit stop along the N axis. The lower limit will be the least positive value for the limit stop.

Stiffness - In this field, the User may specify the stiffness of the limit stop along the N axis. This stiffness will only come into effect after the pipe has deflected freely to the limit position of the limit stop. If the User specifies no value for stiffness, TRIFLEX will assume the limit stop restraint to be totally rigid. To enter a definable value for the stiffness, the User must type the desired numerical value in this field.

Note: G Axis check can only be placed in one check box for each translational axis action. The G axis is normal to the pipe and the most horizontal.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - G direction. These restraints resist movement in the positive and negative G directions. In other words, all movement along the Z-axis will be prevented.

Limit Stops - By placing a check mark in this check box, the User instructs TRIFLEX to apply limit stop acting along the G axis. For more details concerning the application of a typical limit stop, see the discussion for limit stop acting along the L axis

When a check mark is placed in the limit stop check box, the labels of the following three fields will be altered to allow the User to enter the following data:

Upper Limit – In this field, the User may specify the upper limit for the limit stop along the G axis. The upper limit will be the most positive value for the limit stop.

Lower Limit – In this field, the User may specify the lower limit for the limit stop along the G axis. The lower limit will be the least positive value for the limit stop.

Stiffness - In this field, the User may specify the stiffness of the limit stop along the G axis. This stiffness will only come into effect after the pipe has deflected freely to the limit position of the limit stop. If the User specifies no value for stiffness, TRIFLEX will assume the limit stop restraint to be totally rigid. To enter a definable value for the stiffness, the User must type the desired numerical value in this field.

Immediately below the data group entitled "Translational Restraint Action", the User will find a data group entitled "Rotational Restraint Action". Rotational restraints may not be specified when the User has selected the L, N, G coordinate system. Therefore, all of the data fields in this data group are grayed out. The User can enter no data in this data group.

A, B, C coordinate system (with Use Directional Vectors selected and the X vector, the Y vector and the Z vector specified)

Note: An Axis check mark can only be placed in one check box for each translational axis action. The resultant of the X vector, the Y vector and the Z vector defines the A axis.

+ By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + A direction. These restraints resist movement in the negative a direction and allow movement in the positive a direction.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - A direction. These restraints resist movement in the positive and negative a direction. In other words, all movement (plus or minus) along the A axis will be prevented.

- By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - A direction. These restraints resist movement in the positive a direction and allow movement in the negative a direction.

Limit Stops - Limit stops may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

When no check mark is placed in any of the check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative a direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the A axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

Force - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative a direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive a direction, then the User need not enter any sign. When a User has entered a Force along the A axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will impose this force on the piping system and will continue to apply this Force no matter where the piping system moves.

Stiffness - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Damper - Dampers may not be specified when the User has selected the A, B, C coordinate system and has selected the Use Directional Vectors option. Therefore, this data field is grayed out.

Note: That in the B Axis no data can be entered to describe a restraint acting along the B axis and therefore all data fields are grayed out.

Note: That in the C Axis no data can be entered to describe a restraint acting along the C axis and therefore all data fields are grayed out.

Immediately below the data group entitled “Translational Restraint Action”, the User will find a data group entitled “Rotational Restraint Action”. The fields in which data can be entered in this data group are defined below:

A, B, C coordinate system (with Use Directional Vectors selected and the X vector, the Y vector and the Z vector specified)

An Axis

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the A axis. This restraint resist rotation about the A axis in the positive and negative directions. In other words, all rotations about the A axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

Rotation - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the A axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the A axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the A axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the A axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the A axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Note: That in the B Axis no data can be entered to describe a restraint acting about the B axis and therefore all data fields are grayed out. That in the C Axis no data can be entered to describe a restraint acting about the C axis and therefore all data fields are grayed out.

A, B, C coordinate system (with Use Action Angles selected and the A-X, A-Y, A-Z, C-X, C-Y and C-Z angles specified)

Note: That in the A Axis a check mark can only be placed in one check box for each translational axis action.

+ By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + A direction. These restraints resist movement in the negative a direction and allow movement in the positive a direction.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - A direction. These restraints resist movement in the positive and negative a direction. In other words, all movement along the A axis will be prevented.

- By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - A direction. These restraints resist movement in the positive a direction and allow movement in the negative a direction.

Limit Stops - Limit stops may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

When no check mark is placed in any of the check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative a direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the A axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

Force - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative a direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive a direction, then the User need not enter any sign. When a User has entered a Force along the A axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will impose this force on the piping system and will continue to apply this Force no matter where the piping system moves.

Stiffness - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Damper -Dampers may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

Note: That in the B Axis a check mark can only be placed in one check box for each translational axis action.

+ By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + B direction. These restraints resist movement in the negative B direction and allow movement in the positive B direction.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - B direction. These restraints resist movement in the positive and negative B directions. In other words, all movement along the B axis will be prevented.

- By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - B direction. These restraints resist movement in the positive B direction and allow movement in the negative B direction.

Limit Stops - Limit stops may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

When no check mark is placed in any of the check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative B direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the B axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

Force - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative B direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive B direction, then the User need not enter any sign. When a User has entered a Force along the B axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will impose this force on the piping system and will continue to apply this Force no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Damper - Dampers may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

Note: That in the C Axis a check mark can only be placed in one check box for each translational axis action.

+ By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the + C direction. These restraints resist movement in the negative C direction and allow movement in the positive C direction.

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional translational restraint acting in the + and - C direction. These restraints resist movement in the positive and negative C directions. In other words, all movement along the C axis will be prevented.

- By placing a check mark in this check box, the User instructs TRIFLEX to apply a one directional restraint acting in the - C direction. These restraints resist movement in the positive C direction and allow movement in the negative C direction.

Limit Stops - Limit stops may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

When no check mark is placed in any of the check boxes at the top of this column, the labels of the three fields following the Limit Stop check box will be as set forth below and will allow the User to enter the following data:

Movement - In this field, the User may define a Movement that the User wishes to impose on the pipe at the restraint location. If the Movement is to be applied to the pipe in the negative C direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Movement along the C axis, the Force and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a movement in this field, TRIFLEX will impose this Movement on the piping system and will hold the piping system at that position in the analysis.

<u>Force</u> - In this field, the User may define a Force that the User wishes to impose on the pipe at the restraint location. If the Force is to be applied to the pipe in the negative C direction, then the User must enter the numerical value preceded by a negative sign. If the Force is to be applied to the pipe in the positive C direction, then the User need not enter any sign. When a User has entered a Force along the C axis, the Stiffness field will default to Free and the Movement field will be grayed out in order to prevent a movement from being entered in this field by the User. When the User enters a numerical value for the Force in this field, TRIFLEX will impose this force on the piping system and will continue to apply this Force no matter where the piping system moves.

<u>Stiffness</u> - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Damper - Dampers may not be specified when the User has selected the A, B, C coordinate system. Therefore, this data field is grayed out.

Immediately below the data group entitled “Translational Restraint Action”, the User will find a data group entitled “Rotational Restraint Action”. The fields in which data can be entered in this data group are defined below:

<u>A, B, C coordinate system</u> (with Use Action Angles selected and the A-X, A-Y, A-Z, C-X, C-Y and C-Z angles specified)

An Axis

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the A axis. This restraint resist rotation about the A axis in the positive and negative directions. In other words, all rotations about the A axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

<u>Rotation</u> - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the A axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the A axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the A axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the A axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the A axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is "FREE". To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

B Axis

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the B axis. This restraint resists rotation about the B axis in the positive and negative directions. In other words, all rotations about the B axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

Rotation - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the B axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the B axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the B axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the B axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the B axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

C Axis

+ And - By placing a check mark in this check box, the User instructs TRIFLEX to apply a two directional rotational restraint acting about the C axis. This restraint resists rotation about the C axis in the positive and negative directions. In other words, all rotations about the C axis will be prevented.

When no check mark is placed in the + and - check box, the User may enter data in the following three fields:

Rotation - In this field, the User may define a Rotation that the User wishes to impose on the pipe at the restraint location. If the Rotation is to be applied to the pipe about the C axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. When the User has entered a Rotation about the C axis, the Moment and Stiffness fields are grayed out in order to prevent the User from entering data in these fields. When the User enters a rotation in this field, TRIFLEX will impose this Rotation on the piping system and will hold the piping system at that position in the analysis.

Moment - In this field, the User may define a Moment that the User wishes to impose on the pipe at the restraint location. If the Moment is to be applied to the pipe about the C axis in the negative direction, then the User must enter the numerical value preceded by a negative sign. If the Moment is to be applied to the pipe about the C axis in the positive direction, then the User need not enter any sign. When a User has entered a Moment about the C axis, the Stiffness field will default to Free and the Rotation field will be grayed out in order to prevent a rotation from being entered in this field by the User. When the User enters a numerical value for the Moment in this field, TRIFLEX will impose this moment on the piping system and will continue to apply this Moment no matter where the piping system moves.

Stiffness - The default for the Stiffness field is “FREE”. To enter a definable value for the stiffness, the User must type the desired numerical value in this field. To make the stiffness Free after having entered some other value, simply type F and TRIFLEX will display the word FREE.

Immediately below the data group entitled "Rotational Restraint Action", the User will find a data group entitled "Spring Hanger". The fields in which data can be entered in this data group are defined below:

Size a Spring Hanger - By placing a check mark in this check box, the User can instruct TRIFLEX to size a spring hanger at this node location. When the X, Y, Z coordinate system is selected, the spring hanger will be considered to act along the Y-axis. When the L, N, G coordinate system is selected by the User, the spring hanger will be considered to act along the N axis. When the A, B, C coordinate system is selected by the User, the spring hanger will be considered to act along the B axis. Please note that the User may not place a check mark in the "Existing Spring Hanger" check box if a check mark has been placed in this "Size a Spring Hanger" check box.

Allowed Load Variation – The default value that appears in this field is 25 percent. The User can enter any other desired numerical value in this field.

No. Of Spring Hangers - The default value that appears in this field is "1" which means that TRIFLEX will default to sizing one spring hanger at this location. The User can enter another desired numerical value in this field to indicate the number of spring hangers that the User wants TRIFLEX to size at this location. If the User enters a number of two or more, TRIFLEX will divide the total load carried at this node location by the number of desired spring hangers and will size the hangers based upon the resulting loads.

Existing Spring Hanger - By placing a check mark in this check box, the User can instruct TRIFLEX to use the existing spring hanger data entered by the User in the following two fields at this node location. When the X, Y, Z coordinate system is selected, the spring hanger will be considered to act along the Y-axis. When the L, N, G coordinate system is selected by the User, the spring hanger will be considered to act along the N axis. When the A, B, C coordinate system is selected by the User, the spring hanger will be considered to act along the B axis. Please note that the User may not place a check mark in the "Size a Spring Hanger" check box if a check mark has been placed in this "Existing Spring Hanger" check box.

Installed Load - When modeling an existing spring, the User should enter the installed load and the spring rate (in the following field) for the spring hanger. If the installed load is unknown, the User should enter the operating load with a spring rate of 1. By specifying the operating load as essentially a constant load, the load applied to the piping system by TRIFLEX at this location at operating conditions will be equal to the load found in the field at operating conditions. The movement from this type of analysis may be used to re-size the spring hanger, if the User desires.

Spring Rate – The User should enter the spring rate for the spring hanger only when the existing installed load is known. If the installed load is unknown, the User should enter the operating load with a spring rate of 1.

3.6 Coding Elbow Data

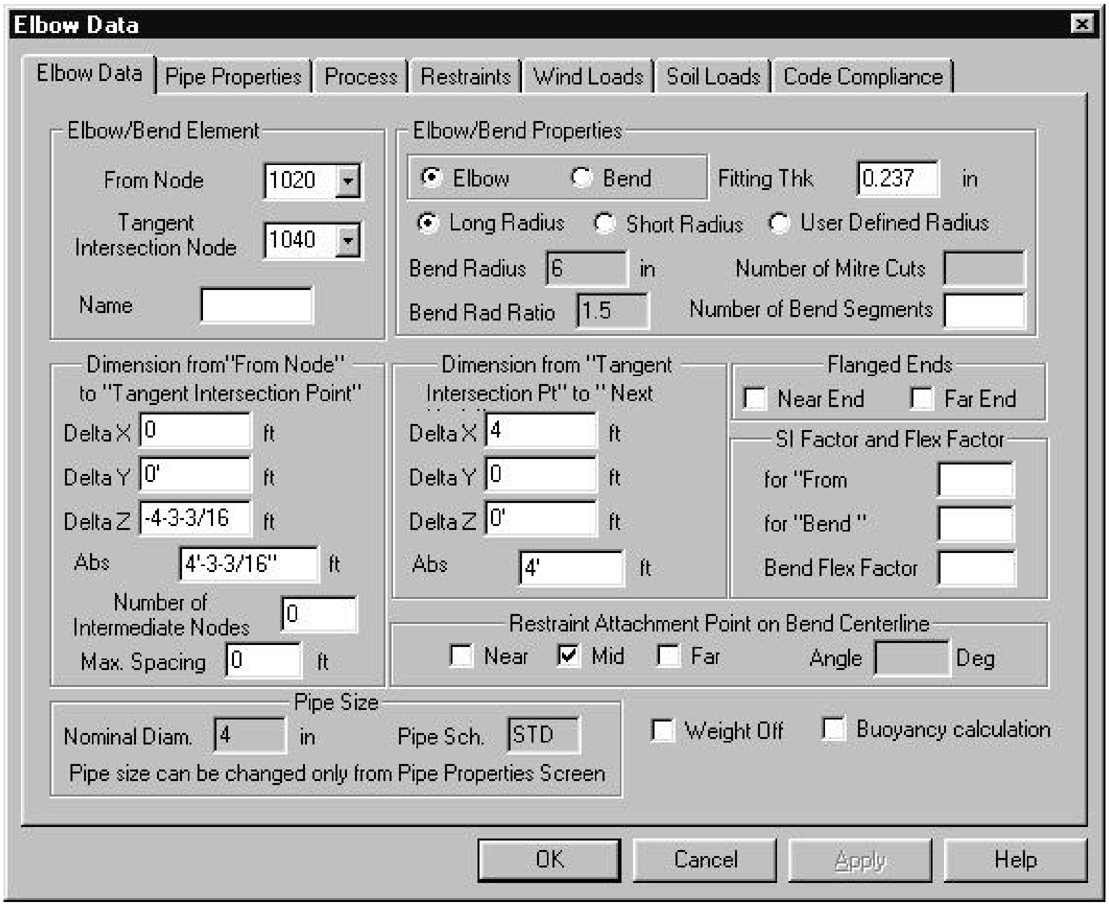


Figure 3.6.1 – Coding Elbow Data, Elbow data Tab

3.6.1 Coding Elbow Data, Elbow Data Tab

To enter an Elbow or Bend component, the User must click on the Elbow Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Elbow on the resulting pull down menu. Upon either of these sequences of actions, an Elbow dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Elbow dialog, a data group entitled "Elbow/Bend Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

Tangent Intersection Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

To the immediate right of the “Elbow/Bend Element” data, the User will find a data group entitled “Elbow/Bend Properties”. The fields in which data can be entered in this data group are defined below:

Elbow or Bend and Fitting Thickness – The first selection to be made by the User is whether the component is an Elbow or a Bend. The default selection is “Elbow”. The radio button just to the left of the Elbow label will be selected indicating that this is the default selection. When the User selects Elbow, the User may also specify the fitting thickness, if it is different from the pipe wall thickness. The default fitting thickness will be the standard pipe wall thickness. The fitting thickness will be used only for the elbow itself, not for the preceding pipe wall thickness, if any. If the User wishes to enter a Bend, the User must select the radio button just to the left of the Bend label.

Long Radius / Short Radius / User Defined Radius – In the next row of fields, the user must select one radio button. The alternatives are Long Radius (the bend radius will be set to 1.5 times the nominal pipe diameter) or Short Radius (the bend radius will be set to 1.0 times the nominal pipe diameter) or User Defined Radius in which the user may specify the desired bend radius or bend radius ratio. The default is set by TRIFLEX to Long Radius.

When the User selects either Long Radius or Short Radius, the bend radius ratio and the bend radius to be used by TRIFLEX will be displayed in the Bend Radius and Bend Radius Ratio fields just below the Long and Short Radius radio buttons. Note that these fields are grayed out and the User may not edit the data in these fields. The data in these fields is calculated by TRIFLEX based upon the Long or Short selection by the User.

When the User selects User Defined Radius, the bend radius ratio and the bend radius to be used by TRIFLEX must be entered by the User. The Bend Radius and Bend Radius Ratio fields are just below the Long and Short Radius radio buttons. Note that these fields are not grayed out now and the User must enter the desired data in these fields.

Number of Miter Cuts – If the User wishes to define the entered bend as a Miter Bend, then the User should specify the number of miter cuts in the field provided. TRIFLEX will automatically determine if the miter bend is closely spaced or widely spaced and the appropriate equations as defined in the piping codes will be used by TRIFLEX. Note that Restraints may not be specified on a widely spaced miter bend when specifying more than 1 miter point.

Number of Bend Segments – If the User has entered a bend, TRIFLEX will split the bend into two segments each equal to one half of the angle. If the User wants TRIFLEX to break the bend into more than two segments, the User may specify the number of segments desired and TRIFLEX will break the bend into the desired number of arcs. Note that a restraint may not be entered on a bend consisting of more than two bend segments or arcs.

Immediately below the "Elbow/Bend Element" data, the User will find a data group entitled "Dimension from "From Node" to "Tangent Intersection Point". The dimension(s) entered in this data group define the vector from the previous node point to the Tangent Intersection Point of the elbow or bend being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z– If the User is specifying the first component after an anchor, TRIFLEX will assume an "X" dimension equal to the minimum allowed for a ninety (90) degree elbow based upon the properties already entered by the User. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or more axes or if it is to be longer, the User may simply enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will calculate the Delta X, Delta Y and Delta Z dimensions automatically and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of any preceding component, and then the User should place a check in the box immediately to the left of the label "Use the Minimum Length". TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the Elbow or Bend being modeled follows a Bend or a Valve, Flange or Joint with the data point specified at a point other than the end point.

<u>Minimum Length</u> – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Tangent Intersection Point Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

<u>Number of Intermediate Nodes</u> – TRIFLEX will break the elbow/bend and preceding pipe into one more segment that the number of intermediate nodes specified by the User in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the tangent intersection point and the previous or from node – TRIFLEX will break the preceding pipe into three (3) segments. Note that an intermediate node point cannot be placed on the bend itself; it must be on the preceding pipe component.

<u>Maximum Spacing</u> – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe components generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

Immediately to the right of the data group entitled "Dimension from "From Node" to "Tangent Intersection Point", the User will find data group entitled "Dimension from "Tangent Intersection Point" to "Next Node". The dimension(s) entered in this data group define the vector from the Tangent Intersection Point of the elbow or bend being entered to the Next Node Point. The next node point may be at any point on the following pipe component or on the following valve or flange or joint or on the following elbow, etc. TRIFLEX will default to delta dimension that will yield a ninety-degree bend or elbow and will be in the most "Y" direction by default. The fields in which data can be entered in this data group are further defined below:

<u>Delta X, Delta Y and Delta Z</u> – A length equal to the bend radius is defaulted to by TRIFLEX. If this dimension is incorrect or if the delta dimension should be along another axis or along two or more axes or if it is to be longer, the User may simply enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

<u>Abs Length</u> – Given the Delta X, Delta Y and Delta Z dimensions, TRIFLEX will automatically calculate the absolute length and display it in this field.

Immediately to the right of the data group entitled "Dimension from Tangent Intersection Point" to "Next Node", the User will find a data group entitled "Flanged Ends". Two check boxes are provided for the User to indicate if the ends are to be considered to be flanged or not. If either end or both ends are checked, TRIFLEX will modify the flexibility of the bend or elbow in accordance with the provisions of the specified piping code. The fields in which data can be entered in this data group are further defined below:

Near End - If the User wishes to tell TRIFLEX that the "Near End" of the elbow or bend is to be considered as flanged, then the User should place a check in the box immediately to the left of the label "Near End".

Far End - If the User wishes to tell TRIFLEX that the "Far End" of the elbow or bend is to be considered as flanged, then the User should place a check in the box immediately to the left of "Far End".

Immediately below the data group entitled "Flanged Ends", the User will find a data group entitled "SI Factors and Flex Factor". The fields in which data can be entered in this data group are further defined below:

For "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the pipe section preceding the elbow or bend defined in this component, then the User should enter the desired numerical value in this field.

For "Bend" - If the User wishes to specify a special stress intensification factor on the elbow or bend defined in this component, then the User should enter the desired numerical value in this field.

Bend Flex Factor - If the User wishes to specify a special flexibility factor for the elbow or bend defined in this component, then the User should enter the desired numerical value in this field.

Immediately below the data groups entitled "Flanged Ends" and "Dimension from Tangent Intersection Point" to "Next Node", the User will find a data group entitled "Restraint Attachment Point on Bend Centerline". In this data group, the User can tell TRIFLEX where on the bend or elbow centerline the User wishes to attach a restraint. The fields in which data can be entered in this data group are further defined below:

Near - If the User wishes to tell TRIFLEX to attach the entered restraint on the centerline of the elbow or bend at the near end of the elbow or bend, then the User should place a check in the box immediately to the left of the label "Near".

Mid - If the User wishes to tell TRIFLEX to attach the entered restraint on the centerline of the elbow or bend at the mid point of the elbow or bend, then the User should place a check in the box immediately to the left of the label "Mid".

Far - If the User wishes to tell TRIFLEX to attach the entered restraint on the centerline of the elbow or bend at the far end of the elbow or bend, then the User should place a check in the box immediately to the left of the label "Far".

Angle Deg - If the User wishes to tell TRIFLEX to attach the entered restraint on the centerline of the elbow or bend at a specific angle from the near end of the elbow or bend, then the User should enter the number of degrees from the Near End to the attachment point in the blank provide.

Immediately below the Dimension data groups, the User will find a data group entitled "Pipe Size". In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

Immediately to the right of the "Pipe Size" data group, the User will find additional data fields for miscellaneous data defined as follows:

Weight Off - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label "Weight Off". The default is for weight to be considered.

Buoyancy - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label "Buoyancy Calculations". The density of the fluid surrounding the pipe should also be specified on the **Setup / Modeling Defaults dialog**. The default is for the effects of buoyancy not to be considered.

3.7 Coding Branch Connection, Branch Connection

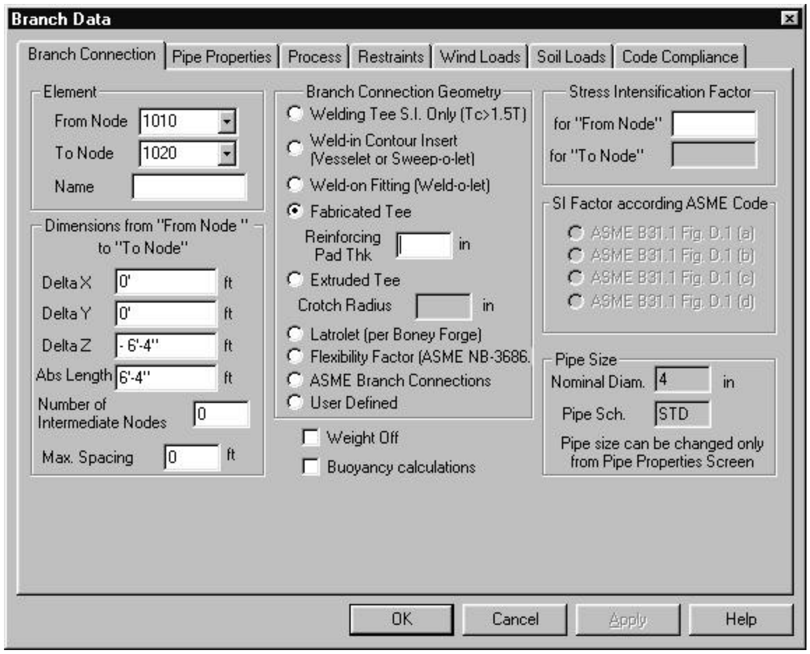


Figure 3.7.1 Coding Branch Connection, Branch Connection Tab

3.7.1 Coding Branch Connection, Branch Connection Tab

To enter a Branch Connection component, the User must click on the Branch Connection Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Branch Connection on the resulting pull down menu. Upon either of these sequences of actions, a Branch Connection dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Branch Connection dialog, a data group entitled "Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Dimension from "From Node" to "To Node". The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the end point) of the Branch Connection being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z – If the User is specifying the first component after an anchor, TRIFLEX will assume an "X" dimension equal to one foot if English units are specified or .35 Meters if metric units are specified. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or more axes, the User may simply enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will calculate the Delta X, Delta Y and Delta Z dimensions automatically and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of any preceding component, and then the User should place a check in the box immediately to the left of the label "Use the Minimum Length". TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the Pipe being modeled follows a Bend or a Valve, Flange or Joint with the data point specified at a point other than the end point.

Minimum Length – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

Number of Intermediate Nodes – If the User enters a number in this field, TRIFLEX will break the pipe leading into the Branch Connection into one more segment that the number of intermediate nodes specified in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the To node and the From node – TRIFLEX will break the pipe leading into the Branch Connection into three (3) segments.

Maximum Spacing – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe leading into the Branch Connection component generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

To the immediate right of the "Element" data, the User will find a data group entitled "Branch Connection Geometry". The User may select one radio button and, in some selections, additional data fields will be made active for the User to enter additional data as defined below. The User is to code a branch connection component only the first time the User defines the branch connection as a To Node. If the User codes away from the branch connection or codes into the branch connection again, the User need only define these members as Pipe components and no Stress Intensification Factors need be indicated. TRIFLEX will automatically intensify all three branches of a branch connection.

Welding Tee S.I. Only (Tc > 1.5 T) – The Welding Tee S.I. Only radio button is the default selection. By accepting the radio button "Welding Tee S.I. Only", the User instructs TRIFLEX to consider the end of the Pipe defined in this component to be a branch intersection and for all three pipes intersecting at this point to be intensified by the applicable piping code stress intensification factors for a welding tee.

Weld-in Contour Insert (Vesselet® or Sweep-o-let®) - By selecting the radio button "Weld-in Contour Insert", the User instructs TRIFLEX to consider the end of the Pipe defined in this component to be a branch intersection and for all three pipes intersecting at this point to be intensified by the applicable piping code stress intensification factors for a weld-in contour insert.

Weld-on Fitting (Pipet® or Weld-o-let®) - By selecting the radio button "Weld-on Fitting, the User instructs TRIFLEX to consider the end of the Pipe defined in this component to be a branch intersection and for all three pipes intersecting at this point to be intensified by the applicable piping code stress intensification factors for a weld-on fitting.

Fabricated Tee - By selecting the radio button "Fabricated Tee", the User instructs TRIFLEX to consider the end of the Pipe defined in this component to be a branch intersection and for all three pipes intersecting at this point to be intensified by the applicable piping code stress intensification factors for a fabricated tee. When the Fabricated Tee radio button is selected, the reinforcing pad thickness field is made active for the User to enter a reinforcing pad thickness, if additional reinforcement is provided at the branch intersection. Entry of the reinforcing pad thickness is optional.

Extruded Tee (Tc < 1.5T) (not an extrusion tee) - By selecting the radio button "Extruded Tee (Tc < 1.5T", the User instructs TRIFLEX to consider the end of the Pipe defined in this component to be a branch intersection and for all three pipes intersecting at this point to be intensified by the applicable piping code stress intensification factors for an extruded tee. When the Extruded Tee radio button is selected, the crotch radius field is made active for the User to enter the applicable crotch radius. Entry of the crotch radius is mandatory. **Note:** An extrusion tee is not the same as an extruded tee. If the User has an extrusion tee, it is highly recommended that the User consult with the vendor to obtain the correct stress intensification factors.

Latrolet® (per Bonney Forge) **-** By selecting the radio button "Latrolet®", the User instructs TRIFLEX to consider the end of the Pipe defined in this component to be a branch intersection and for all three pipes intersecting at this point to be intensified by the applicable piping code stress intensification factors for a Latrolet.

When the "ASME Branch Connections" radio button is selected, TRIFLEX will activate the SI Factors according to ASME Code data group that can be found just below the Stress Intensification Factor data group on the right edge of the dialog. TRIFLEX will default to ASME B31.1 Fig. D1 (a). This means that TRIFLEX will calculate the stress intensification factors in accordance with the equation set forth in ASME B31.1 Fig. D1 (a). If the User wishes, ASME B31.1 Fig. D1 (b), ASME B31.1 Fig. D1 (c) or ASME B31.1 Fig. D1 (d) may be selected by clicking on the radio button just to the left of each such field. For more information about these equations, please refer to ASME B31.1 Fig. D1.

User Defined - When the radio button "User Defined" is selected, TRIFLEX will activate the "for To Node" SI Factor in the stress intensification factor data group. See the discussion for this data group for more details.

Immediately below the "Branch Connection Geometry" data group, the User will find additional data fields for miscellaneous data defined as follows:

Weight Off - If the User wishes to instruct TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label "Weight Off". The default is for weight to be considered.

Buoyancy - If the User wishes to instruct TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label "Buoyancy Calculations". The density of the fluid surrounding the Branch Connection should also be specified on the **Setup /**

Modeling Defaults dialog. The default is for the effects of buoyancy not to be considered.

Immediately to the right of the data group entitled "Branch Connection Geometry", the User will find a data group entitled "Stress Intensification Factor". The fields in which data can be entered in this data group are further defined below:

For "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the Branch Connection component, then the User should enter the desired numerical value in this field.

For "To Node" - If the User wishes to specify a numerical stress intensification factor on the end of the Branch Connection component, then the User should enter the desired numerical value in this field. This value will be used on all pipes intersecting at this branch connection point.

Immediately below the "Stress Intensification Factor" data group, the User will find a data group entitled "SI Factor according to ASME Code". The details of this data group have been given under the ASME Branch Connections discussion.

Immediately below the "SI Factor according to ASME Code" data group, the User will find a data group entitled "Pipe Size". In this data group, the User can see the Pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the Pipe diameter or the Pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

If the user enters **A**, **B**, **C**, or **D** in **SIE** field then the following field changes occur:

1. Pthk becomes L1

2. Crotch becomes pad dia

3. Bflex becomes theta

These fields are explained below:

L1 (in, mm, cm, mm)

This is the effective pad thickness of an integrally reinforced branch connection. For a description and diagram of how this value is determined, see fig. NC-3673.3(b) - 2 in the ASME Class 2 code.

Pad dia (in, mm, cm, mm)

Outside diameter (including reinforcement) of an integrally reinforced branch connection.

Theta (degree) this is the angle between a vertical line and the slope of the integral reinforcement of a branch connection. For more information, see ANSI B32.1, fig. D.1.

========

A - D - ANSI B31.1, fig.D.1 (a) - (d)

ASME Class 2 figs. NC - 3673.3(b) - 2

ASME Class 3 Figs. ND - 3673.2(b) - 2

3.8 Coding Valve Data, Valve Data Tab

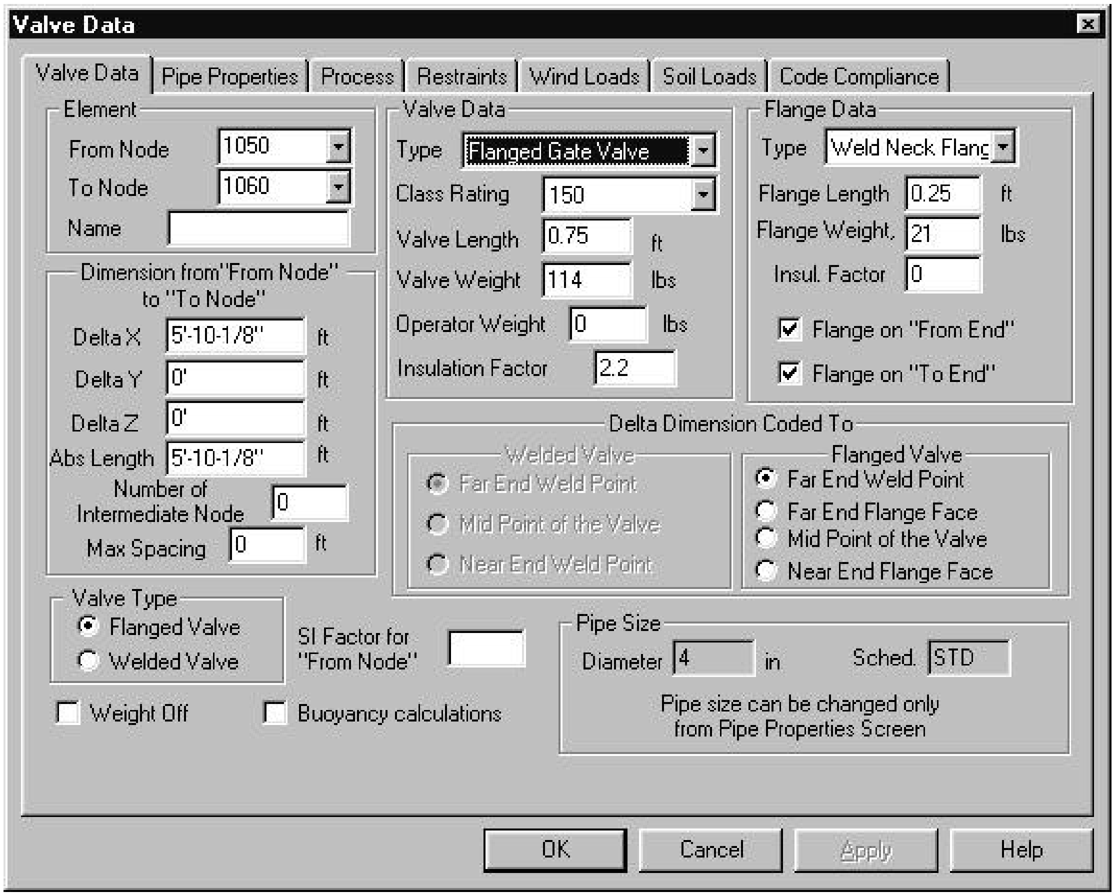


Figure 3.8.1 Coding Valve Data, Valve Data Tab

3.8.1 Coding Valve Data, Valve Data Tab

To enter a Valve component with or without a preceding Pipe, the User must click on the Valve Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Valve on the resulting pull down menu. Upon either of these sequences of actions, a Valve dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Valve dialog, a data group entitled “Element” is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Dimension from "From Node" to "To Node". The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the point where the User is placing the Node) of the valve being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z– If the User is specifying the first component after an anchor, TRIFLEX will assume an "X" dimension equal to one foot, if English units are specified, or .35 Meters, if metric units are specified. If the node being defined follows another component other than an anchor, the assumed dimension will be along the vector line defined by the previous component. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or three axes, the User may enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will automatically calculate the Delta X, Delta Y and Delta Z dimensions and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of the valve, any flanges and the preceding component, then the User should place a check in the box immediately to the left of the label "Use the Minimum Length". TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the User desires to place a valve fitting make-up with the previous component and it eliminates the need for the User to perform manual math calculations.

Minimum Length – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

Number of Intermediate Nodes – If the User enters a number in this field, TRIFLEX will break the pipe preceding the valve into one more segment that the number of intermediate nodes specified in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the to node and the from end of the valve. TRIFLEX will then break the pipe into three (3) segments.

Maximum Spacing – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe components generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

Immediately below the data group entitled "Dimension from "From Node" to "To Node", the User will find a data group entitled "Valve Type". In this data group, the User must instruct TRIFLEX whether the valve is a flanged valve or a welded valve.

Flanged Valve – The flanged valve radio button is the default selection. By accepting the radio button "Flanged Valve", the User is telling TRIFLEX that a flanged valve is desired.

Welded Valve - To tell TRIFLEX that a Welded Valve is desired, by clicking on the radio button immediately preceding "Welded Valve", the User is telling TRIFLEX that a welded valve is desired.

To the immediate right of the "Element" data group, the User will find a data group entitled "Valve Data". The fields in which data can be entered in this data group are defined below:

Type – The User must select a Valve Type from the drop down combo list in this field. The default valve type is the Flanged AAAT Standard Valve. From our staff's past experience, this valve is an average valve. The Type of valve, along with the Rating, allows TRIFLEX to search through the valve database to find the desired valve. The current TRIFLEX Valve database consists of four flanged valve types and four welded valve types. In addition, the User may select "User Specified" in order to be able to enter their own weight and length. The valve types available in TRIFLEX are:

Flanged – AAAT Std. Valve, Globe Valve, Gate Valve, Swing Check Valve and User Specified

Welded - AAAT Std. Valve, Globe Valve, Gate Valve, Swing Check Valve and User Specified

Class Rating - The User must select a class rating from the drop down combo list in this field. The available class ratings are 150, 300, 400, 600, 900 and 1500. Given the valve type and the class rating, TRIFLEX will look up the appropriate corresponding weight, length, and insulation factor to be used in the calculations.

Valve Length – The data shown in this field is looked up from the database by TRIFLEX or, if the User Specified valve is selected, the User can enter a desired value in this field.

Valve Weight – The data shown in this field is looked up from the database by TRIFLEX or, if the User Specified valve is selected, the User can enter a desired value in this field.

Operator Weight – the User enters the value shown in this field. The weight entered will be applied by TRIFLEX at the centric of the valve.

Insulation Factor (ft, mm, m, mm) - if the User has specified the Type and Class Rating, the program will extract the insulation factor for the specified valve from the database. The user can input an insulation factor by overriding the value extracted from the database. The insulation factor is a value that when multiplied times the insulation weight per unit length will result in the weight of insulation to be placed on the valve. For example, an insulation factor of 1.5 means that the weight of insulation to be added to the valve weight will be equivalent to the weight of one foot of insulation on the adjacent pipe times a 1.5 factor when using the ENG input units.

If the User wants to enter a valve length or a valve weight or an insulation factor that is different than those selected by TRIFLEX from the TRIFLEX database, then the User should select User Specified on the Valve Type and then enter the desired values.

To the immediate right of the “Valve Data” data group, the User will find a data group entitled “Flange Data”. The fields in which data can be entered in this data group are defined below:

Type – The User must select a Flange Type from the drop down combo list in this field. The default valve type is the AAAT Standard Flange. From our staff’s past experience, this flange is an average flange. The Type of flange, along with the Rating, allows TRIFLEX to search through the flange database to find the desired flange. The current TRIFLEX Flange database consists of five flange types. In addition, the User may select “User Specified” in order to be able to enter their own weight and length. The flange types available in TRIFLEX are:

AAAT Std. Flange, Blind Flange, Lap Joint Flange, Slip-on Flange, Weld Neck Flange and User Specified.

Flange Length – The data shown in this field is looked up from the database by TRIFLEX or, if the User Specified flange is selected, the User can enter a desired value in this field.

Flange Weight – The data shown in this field is looked up from the database by TRIFLEX or, if the User Specified flange is selected, the User can enter a desired value in this field.

Insulation Factor (ft, mm, m, mm) - if the User has specified the Type and Class Rating, the program will extract the insulation factor for the specified flange from the database. The user can input an insulation factor by overriding the value extracted from the database. The insulation factor is a value that when multiplied times the insulation weight per unit length of adjoining pipe will result in the weight of insulation to be placed on the flange. For example, an insulation factor of 1.5 means that the weight of insulation to be added to the flange weight will be equivalent to the weight of one foot of insulation on the adjacent pipe times a 1.5 factor when using the ENG input units.

If the User wants to enter a flange length or a flange weight or an insulation factor that is different than those selected by TRIFLEX from the TRIFLEX database, then the User should select User Specified on the Flange Type and then enter the desired values.

Flange on "From End" - If the User wishes to instruct TRIFLEX to have a flange on the beginning end or from end of the valve, then the User should place a check in the box immediately to the left of the label "Flange on From End".

Flange on "To End" - If the User wishes to instruct TRIFLEX to have a flange on the far end or to end of the valve, then the User should place a check in the box immediately to the left of the label "Flange on To End".

Immediately below the data groups entitled "Valve Data" and "Flange Data", the User will find a data group entitled "Delta Dimension Coded To". If the User has selected a Flanged Valve in the Valve Type, then the Flanged Valve data fields on the right of this data group will be active and the Welded Valve data fields on the left of this data group will be inactive. If the User has selected a Welded Valve in the Valve Type, then the Welded Valve data fields on the left of this data group will be active and the Flanged Valve data fields on the right of this data group will be inactive. The fields in which data can be entered in this data group will be either for Welded Valve or Flanged Valve exclusively as further defined below:

Welded Valve

Far End Weld Point – If the User wishes to locate the Node Point at the Far End Weld Point of the valve, then the User should click on the radio button in front of the label "Far End Weld Point". When the User selects this modeling option, the entire valve length precedes the Node Point. When the User selects a welded valve, this node point location is the default.

Mid Point of the Valve – If the User wishes to locate the Node Point at the Mid Point of the valve, then the User should click on the radio button in front of the label "Mid Point of the Valve". When the User selects this modeling option, one half of the valve length precedes the Node Point and one half of the valve follows the node point. In such case, the User must not specify a delta dimension to the next Node Point of less than one half of the valve length

Near End Weld Point – If the User wishes to locate the Node Point at the Near End Weld Point of the valve, then the User should click on the radio button in front of the label "Near End Weld Point". When the User selects this modeling option, the entire length of the valve follows the Node Point. In such case, the user must not specify a delta dimension to the next Node Point of less than the valve length

Flanged Valve

Far End Weld Point – If the User wishes to locate the Node Point at the Weld Point of the flange on the far end of the valve, then the User should click on the radio button in front of the label "Far End Weld Point". When the User selects this modeling option, the entire valve length precedes the Node Point. When the User selects a flanged valve, this node point location is the default.

Far End Flange Face – If the User wishes to locate the Node Point at the flange face on the far end of the valve, then the User should click on the radio button in front of the label "Far End Flange Face". When the User selects this modeling option, the entire valve length precedes the Node Point and the far end flange follows the node point.

Mid Point of the Valve – If the User wishes to locate the Node Point at the Mid Point of the valve, then the User should click on the radio button in front of the label "Mid Point of the Valve". When the User selects this modeling option, one half of the valve length plus the near end flange length precedes the Node Point and one half of the valve length plus the far end flange length follows the node point. In such case, the User must not specify a delta dimension to the next Node Point of less than one half of the valve length plus the far end flange length.

Near End Flange Face – If the User wishes to locate the Node Point at the flange face on the near end of the valve, then the User should click on the radio button in front of the label "Near End Flange Face". When the User selects this modeling option, the entire length of the valve plus the far end flange length follows the Node Point. In such case, the User must not specify a delta dimension to the next Node Point of less than the valve length plus the far end flange length.

Immediately below the data group entitled "Delta Dimension Coded To - Welded Valve", the User will find additional data fields for miscellaneous data defined as follows:

SI Factor for "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the pipe section preceding the valve defined in this component, then the User should enter the desired numerical value in this field.

Weight Off - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label "Weight Off". The default is for weight to be considered.

Buoyancy - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label "Buoyancy Calculations". The density of the fluid surrounding the pipe should also be specified on the **Setup / Modeling Defaults dialog**. The default is for the effects of buoyancy not to be considered.

Immediately to the right of the miscellaneous data fields, the User will find a data group entitled "Pipe Size". In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

3.9 Coding Flange Data

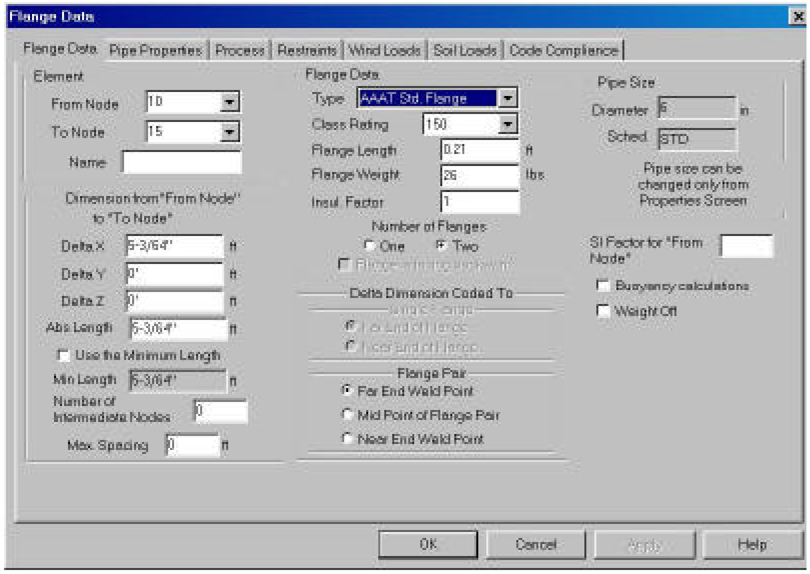


Figure 3.9.1 Coding Flange Data, Flange Data Tab

3.9.1 Coding Flange Data, Flange Data Tab

To enter a Flange component with or without a preceding Pipe, the User must click on the Flange Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Flange on the resulting pull down menu. Upon either of these sequences of actions, a Flange dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Flange dialog, a data group entitled "Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Dimension from "From Node" to "To Node". The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the point where the User is placing the Node) of the flange being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z– If the User is specifying the first component after an anchor, TRIFLEX will assume an "X" dimension equal to one foot, if English units are specified, or .35 Meters, if metric units are specified. If the node being defined follows another component other than an anchor, the assumed dimension will be along the vector line defined by the previous component. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or three axes, the User may enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will automatically calculate the Delta X, Delta Y and Delta Z dimensions and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of the flange(s) and the preceding component, then the User should place a check in the box immediately to the left of the label "Use the Minimum Length". TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the User desires to place a flange or flange pair fitting make-up with the previous component and it eliminates the need for the User to perform manual math calculations.

Minimum Length – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

Number of Intermediate Nodes – If the User enters a number in this field, TRIFLEX will break the pipe preceding the flange into one more segment that the number of intermediate nodes specified in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the to node and the from end of the flange. TRIFLEX will then break the pipe into three (3) segments.

Maximum Spacing – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe components generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

To the immediate right of the "Element" data group, the User will find a data group entitled "Flange Data". The fields in which data can be entered in this data group are defined below:

Type – The User must select a Flange Type from the drop down combo list in this field. The default flange type is the AAAT Standard Flange. From our staff's past experience, this flange is an average flange. The Type of flange, along with the Rating, allows TRIFLEX to search through the flange database to find the desired flange. The current TRIFLEX Flange database consists of five flange types. The flange types available in TRIFLEX are: AAAT Std. Flange, Blind Flange, Lap Joint Flange, Slip On Flange & Weld Neck Flange

Class Rating - The User must select a class rating from the drop down combo list in this field. The available class ratings are 150, 300, 400, 600, 900 and 1500. Given the flange type and the class rating, TRIFLEX will look up the appropriate corresponding weight, length, and insulation factor to be used in the calculations.

Flange Length – The data shown in this field is looked up from the data base by TRIFLEX or can be entered (over-typed) by the User, if desired.

Flange Weight – The data shown in this field is looked up from the data base by TRIFLEX or can be entered (over-typed) by the User, if desired.

Insulation Factor (ft, mm, m, mm) - if the User has specified the Type and Class Rating, the program will extract the insulation factor for the specified flange from the database. The user can input an insulation factor by overriding the value extracted from the database. The insulation factor is a value that when multiplied times the insulation weight per unit length will result in the weight of insulation to be placed on the flange. For example, an insulation factor of 1.5 means that the weight of insulation to be added to the flange weight will be equivalent to the weight of one foot of insulation on the adjacent pipe times a 1.5 factor when using the ENG input units.

If the User wants to enter a flange length or a flange weight or an insulation factor that is different than those selected by TRIFLEX from the TRIFLEX database, then the User should select User Specified on the Flange Type and then enter the desired values.

Number of Flanges - The User can specify one flange or two flanges by clicking on the radio button in front of one or two. TRIFLEX defaults to two.

When the User selects One Flange, the User will also then be given the opportunity to define the orientation of the flange face. TRIFLEX defaults to a flange facing forward or in the direction that the User is coding. For the User to instruct TRIFLEX to orient the flange face in the From direction, the User must place a check mark in the check box just to the left of the field entitled "Flange is facing backward".

Immediately below the data group entitled "Flange Data", the User will find a data group entitled "Delta Dimension Coded To". If the User has selected One Flange in the Number of Flanges radio buttons, then the Single Flange data fields at the top of this data group will be active and the Flange Pair data fields at the bottom of this data group will be inactive. If the User has selected Two Flanges in the Number of Flanges radio buttons, then the Flange Pair data fields at the bottom of this data group will be active and the Single Flange data fields at the top of this data group will be inactive. The fields in which data can be entered in these data groups will be either for Single Flange or Flange Pair exclusively as further defined below:

Single Flange

Far End of Flange – If the User wishes to locate the Node Point at the Far End of the flange, then the User should click on the radio button in front of the label "Far End of Flange". When the User selects this modeling option, the entire flange length precedes the Node Point. When the User selects a single flange, this node point location is the default.

Near End of Flange – If the User wishes to locate the Node Point at the Near End of the flange, then the User should click on the radio button in front of the label "Near End of Flange". When the User selects this modeling option, the entire length of the flange follows the Node Point. In such case, the User must not specify a delta dimension to the next Node Point of less than the flange length

Flange Pair

Far End Weld Point – If the User wishes to locate the Node Point at the Weld Point of the flange on the far end of the flange pair, then the User should click on the radio button in front of the label "Far End Weld Point". When the User selects this modeling option, the entire flange pair length precedes the Node Point. When the User selects a flange pair, this node point location is the default.

Mid Point of Flange Pair – If the User wishes to locate the Node Point at the Mid Point of the flange pair, then the User should click on the radio button in front of the label “Mid Point of the Flange Pair”. When the User selects this modeling option, the length of one flange precedes the Node Point and the length of one flange follows the node point. In such case, the User must not specify a delta dimension to the next Node Point of less than one flange length.

Near End Weld Point – If the User wishes to locate the Node Point at the Weld Point of the flange on the near end of the flange pair, then the User should click on the radio button in front of the label “Near End Weld Point”. When the User selects this modeling option, the entire length of the flange pair follows the Node Point. In such case, the User must not specify a delta dimension to the next Node Point of less than the length of the flange pair.

Immediately to the right of the “Flange Data” data fields, the User will find a data group entitled “Pipe Size”. In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

Immediately below the data group entitled “Pipe Size”, the User will find additional data fields for miscellaneous data defined as follows:

SI Factor for “From Node” – If the User wishes to specify a numerical stress intensification factor on the beginning of the pipe section preceding the flange defined in this component, then the User should enter the desired numerical value in this field.

Buoyancy - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label “Buoyancy Calculations”. The density of the fluid surrounding the pipe should also be specified on the **Setup / Modeling Defaults dialog**. The default is for the effects of buoyancy not to be considered.

Weight Off - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label “Weight Off”. The default is for weight to be considered

3.10 Coding Reducer Data

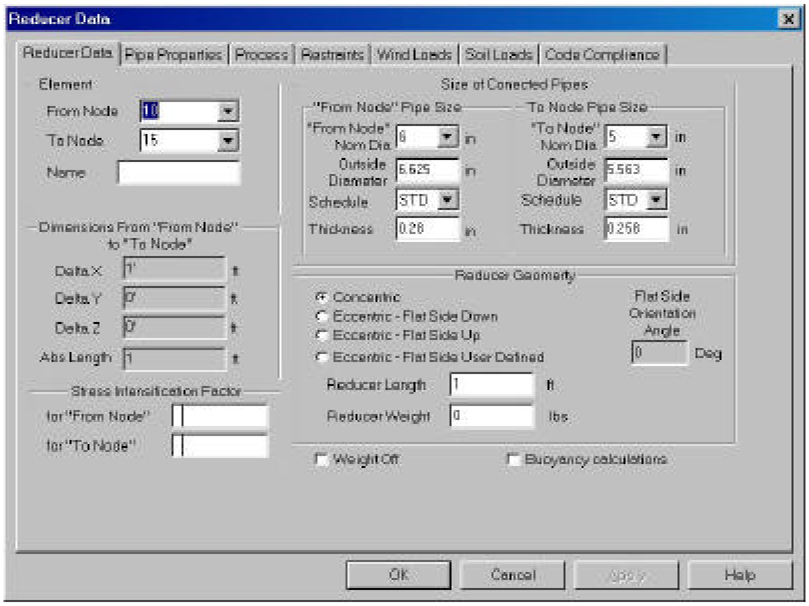


Figure 3.10.1 Coding Reducer Data, Reducer Data Tab

3.10.1 Coding Reducer Data, Reducer Data Tab

To enter a Reducer component, the User must click on the Reducer Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Reducer on the resulting pull down menu. Upon either of these sequences of actions, a Reducer dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Reducer dialog, a data group entitled "Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default

Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Dimension from "From Node" to "To Node". The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the end point) of the reducer being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z – The delta dimensions shown in the spaces provided are calculated by TRIFLEX based upon the vector direction defined in the previous component and the reducer length entered by the User. TRIFLEX will assume a dimension equal to one foot if English units are specified or .35 Meters if metric units are specified. The vector direction will be the same as the previously entered component. If the assumed length is incorrect, the User may enter the desired length in the Reducer Length field. Entering the reducer length may only change the delta dimensions.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. Entering the reducer length may only change the Absolute Length.

Immediately below the data group entitled Dimension from "From Node" to "To Node", the User will find a data group entitled "Stress Intensification Factor". The fields in which data can be entered in this data group are further defined below:

For "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the reducer component, then the User should enter the desired numerical value in this field.

For "To Node" - If the User wishes to specify a numerical stress intensification factor on the end of the reducer component, then the User should enter the desired numerical value in this field.

Immediately to the right of the Element data group, the User will find a data group entitled "Size of Connected Pipes". In this data group, the User can see the pipe diameter and schedule for the From Node, but should not change it in this data group. In this data group, the User can see and enter the desired pipe diameter and schedule for the To Node. The pipe size data for the To Node is then automatically transferred by TRIFLEX to the Pipe Properties dialog.

From Node Pipe Size

From Node Nom. Dia. – In this field, the Pipe Diameter specified for the from end of this component is displayed.

Outside Diameter - In this field, the Outside Diameter specified for the From end of this component is displayed.

Schedule - In this field, the Schedule specified for the from end of this component is displayed.

Thickness - In this field, the Thickness specified for the from end of this component is displayed.

To Node Pipe Size

To Node Nom. Dia. – In this field, the Pipe Diameter for the to end of the reducer is to be entered by the User. It must be a different diameter than the diameter specified for the From Node.

Outside Diameter - In this field, the Outside Diameter for the to end of the reducer is displayed based upon the Nominal Diameter entered by the User.

Schedule - In this field, the Schedule for the to end of the reducer is to be entered by the User. The default value for this field will be "Standard". The User may select the schedule from the drop down combo list in this field or enter the schedule, as desired. If the pipe wall thickness is not represented by a schedule, then the User can select "Custom" from the drop down combo list and specify the desired wall thickness in the following field.

Thickness - In this field, the Thickness specified for the to end of the reducer is displayed. If desired, the User may enter a numerical value for the thickness in this field.

Immediately below the "Size of Connected Pipes" data group, the User will find a data group entitled "Reducer Geometry". In this data group, the User must tell TRIFLEX whether the reducer is concentric or eccentric and, if eccentric, what the orientation is.

Concentric – The concentric radio button is the default selection. By accepting the radio button "Concentric", the User instructs TRIFLEX to consider the reducer to be concentric.

Eccentric – Flat Side Down – By clicking on this radio button, the User instructs TRIFLEX to consider the reducer to have the outside diameter of the From pipe and the To pipe on the same elevation on the bottom side. In so doing, TRIFLEX will automatically calculate the offset in the centerline of the to pipe and incorporate it into the piping model.

Eccentric – Flat Side Up – By clicking on this radio button, the User tells TRIFLEX to consider the reducer to have the outside diameter of the from pipe and the to pipe on the same elevation on the topside. In so doing, TRIFLEX will automatically calculate the offset in the centerline of the to pipe and incorporate it into the piping model.

Eccentric – Flat Side User Defined – By clicking on this radio button, the User tells TRIFLEX to consider the reducer to have the outside diameter of the from pipe and the to pipe on the same plane on a user specified orientation. When the User selects this radio button, the data field labeled "Flat Side Orientation Angle" will be made active to enable the User to enter the proper orientation angle. In so doing, TRIFLEX will automatically calculate the offset in the centerline of the to pipe and incorporate it into the piping model.

For lines running along the vertical axis, TRIFLEX will consider the flat side orientation angle equal to zero when aligned with the +X axis. For lines in the horizontal plane, TRIFLEX will consider the flat side orientation angle equal to zero when aligned with the +Y axis.

Reducer Length – In this field, the User must specify the desired length of the reducer, if other than the TRIFLEX default length. TRIFLEX will default to a length of one foot, if English units are specified, or .35 Meters, if metric units are specified.

Reducer Weight - In this field, the User must specify the weight of the reducer.

Immediately below the "Reducer Geometry" data group, the User will find additional data fields for miscellaneous data defined as follows:

Weight Off - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label "Weight Off". The default is for weight to be considered.

Buoyancy - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label "Buoyancy Calculations". The density of the fluid surrounding the reducer should also be specified on the **Setup / Modeling Defaults dialog**. The default is for the effects of buoyancy not to be considered.

3.11 Coding Joint Data

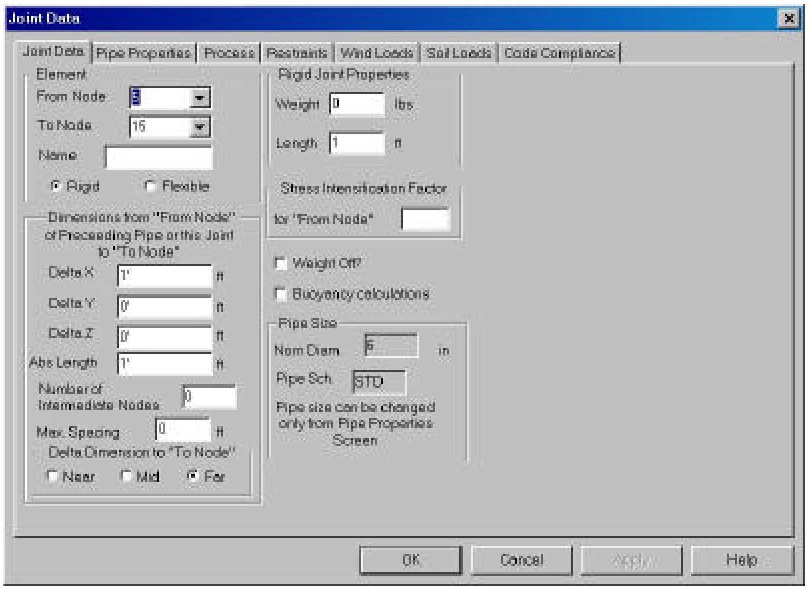


Figure3.11.1 Coding Joint Data, Joint Data Tab

3.11.1 Coding Joint Data, Joint Data Tab

To enter a Joint component with or without a preceding Pipe, the User must click on the Joint Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Joint on the resulting pull down menu. Upon either of these sequences of actions, a Joint dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Joint dialog, a data group entitled "Element" is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Rigid/Flexible – Immediately below the Name Field, the User is given the option to define whether the joint is rigid or flexible. A flexible joint is used to model structural members like angles, beams, channels, etc. A rigid joint is used to model anything that is completely rigid such as a casing for a piece of rotating equipment or a special valve, etc. The User is given two radio buttons to select from – rigid or flexible. TRIFLEX defaults to rigid. If the User wishes to select flexible, then the User must click on flexible.

Immediately below the "Element" data, the User will find a data group entitled "Dimension from "From Node" to "To Node". The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the point where the User is placing the Node) of the Joint being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z– If the User is specifying the first component after an anchor, TRIFLEX will assume an "X" dimension equal to one foot, if English units are specified, or .35 Meters, if metric units are specified. If the node being defined follows another component other than an anchor, the assumed dimension will be along the vector line defined by the previous component. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or three axes, the User may enter the desired data in the Delta X, Delta Y and/or Delta Z fields.

Abs Length – TRIFLEX will automatically calculate the absolute length and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will automatically calculate the Delta X, Delta Y and Delta Z dimensions and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of the Joint and the preceding component, then the User should place a check in the box immediately to the left of the label "Use the Minimum Length". TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the User desires to place a Joint fitting make-up with the previous component (especially if the previous component is a bend or a flange, valve or joint with the data point located at a point other than the end point) and it eliminates the need for the User to perform manual math calculations.

Minimum Length – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

Number of Intermediate Nodes – If the User enters a number in this field, TRIFLEX will break the pipe preceding the Joint, if any, into one more segment that the number of intermediate nodes specified in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the from node and the from end of the Joint. TRIFLEX will then break the pipe into three (3) segments.

Maximum Spacing – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe components generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

To the immediate right of the “Element” data group, the User will find a data group entitled “Rigid Joint Properties”. The fields in which data can be entered in this data group are defined below:

Weight – This field is provided to enable the User to enter a specific weight that will be applied by TRIFLEX at the centroid of the Joint Element, excluding the preceding pipe. If the User wishes the joint to be weightless, this field may be left blank or the User can enter a zero in this field. The default value is blank.

Length – This field is provided to enable the User to enter a specific length for the Joint Element itself. The User is not required to enter a joint length if the joint length is equal to the absolute length as entered in the delta dimensions. If the User wishes the joint to be preceded by a segment of pipe, then the User should enter the desired length in this field. The default value is blank.

Immediately below the data groups entitled “Rigid Joint Properties”, the User will find a data group entitled “Delta Dimension To “To Node””. The fields in which data can be entered in this data group are defined below:

Near – If the User wishes to locate the Node Point at the Near End of the Joint, then the User should click on the radio button in front of the label “Near”. When the User selects this modeling option, the entire length of the Joint follows the Node Point. In such case, the User must not specify a delta dimension to the next Node Point of less than the Joint length.

Mid – If the User wishes to locate the Node Point at the Mid Point of the Joint, then the User should click on the radio button in front of the label “Mid”. When the User selects this modeling option, one half of the Joint length precedes the Node Point and one half of the Joint follows the node point. In such case, the User must not specify a delta dimension to the next Node Point of less than one half of the Joint length

Far – If the User wishes to locate the Node Point at the Far End of the Joint, then the User should click on the radio button in front of the label "Far". When the User selects this modeling option, the entire Joint length precedes the Node Point. The default location for the node point location on a joint is the "Far" point or the end of the joint.

Immediately below the data group entitled "Delta Dimension To "To Node"" Flexible Joint Properties

For "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the pipe section preceding the Joint defined in this component, then the User should enter the desired numerical value in this field.

Immediately below the data group entitled "Stress Intensification Factor", the User will find additional data fields for miscellaneous data defined as follows:

Weight Off - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label "Weight Off". TRIFLEX will treat the Joint and the preceding Pipe coded on this component, if any, as weightless, if the User places a check in this check box. The default is for weight to be considered.

Buoyancy - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label "Buoyancy Calculations". The density of the fluid surrounding the pipe should also be specified on the **Setup / Modeling Defaults dialog**. The effects of Buoyancy will only be applied to pipe members that precede joints and will not be applied to joints themselves. The default is for the effects of buoyancy not to be considered.

Immediately below the miscellaneous data fields, the User will find a data group entitled "Pipe Size". In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

When the User in the Element Data Group selects a Flexible Joint, the third column in the right portion of the data dialog is displayed in which the User is to enter data. The data group at the top of the column is entitled "Flexible Joint Properties". Data must be entered in this data group as follows:

Designation – The User can click on the drop down combo list in this field and then select the desired structural member from the list of available members. In the event that the desired member is not available, the User can select "User Specified" and enter the applicable geometric properties.

In the event that the User wishes to enter a library of frequently used structural shapes, all such entries are made through "Utilities" then "Databases" then "Structural Steel".

Moment of Inertia about "B" Axis – When the User has selected a structural member in the field labeled "Designation"; this field will automatically be filled by TRIFLEX with the appropriate property value. When the User has selected "User Specified" in the field labeled "Designation", the User is expected to enter the appropriate property value in this field.

Moment of Inertia about "C" Axis – When the User has selected a structural member in the field labeled "Designation", this field will automatically be filled by TRIFLEX with the appropriate property value. When the User has selected "User Specified" in the field labeled "Designation", the User is expected to enter the appropriate property value in this field.

Polar Moment of Inertia "J_o" – When the User has selected a structural member in the field labeled "Designation"; this field will automatically be filled by TRIFLEX with the appropriate property value. When the User has selected "User Specified" in the field labeled "Designation", the User is expected to enter the appropriate property value in this field.

Distance from centerline to Outer Surface on "B" Axis – When the User has selected a structural member in the field labeled "Designation", this field will automatically be filled by TRIFLEX with the appropriate property value. When the User has selected "User Specified" in the field labeled "Designation", the User is expected to enter the appropriate property value in this field.

Distance from centerline to Outer Surface on "C" Axis – When the User has selected a structural member in the field labeled "Designation", this field will automatically be filled by TRIFLEX with the appropriate property value. When the User has selected "User Specified" in the field labeled "Designation", the User is expected to enter the appropriate property value in this field.

Cross Sectional Area – When the User has selected a structural member in the field labeled "Designation", this field will automatically be filled by TRIFLEX with the appropriate property value. When the User has selected "User Specified" in the field labeled “Designation”, the User is expected to enter the appropriate property value in this field.

(Beam Scan missing)

Immediately below the data group entitled “Flexible Joint Properties”, the User will find a data group entitled “Flexible Joint”. The fields in which data must be entered in this data group are defined below:

Angle between Joint C Axis and X, Y, Z Axes – In the following fields, the User must enter the “C” axis of the joint and each of the X, Y and Z axes. The “A” axis is defined as the axis of the joint. By entering these angles, the User defines to TRIFLEX how to orient the structural properties.

Shear Distribution Factor for Forces Parallel to “B” Axis – This field is provided to allow the User to specify the Shear Distribution Factor for forces acting parallel to the “B” axis of the flexible joint. This factor is multiplied times the force acting along the “B” axis. This factor may be considered as a stress intensification factor for the shearing force acting on the member attachment. It is generally taken to be the area of the member defined by the component divided by the area of the attachment, for example, a beam clip:

Beam Area / Clip Area = 41.2/11.2 = 3.71

Shear Distribution Factor for Forces Parallel to “C” Axis – This field is provided to allow the User to specify the Shear Distribution Factor for forces acting parallel to the “C” axis of the flexible joint. This factor is multiplied times the force acting along the “C” axis. This factor may be considered as a stress intensification factor for the shearing force acting on the member attachment.

3.12 Coding Expansion Joint

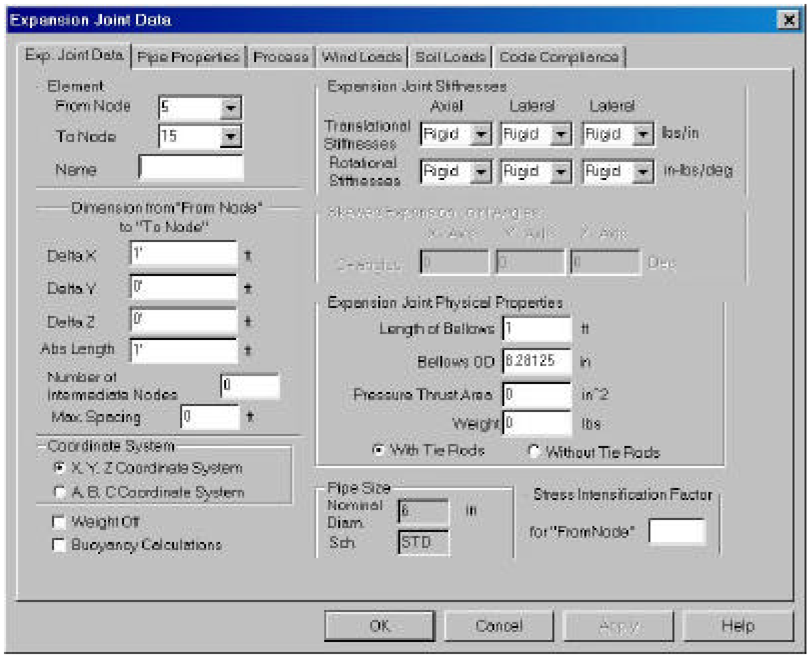


Figure 3.12.1 Coding Expansion Joint, Expansion Joint Tab

3.12.1 Coding Expansion Joint, Expansion Joint Tab

To enter an Expansion Joint component, the User must click on the Expansion Joint Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Expansion Joint on the resulting pull down menu. Upon either of these sequences of actions, an Expansion Joint dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Expansion Joint dialog, a data group entitled “Element” is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Dimension from "From Node" to "To Node". The dimension(s) entered in this data group define the vector from the previous node point to the To Node Point (the mid point) of the expansion joint being entered. The fields in which data can be entered in this data group are defined below:

Delta X, Delta Y and Delta Z– If the User is specifying the first component after an anchor, TRIFLEX will assume an "X" dimension equal to one foot if English units are specified or .35 Meters if metric units are specified. If the assumed length is incorrect or if the delta dimension should be along another axis or along two or more axes, the User may simply enter the desired data in the Delta X, Delta Y and/or Delta Z fields. The minimum dimension that can be entered is one half the length of the expansion joint entered by the User on this dialog.

Abs Length – TRIFLEX will automatically calculate the absolute length from the From Point to the mid point of the Expansion Joint and display it in this field. If the vector is in the same direction as the previously entered component, then the User may enter the absolute length desired and TRIFLEX will calculate the Delta X, Delta Y and Delta Z dimensions automatically and will display them in the appropriate fields.

Use the Minimum Length - If the User wishes to instruct TRIFLEX to use the Minimum Length as calculated by TRIFLEX based upon the length of any preceding component, and then the User should place a check in the box immediately to the left of the label "Use the Minimum Length". TRIFLEX will then replace the absolute length with the minimum required length. This is particularly useful when the piping component being modeled follows a Bend or a Valve, Flange or Joint with the data point specified at a point other than the end point.

Minimum Length – This field is a display field only. In this field, TRIFLEX displays the minimum length that must be provided between the previous Node Point and the To Node location being entered by the User. The Absolute Length dimension entered by the User must be equal to or greater than the minimum length computed by TRIFLEX.

<u>Number of Intermediate Nodes</u> – If the User enters a number in this field, TRIFLEX will break the pipe preceding the expansion joint into one more segment that the number of intermediate nodes specified by the User in this field. In other words, if the User enters a 2 in this field, TRIFLEX will place two (2) intermediate nodes between the to node and the beginning of the expansion joint – TRIFLEX will break the pipe into three (3) segments.

<u>Maximum Spacing</u> – The User may specify the maximum spacing between nodes in this field. If the lengths of the pipe components generated by TRIFLEX when the number of intermediate nodes is used by TRIFLEX to generate the intermediate nodes is longer than the length specified by the User in this field, then TRIFLEX will generate additional node points until the lengths between intermediate node points is less than the length specified by the User in this field.

Immediately below the data group entitled “Dimension from “From Node” to “To Node”, the User will find a data group entitled “Coordinate System”. The User must select one of the two coordinate systems listed below:

<u>X, Y, and Z Coordinate System</u> - If the User wishes to enter the expansion joint flexibilities along and about the X, Y, Z axis system, then the User accept the radio button being selected for this option.

<u>A, B, C Coordinate System</u> - If the User wishes to enter the expansion joint flexibilities along and about an axis system that is skewed with respect to the X, Y, Z-axis system, then the User should select this option by clicking on the radio button just to the left of the text. When this coordinate system is selected, the User will be expected to define the orientation angles as defined later in the discussion for this component.

Immediately below the data group entitled “Coordinate System”, the User will find additional data fields for miscellaneous data defined as follows:

<u>Weight Off</u> - If the User wishes to tell TRIFLEX to consider the component being entered as weightless, then the User should place a check in the box immediately to the left of the label “Weight Off”. The default is for weight to be considered.

<u>Buoyancy</u> - If the User wishes to tell TRIFLEX to consider the effects of buoyancy on this component, then the User should place a check in the box immediately to the left of the label “Buoyancy Calculations”. The density of the fluid surrounding the pipe should also be specified on the **Setup / Modeling Defaults dialog**. The default is for the effects of buoyancy not to be considered.

Immediately to the right of the data group entitled “Element”, the User will find a data group entitled “Expansion Joint Stiffness’”. The fields in which data can be entered in this data group are further defined below:

<u>Translational Stiffness’</u> – The User may enter the desired translational stiffness along the axial direction (along the axis of the expansion joint) and along the

Lateral directions (along each of the two perpendicular axes). The first of the two lateral translational stiffness' will be the one oriented most vertically and the second of the two lateral translational stiffness' will be the one oriented most horizontally. However, if the expansion joint is oriented along the Y-axis, then the three values must be the axial (along the Y axis), the lateral along the X-axis second, and the lateral along the Z-axis third.

To define the desired translational stiffness along each of the three axes, then the User can select Free or Rigid from the drop down combo list in each of the fields or enter a numerical value in any of these fields.

Rotational Stiffness' – The User may enter the desired rotational stiffness about the axial direction (torsion about the axis of the expansion joint) and about the Lateral directions (bending about the two perpendicular axes). The first of the two lateral rotational stiffness' will be the one about the axis oriented most vertically and the second of the two lateral rotational stiffness' will be the one about the axis oriented most horizontally. However, if the expansion joint is oriented along the Y-axis, then the three values must be the axial (about the Y axis), the lateral about the X-axis second, and the lateral about the Z-axis third.

To define the desired rotational stiffness about each of the three axes, then the User can select Free or Rigid from the drop down combo list in each of the fields or enter a numerical value in any of these fields.

Immediately below the data group entitled "Expansion Joint Stiffness'", the User will find a data group entitled "Skewed Expansion Joint Angles". If the User has selected the A, B, C Coordinate System, then TRIFLEX will activate this data group for the User to enter the C Axis angles. Given the C Axis and the axial direction, TRIFLEX has the required data to properly orient and apply the translational and rotational stiffness'. The fields in which data is to be entered in this data group are defined below:

C angle - X Axis, C angle - Y Axis and C angle - Z Axis – When the expansion joint is oriented along an axis that is skewed with respect to the X axis and/or the Y axis and/or the Z axis, the User must define the angle in degrees between the C Axis and the X axis, between the C Axis and the Y axis and between the C Axis and the Z axis. The angles should always be 180 degrees or less.

Immediately below the data group entitled "Skewed Expansion Joint Angles", the User will find a data group entitled "Expansion Joint Physical Properties". In the fields in this data group, the User is to enter the physical properties that describe the expansion joint. The fields in which data is to be entered in this data group are further defined below:

Length of Bellows – In this field, the User is to enter the physical length of the bellows. Since the User is to define the delta dimension to the middle of the expansion joint, the delta dimension must be greater than or equal to one half of the length of the bellows. The delta dimension for the component following the expansion joint must also allow for the second half of the bellows.

Bellows O.D. – In this field, the User is to enter the physical outside diameter of the corrugated section of the expansion point. This value is used to properly represent the expansion joint in the graphic model.

Pressure Thrust Area - In this field, the User is to enter the effective pressure thrust area. This value is commonly available from the manufacturer of the expansion joint being used. When tie rods do not restrain the expansion joint, the pressure thrust load will be exerted by the expansion joint on the pipe on both ends of the expansion joint. The pressure thrust load is determined by multiplying the pressure thrust area by the internal pressure.

With Tie Rods – When "With Tie Rods" is selected, TRIFLEX will make the axial spring constant as rigid and pressure thrust forces will not be applied to the pipe components on either side of the expansion joint. TRIFLEX defaults to an expansion joint with tie rods and, therefore, the radio button just to the left of the "With Tie Rods" label is selected. In the event that the User does not desire to have tie rods, the User should select the other option.

Without Tie Rods – To select this option, click on the radio button just to the left of the "Without Tie Rods" label. When "Without Tie Rods" is selected, TRIFLEX will use the axial spring constant entered by the User and pressure thrust forces will be generated and applied to the pipe components on either side of the expansion joint.

Immediately below the data group entitled "Expansion Joint Physical Properties", the User will find a data group entitled "Pipe Size". In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

Immediately below the data group entitled "Expansion Joint Physical Properties" and to the right of the data group entitled "Pipe Size", the User will find an additional data group entitled "Stress Intensification Factor". Data may be entered as follows:

SI Factor for "From Node" – If the User wishes to specify a numerical stress intensification factor on the beginning of the pipe section preceding the expansion joint defined in this component, then the User should enter the desired numerical value in this field.

3.13 Release Element

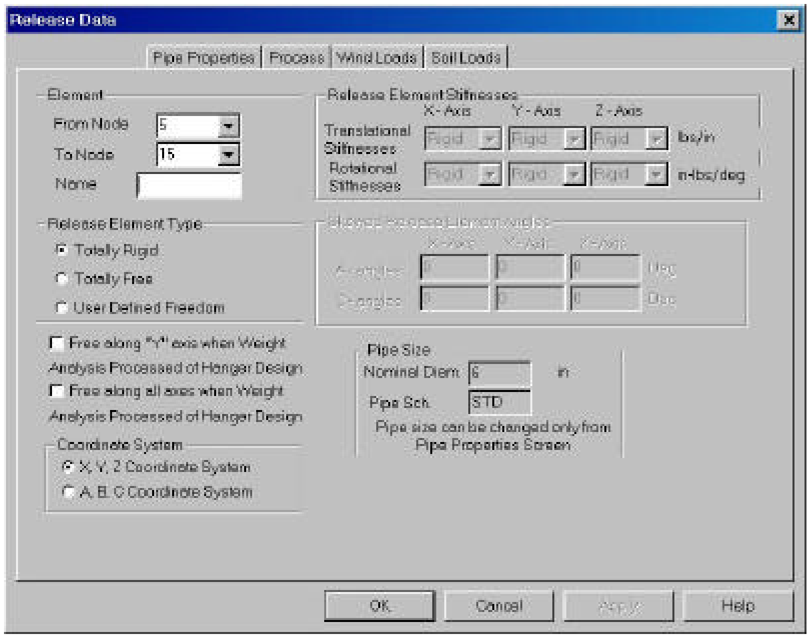


Figure 3.11.1 Coding Joint Data, Joint Data Table

3.13.1 Coding Release Element, Release Element Tab

To enter a Release Element component, the User must click on the Release Element Icon on the Component Toolbar on the left border of the dialog or click on Components on the main menu at the top of the dialog and then on Release Element on the resulting pull down menu. Upon either of these sequences of actions, a Release Element dialog with a series of related dialogs will be presented to the User. Enter the data as noted below:

The data is organized in related data groups on each and every dialog. In the upper left corner of the Release Data dialog, a data group entitled “Element” is available for User data entry. The fields in which data can be entered in this data group are defined below:

From Node – In this field, TRIFLEX will generate a Node Number equal to the To Node number for the previously entered component. If the node number generated by TRIFLEX is not the desired node number, then the User may select a node number from the drop down combo list in this field or enter a node number, as desired.

To Node - In this field, TRIFLEX will generate a Node Number based upon the From Node number and the node increment specified by the User in the Default

Settings. If the Node Number generated by TRIFLEX is not the desired node number, the User may select a node number from the drop down combo list in this field or enter any node number desired.

Name – In this field, the User may specify any name that will fit within the field. This name indicator will likely assist the User or other interested parties in identifying the significance of the node. Entry of the name in this field is optional.

Immediately below the "Element" data, the User will find a data group entitled "Release Element Type". The data group contains three options for the User to select from. Each option has a radio button just to the left of it.

Totally Rigid – This is the default selection. When the User selects this option, TRIFLEX will treat the release element as totally rigid for all analyses processed.

Totally Free - When the User selects this option, TRIFLEX will treat the release element as totally free for all analyses processed.

User Defined - When the User selects this option; TRIFLEX will treat the release element as having the stiffness' as defined by the User in a separate portion of the dialog.

Immediately below the data group entitled "Release Element Type", the User will find two hanger design options to select from. These design options enable a User to more accurately select spring hangers when the piping system model includes rotating equipment.

Free along "Y" axis when Weight Analysis Processed for Spring Hanger Design - If the User has elected to have TRIFLEX size and select spring hangers in this analysis and wishes to instruct TRIFLEX to consider this release element to be free along the "Y" only during the Weight Analysis, then the User should place a check in the box immediately to the left of the label "Free along "Y" axis when Weight Analysis Processed for Spring Hanger Design". The default for this option is that it is not selected. For a further discussion about the use of this option, see the Chapter 5 – Use of Restraints.

Free along "All" axes when Weight Analysis Processed for Spring Hanger Design - If the User has elected to have TRIFLEX size and select spring hangers in this analysis and wishes to instruct TRIFLEX to consider this release element to be free along all axes only during the Weight Analysis, then the User should place a check in the box immediately to the left of the label "Free along "All" axes when Weight Analysis Processed for Spring Hanger Design". The default for this option is that it is not selected. For a further discussion about the use of this option, see the Chapter 5 – Use of Restraints.

Immediately below the two hanger design options, the User will find a data group entitled “Coordinate System”. The User must select one of the two coordinate systems listed below:

X, Y, and Z Coordinate System - If the User wishes to enter the release element flexibilities along and about the X, Y, Z-axis system, then the User accepts the radio button being selected for this option.

A, B, C Coordinate System - If the User wishes to enter the release element flexibilities along and about an axis system that is skewed with respect to the X, Y, Z-axis system, then the User should select this option by clicking on the radio button just to the left of this text. When this coordinate system is selected, the User will be expected to define the orientation angles as defined later in the discussion for this component.

Immediately to the right of the data group entitled “Element”, the User will find a data group entitled “Release Element Stiffness’”. The fields in which data can be entered in this data group are further defined below:

Translational Stiffnesses – The User may enter the desired translational stiffness along the axial direction (along the axis of the release element) and along the Lateral directions (along each of the two perpendicular axes). The first of the two lateral translational stiffness’ will be the one oriented most vertically and the second of the two lateral translational stiffness’ will be the one oriented most horizontally. However, if the release element is oriented along the Y-axis, then the three values must be the axial (along the Y axis), the lateral along the X-axis second, and the lateral along the Z-axis third.

To define the desired translational stiffness along each of the three axes, then the User can select Free or Rigid from the drop down combo list in each of the fields or enter a numerical value in any of these fields.

Rotational Stiffness’ – The User may enter the desired rotational stiffness about the axial direction (torsion about the axis of the expansion joint) and about the Lateral directions (bending about the two perpendicular axes). The first of the two lateral rotational stiffness’ will be the one about the axis oriented most vertically and the second of the two lateral rotational stiffness’ will be the one about the axis oriented most horizontally. However, if the expansion joint is oriented along the Y-axis, then the three values must be the axial (about the Y axis), the lateral about the X-axis second, and the lateral about the Z-axis third.

To define the desired rotational stiffness about each of the three axes, then the User can select Free or Rigid from the drop down combo list in each of the fields or enter a numerical value in any of these fields.

Immediately below the data group entitled “Release Element Stiffness’”, the User will find a data group entitled “Skewed Release Element Angles”. If the User has selected the A, B, C Coordinate System, then TRIFLEX will activate this data group for the User to enter the A Axis and C Axis angles. Given the orientation of the A Axis and the C Axis, TRIFLEX has the required data to properly orient and apply the translational and rotational stiffness'. The fields in which data is to be entered in this data group are defined below:

A angle - X Axis, A angle - Y Axis and A angle - Z Axis – When the Release Element is skewed with respect to the X axis and/or the Y axis and/or the Z axis, the User must define the angle in degrees between the A Axis and the X axis, between the A Axis and the Y axis and between the A Axis and the Z axis. The angles should always be 180 degrees or less.

C angle - X Axis, C angle - Y Axis and C angle - Z Axis – When the Release Element is skewed with respect to the X axis and/or the Y axis and/or the Z axis, the User must define the angle in degrees between the C Axis and the X axis, between the C Axis and the Y axis and between the C Axis and the Z axis. The angles should always be 180 degrees or less.

Immediately below the data group entitled "Skewed Release Element Angles", the User will find a data group entitled "Pipe Size". In this data group, the User can see the pipe diameter and schedule as entered on a different dialog for this component. If the User wishes to change either the pipe diameter or the pipe schedule, the User must go to the Pipe Properties tab.

Pipe Diameter – In this field, the Pipe Diameter specified for this component is displayed.

Pipe Schedule - In this field, the Pipe Schedule specified for this component is displayed.

3.14 Executing a Static Analysis

To process a TRIFLEX®Windows analysis of the piping system you just entered, click on the **Green Arrow** icon in the **Main Menu** or from the **Setup** menu, select the **Basic** option as shown in Figure 3.14-1.

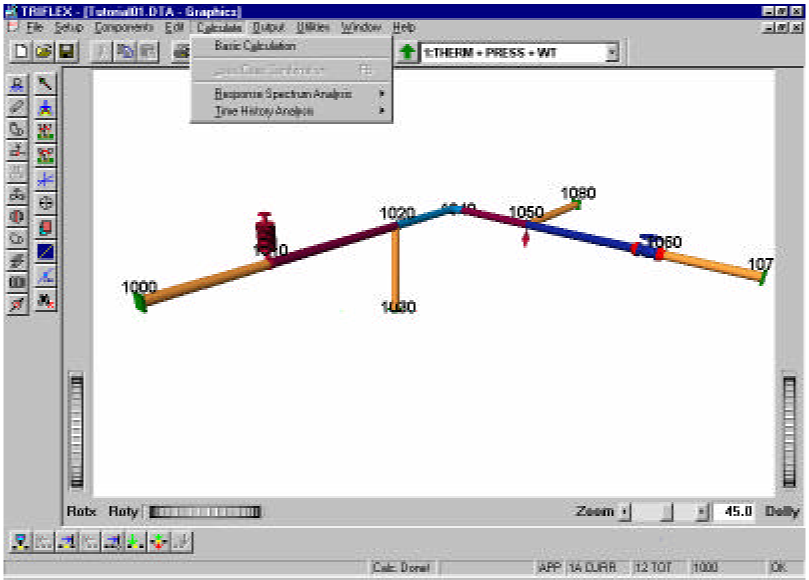


Figure 3.14.1 Main Screen, Calculate Pull-Down Menu

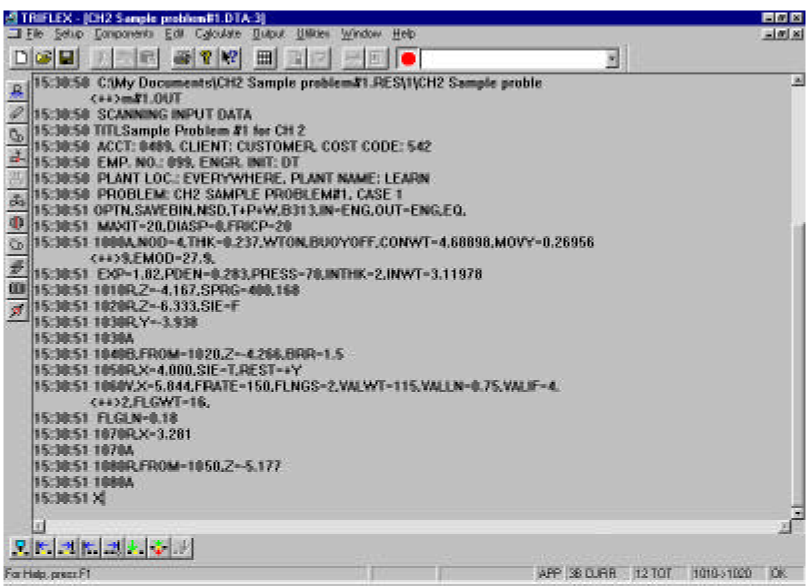


Figure 3.14.2 Main Screen, Calculation Ready/Stop Icon

Note: A case number must be filled in before TRIFLEX® Windows will perform the stress calculations.

Once TRIFLEX has been instructed to process the analysis, the program will begin executing the stress calculations. The status of the calculations will be displayed in the TRIFLEX®Windows screen.

While the calculation is in progress, the **Calculation Ready/Stop** Icon will be displayed as a red stop sign as shown in Figure 3.14-2. To stop the calculation process, click the **Calculation Ready/Stop** Icon and the calculations will be immediately aborted.

Upon completion of the calculation process, the **Calculation Ready/Stop** Icon will be returned to the green arrow ready state as shown in Figure 3.14-1.

3.15 View Run Output

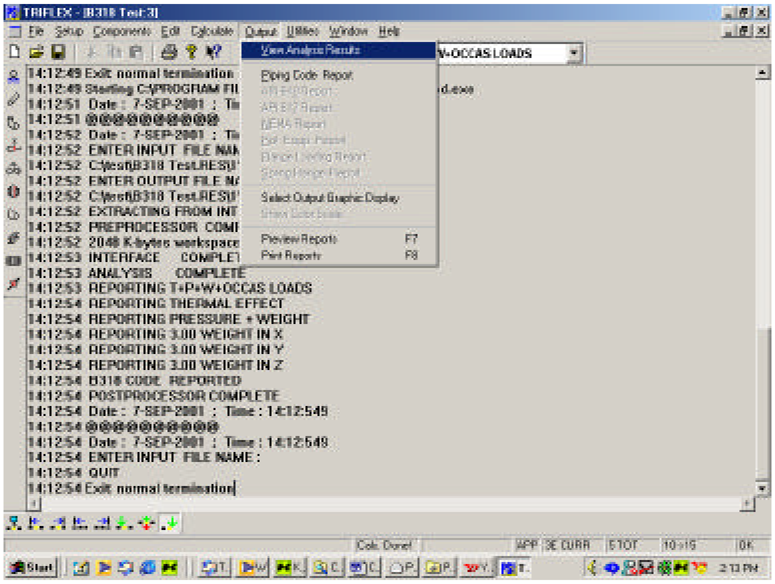


Figure 3.15-1 Output Pull-Down Menus

To view the results of the stress calculations in spreadsheet format, do the following: From the **Output Pull Down** menu, select **View Results.** See Figure 3.15-1 below for this menu. The TRIFLEX®Windows calculation results will be displayed as shown in Figure 3.15-2

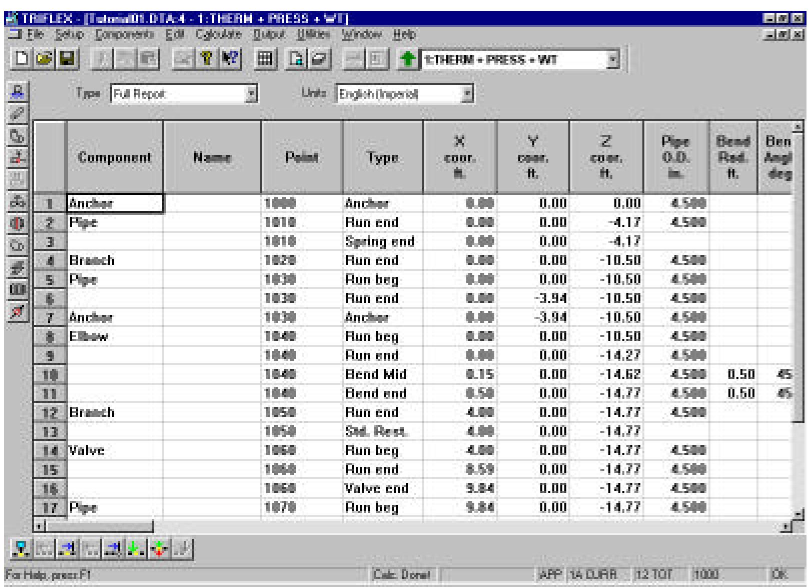


Figure 3.15-2 – Output Report, View results

To view Code Compliance Report

1. Select the **Load Case** that you wish to view using the **Load Case pull-down** menu as shown in Figure 3.15-2.

2. Select the report that you wish to view using the **Type Report Selector pull down** menu as shown in Figure 3.15-3.

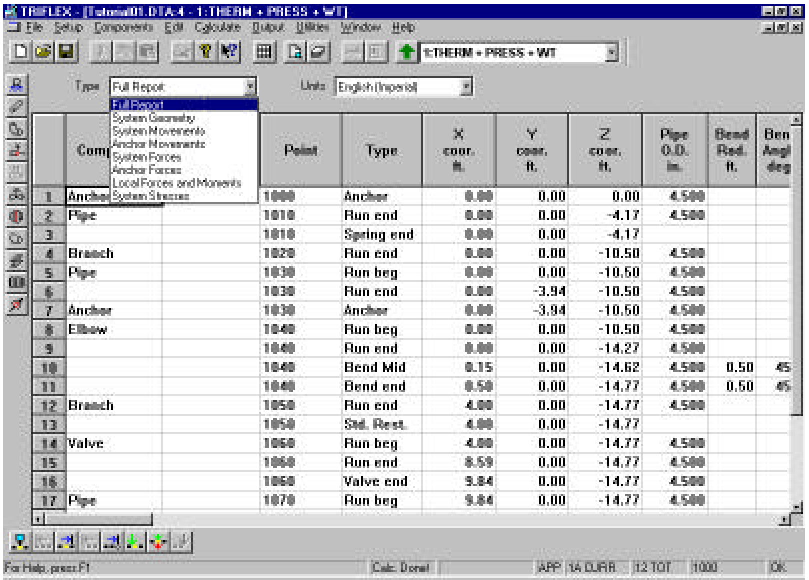


Figure 3.15-3 – Output Report, Type Report Selector

3. From the **Output Pull Down** menu, select **Piping Code Report** similar to that shown in Figure 3.15-1. The TRIFLEX®Windows calculation results will be displayed as shown in Figure 3.15-4.

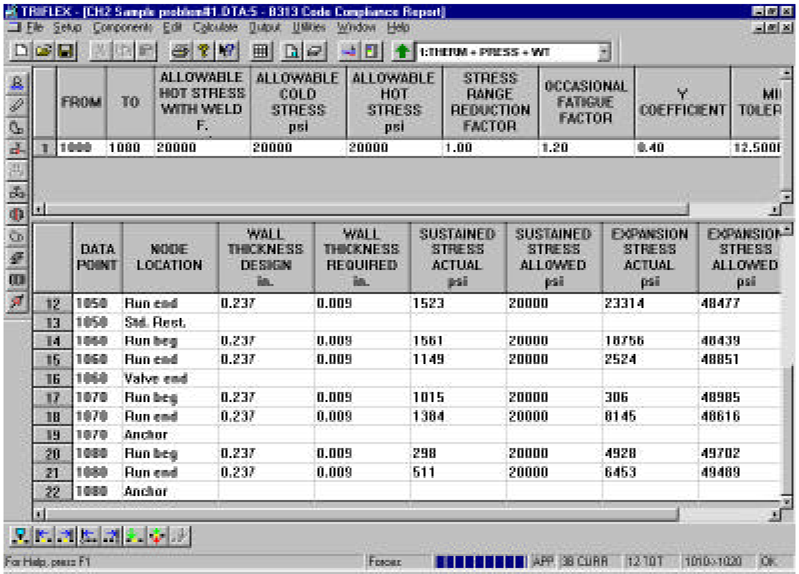


Figure 3.15-4 Output Code Compliance Report

To view the piping model output graphically,

4. Click on the **Output Display** icon in the **Main Menu** as shown in Figure 3.15-5 or, from the **Output Pull Down** menu, select **Output Graphic Display** similar to that shown in Figure 3.15-1. An Output display screen will appear in the middle of the screen.

5. In the **Output Display** screen, click on the **Display Pull Down** menu as shown in Figure 3.15-6 to select the calculated output data that you wish to view.

6. If you select deflections, rotations, forces or moments, you must then select the **Line of Action** that you wish. Under Line of Action, TRIFLEX will default to Resultant values unless you specify another category. Then click OK.

   If you select any of the stresses calculated by TRIFLEX, then you must select either **Absolute Value** or **Sign (+/-)** from the Stress Display group. Under the Stress Display group, TRIFLEX will default to Absolute values unless you specify Sign (+/-). Then click OK.

Note: If your piping model does not appear on the screen at this point, then press Control + Tab to toggle between all screens available describing the piping system. Stop when you see the piping model. Alternatively, you can click on the Spreadsheet Icon to toggle between the spreadsheet view and the piping model.

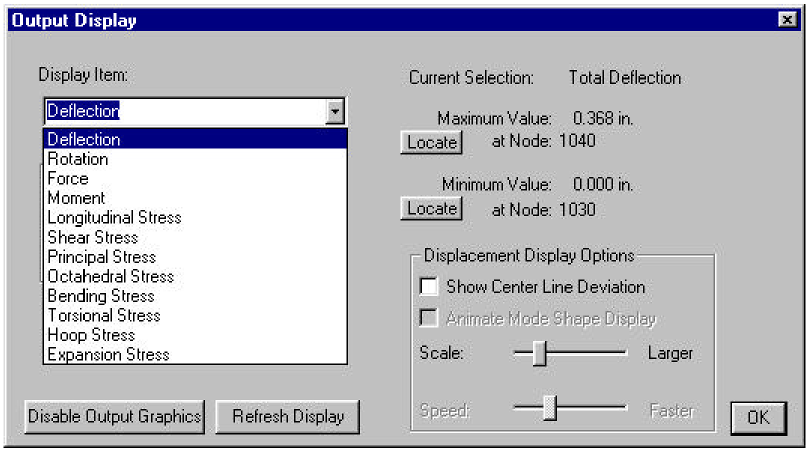


Figure 3.15-6 – Output Display Dialog

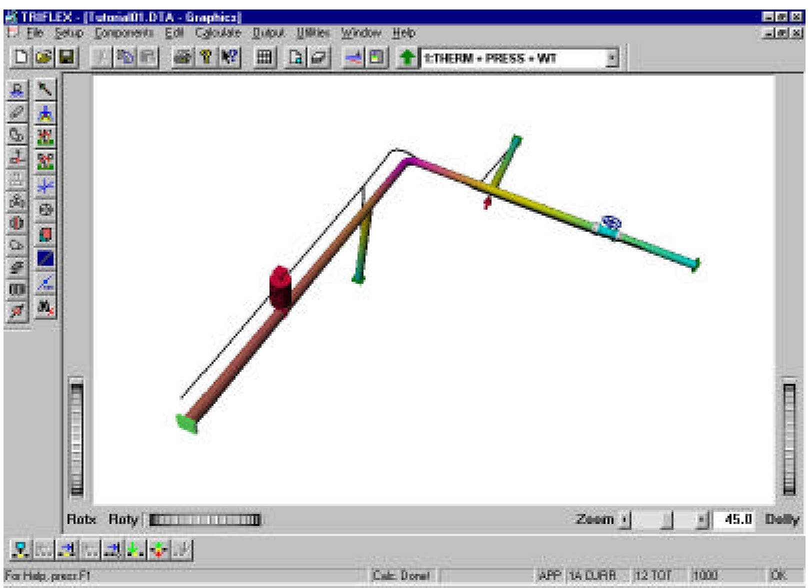


Figure 3.15-7 – Output Display Deformed Graphics

To view the piping model with a superimposed deformed shape,

7. In the **Output Display** screen shown in Figure 3.15-7, click on the **Display Pull Down** menu and select **Deflection.**

8. Then on the **Output Display** screen, click on the check box for **Show Center Line Deviation** and enter a number in the **Scale** field indicating the multiplier factor to be applied to the deflection shown on the model. Then click OK. A screen showing the deformed piping model will then appear as shown in Figure 3.15-7.

3.1.6 Printing

TRIFLEX®Windows has an extremely easy to use facility for printing output reports and screens.

1. From the **Output Pull Down** menu, select **Preview Report** or **Print Report** similar to that shown in Figure 3.16-1. The Report Print (Print Static / Dynamic Reports) screen will then appear.

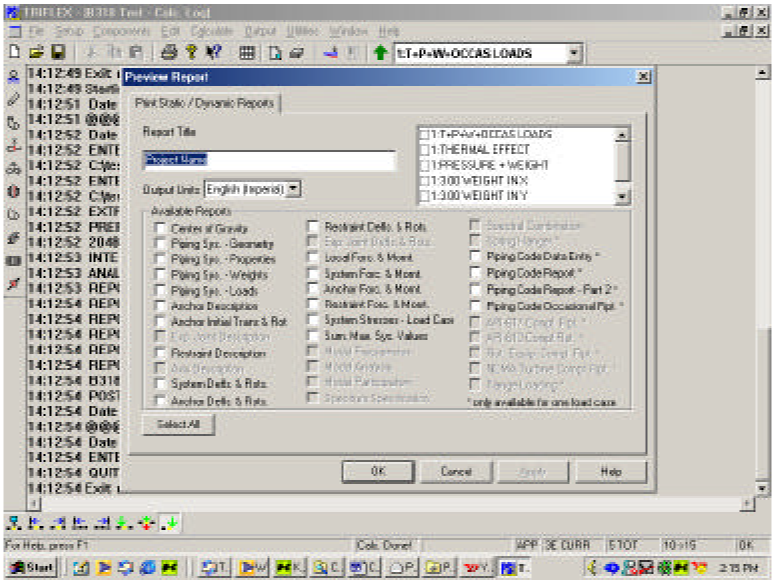


Figure 3.16-1 – Print Report

2. In **Print Report** or **Preview Report** screen, select the **Loading Case** and the reports from the **Available Report** group by placing a check in the box adjacent to each desired report as shown in Figure 3.16-1.

3. A **Preview Report** sample screen is shown in Figure 3.16-2

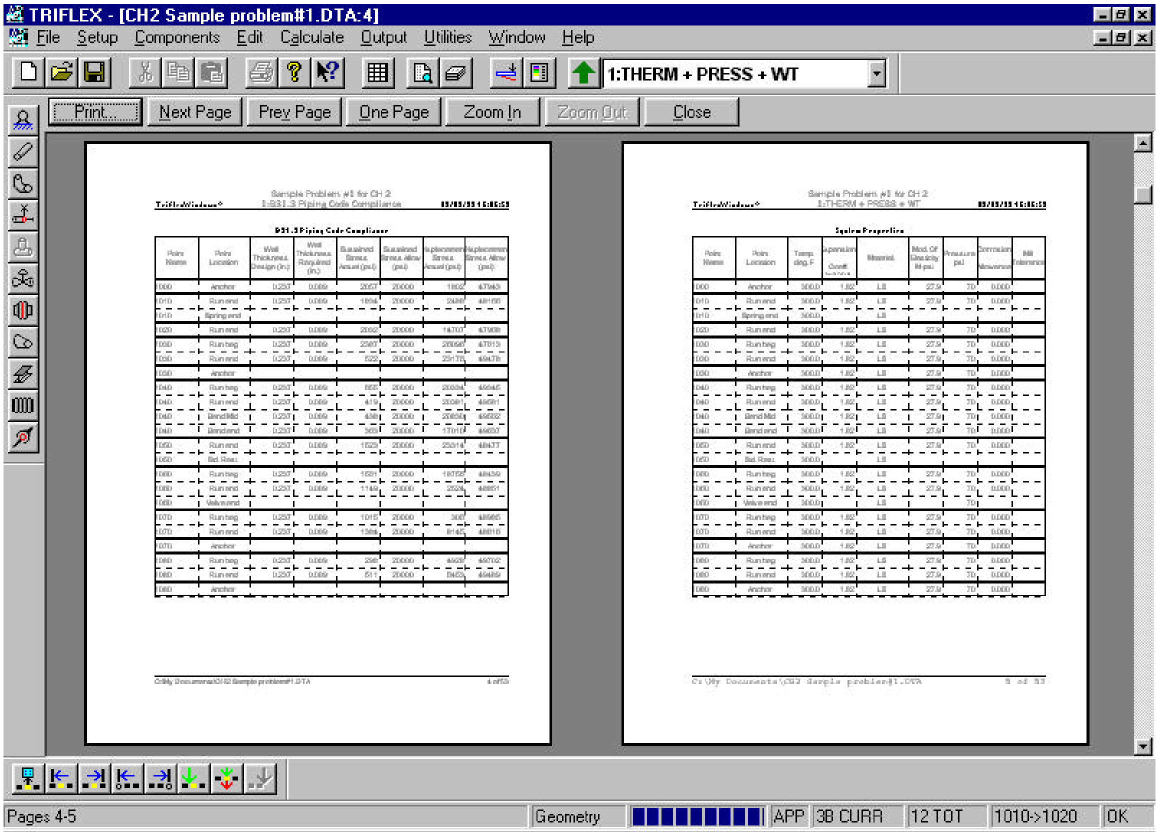


Figure 3.3.6-2 Report Print Menu

To exit TRIFLEX®Windows, click **EXIT** under **FILE**. Please remember before you exit TRIFLEX to save your model. To do so, from the **File Pull Down** menu, select **Save As** similar to the procedure used in most Windows programs.

APPENDIX A

TRIFLEXâ Windows Command and Shortcut Keys

(Graphics Mode)

COMMANDS	**SHORTCUTS**
• Help	F1 (not active currently)
• Worksheet Toggle	F4
• Start Calculation	F5
• Print Report Preview	F7
• Print Report	F8
• Move to End	END
• Edit Current Component	F9
• Move to First Component	HOME
• Insert	INS
• Move to Next Component	PGDN
• Move to Previous Component	PGUP
• Copy	CTRL + C
• Cut	CTRL + X
• Delete	DEL
• New	CTRL + N
• Open	CTRL + O
• Paste	CTRL + V
• Print	CTRL + P
• Save	CTRL + S
• Undo	CTRL + Z

• "Arrow" moves model	ARROW KEYS
• Bring up "Start"	CTRL + ESC
• Capture display to Clipboard	ALT + PRTSC
• Change pointer/manipulator	ESC
• Change pointer to manipulator	ALT + SHIFT
• Change to manipulator (temporary)	ALT
• Display all available windows	ALT + ESC
• Manipulator moves model	SHIFT
• Next (toggle between graphics & Spreadsheet input)	CTRL + F6
• Toggle through all available windows	ALT + TAB