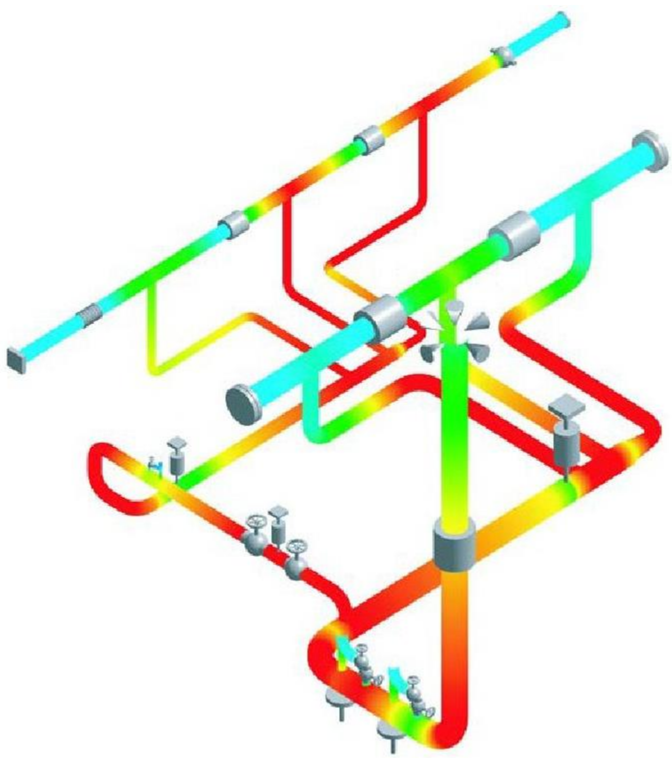
CAEPIPE™

User's Manual



From the CAdvantagE™ Library

SYSTEMS, INC.

The *FASTEST* Solutions for Piping Design and Analysis

Disclaimer

Please read the following carefully:

This software and this manual have been developed and checked for correctness and accuracy by SST Systems, Inc. However, no warranty, expressed or implied, is made by SST Systems, Inc. as to the accuracy and correctness of the manual or the functioning of the software and the accuracy, correctness and utilization of its calculations. Users must carry out all necessary tests to assure the proper functioning of the software and the applicability of its results. All information presented by the software is for review, interpretation, approval and application by a Registered Professional Engineer.

CAEPIPE and CAdvantagE are trademarks of SST Systems, Inc. All other product names mentioned in this document are trademarks or registered trademarks of their respective companies/holders.

For Info and Support on CAEPIPE, contact:

SST Systems, Inc.
1798 Technology Drive, Ste. 236
San Jose, CA 95110, USA
Tel: (408) 452-8111
Fax: (408) 452-8388

info@sstusa.com

support@sstusa.com

www.sstusa.com

Table of Contents

Table of Contents

Table of Contents

Table of Contents

This page is blank

Introduction

Thank you for licensing CAEPIPE (pronounced kay-pipe), the simple yet powerful software for solving a variety of piping design and stress analysis problems in numerous industries listed below.

Power(fossil & nuclear)	Oil & Gas production (onshore & offshore)
Refinery	Chemical & Petrochemical
Fertilizers	Pharmaceutical
Sugar & Food Processing	Paper & Pulp
Steel / Metal Process	Water & Waste Treatment
Aircraft and Aerospace	Building Services
Defense Industries	Ship Building

CAEPIPE performs linear and nonlinear static and linear dynamic analyses of piping systems by imposing various loads such as deadweight, thermal, seismic, wind, spectrum, time history or harmonic, and calculates displacements, forces, moments, stresses, support loads etc. Further, it checks whether the piping system is piping code and guideline compliant (ASME, B31, European, Swedish, API 610, etc.) and produces concise, formatted and easy to understand reports.

For rapid modeling, CAEPIPE offers you a friendly and productive user interface that rigorously adheres to Windows standards. Open up to four windows simultaneously to get feedback on different aspects of the model. Extensive graphical display capabilities allow you to zoom, pan, rotate the image and see the model from different viewpoints. CAEPIPE uses the industry standard OpenGL® to render 3D images realistically for easy visualization. As the model is input and modified, CAEPIPE updates the graphics simultaneously to provide visual feedback. It animates deflected shapes and mode shapes, and shows color-coded stress contours, among others.

A true powerhouse in its speed of operation, CAEPIPE uses advanced Windows programming techniques such as intelligent repainting, scroll box tracking, multithreading, memory-mapped files for faster data access, among others, to make your job easier and faster. Every effort is made to keep the program and data file sizes small (e.g., program size is ~2 MB! And a 665-element piping model is 85 KB!).

Many thoughtful and useful details in the program allow you to work more productively. For example, you can annotate your model with copious comments for enhanced documentation, or duplicate repetitive input with one hotkey combination or rotate sections of the model with one operation. No unnecessary buttons clutter the toolbar nor are you forced to use a mouse unnecessarily. The many thoughtful keyboard shortcuts, too, add to your productivity.

Overall, CAEPIPE stands peerless among the tools available today for piping design and stress analysis. We invite you to explore the software so that you can make full use of its capabilities. Our friendly and knowledgeable support engineers are available to assist you. Should you need to reach them, please email: support@sstusa.com.

Two sections make up this manual:

1. Explanation of menus from the different CAEPIPE windows,
2. Appendices with related information.

The manual ends with an index.

The full version of CAEPIPE is supplied with either of two protection schemes viz Hardware Protected Key or a Software Protected Key. For both the Hardware Protected Key and Software Protected Key versions of CAEPIPE, installing requires two steps:

1. For the Hardware Protected Key, install the hardware key driver or for the Software Protected Key install the SST License Manager (SSTLM).
2. Install the CAEPIPE software.

Follow the Installation Instructions provided by SST Systems, Inc. via email as the CAEPIPE setup files for the Hardware Protected Key and Software Protected Key are hard coded to the specific serial number assigned to your company.

CAEPIPE with Hardware Protected Key

The Server machine hosts the CAEPIPE license via USB Hardware Protected Key and can also have CAEPIPE software installed. This machine can be any computer connected to a network.

1. **Download** the "Sentinel Installer (exe)" or "Sentinel Installer (zip)" file using the link https://www.sstusa.com/support-mes.php to the server machine where the CAEPIPE USB Hardware Protected Key will be residing.
2. **Double Click** on the executable and follow all instructions to complete the installation of the hardware key driver.
3. Insert your hardware key into the USB port.
4. On the server machine, configure your firewall to allow all **In-Bound** and **Out-Bound** network traffic on **UDP Port 6001**. Please visit our **SuperProNet Firewall Configuration** page for more details.
5. **Download** the CAEPIPE setup file (.exe or .zip) from PEXit, using the installation instructions provided by SST Systems, Inc. via email.
6. Unzip the downloaded file. Double Click on "Setup.exe" to open the installation menu.
7. **Identify the client machine(s) and Install CAEPIPE** by following all instructions to complete the installation.
8. Create **NSP_HOST environment variable** to connect server machine hosting CAEPIPE license via Hardware Protected Key to the client machine. Please visit our **NSP_HOST Configuration** page for detailed instructions on how to do this.

CAEPIPE with Software Protected Key [SST License Manager (SSTLM)]

1. **Download** the SST License Manager (SSTLM) from PEXit, using the installation instructions provided by SST Systems, Inc. via email.
2. Unzip the downloaded file. Double Click on "Setup.exe" to open the installation menu.
3. Follow all the instructions to complete the installation of SST License Manager.

4. For a Network configuration, on the server machine where SSTLM is installed, configure your firewall to allow all **In-Bound** and **Out-Bound** network traffic on **TCP Port 12000**. This also needs to be done on any client machines where CAEPIPE will be installed.

5. **Download** the CAEPIPE setup file (.exe or .zip) from PEXit, using the installation instructions provided by SST Systems, Inc. via email.

6. **Identify the client machine(s) and Install CAEPIPE** by following all instructions to complete the installation.

7. Create **SSTLM environment variable** to connect server machine hosting CAEPIPE license to client machine(s). Please visit our configuration page http://www.sstusa.com/sstlm-variable.php for details on setting this variable.

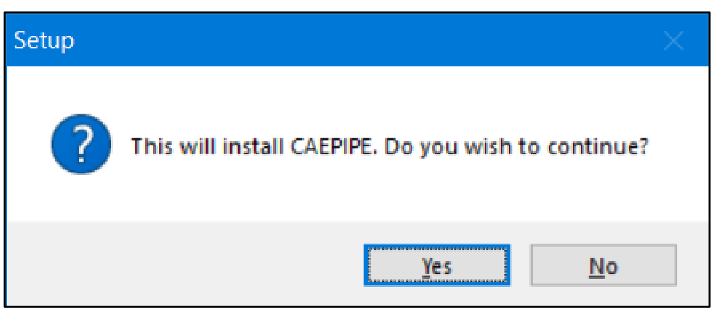
8. Create **SKIPIP environment variable**, if needed and set the value to "YES". This is typically used on laptops where CAEPIPE might be used in a different network environment with a different TCP/IP address (i.e. at the office/at home/on a project site).This should also be used if your machine (where SSTLM is installed) is configured to obtain the IP Address automatically from a DHCP server, then user should disable the feature of checking the IP Address by defining an environmental variable with name "SKIPIP" and setting its value as "YES". Please visit our configuration page http://www.sstusa.com/support-install2.php for details on setting this variable.

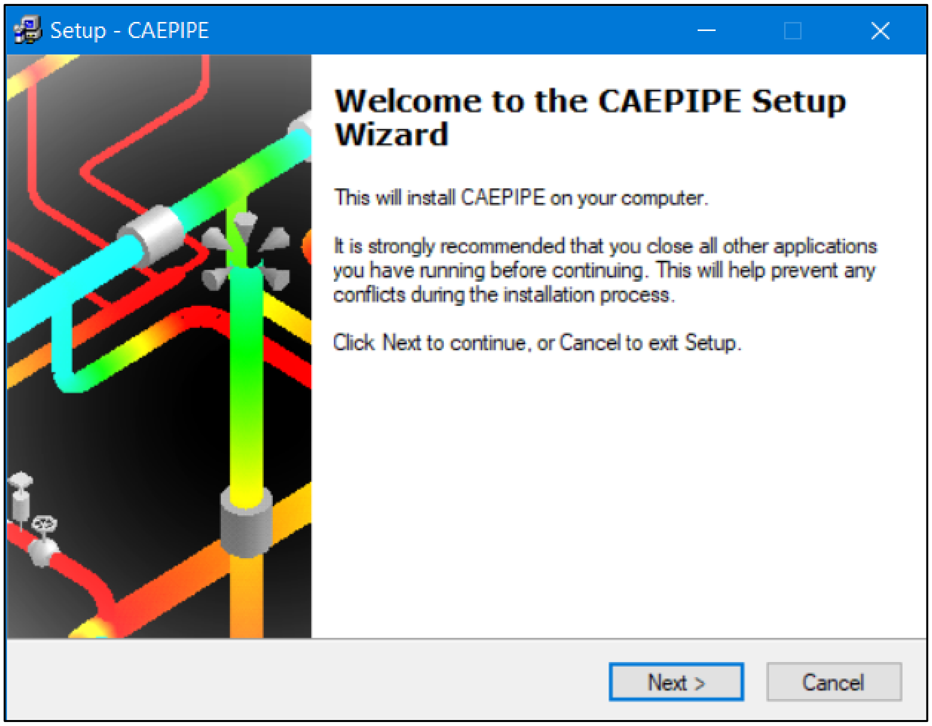
For more details on installing the driver for the Hardware Protected Key or installing the SST License Manager (SSTLM) for the Software Protected Key scheme, please refer to the documentations that are available at the link http://www.sstusa.com/support-install.php.

Installing CAEPIPE

To install CAEPIPE, double click on setup executable.

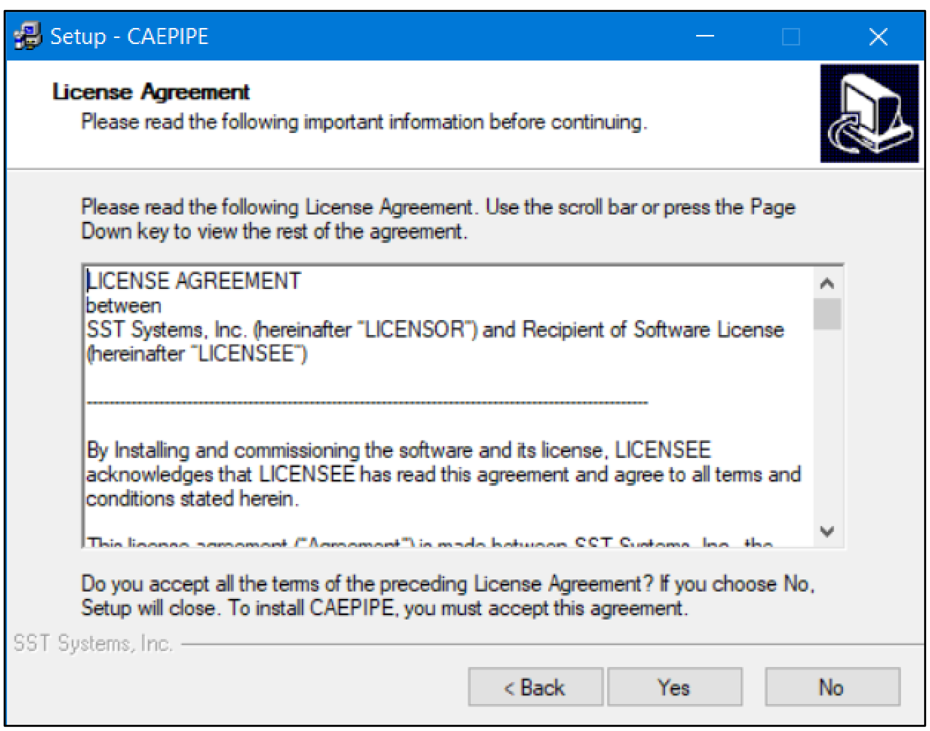
You will now see the following interactive screens one after another in which you will have to answer the usual installation questions. Respond appropriately.





Click Next to continue.

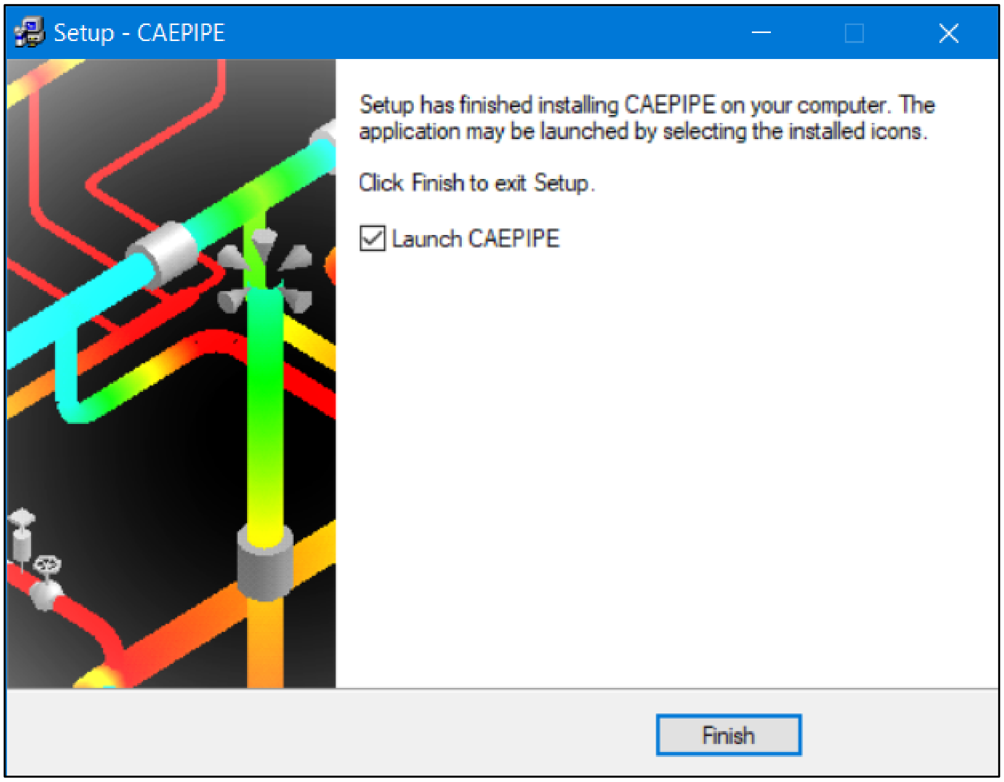
You must accept the terms of the License agreement by clicking on Yes before proceeding.



A copy of the license agreement (PDF file) is available on the distribution.

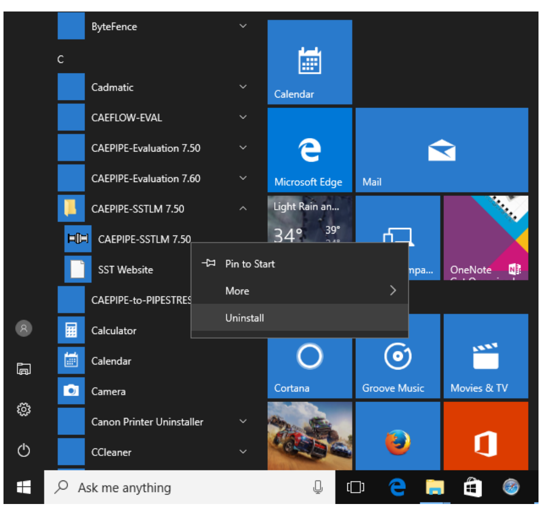
We recommend installing the software to the default folder shown during installation. If you have previous installations of CAEPIPE, it will be helpful to name the Start menu folder differently for the new version (e.g., 1050 for version 10.50).

Once you have picked your choices, you can now begin to install by clicking on "Install" button or use the "Back" button to return to previous screens to change your choices.



After installation, you can execute CAEPIPE by double clicking on the Desktop icon or by selecting CAEPIPE from the newly created CAEPIPE folder on the Start menu.

To uninstall CAEPIPE from your Windows 8 and Windows 10 computer, select the option "Uninstall" through Start > CAEPIPE > Mouse right click on CAEPIPE executable as shown below.



To uninstall CAEPIPE from your Windows versions earlier than Windows 8, select Uninstall CAEPIPE from the CAEPIPE folder on the Start menu.

Menus

This section explains the commands under the different menus in the four independent CAEPIPE windows: Layout, Graphics, List and Results. Each window contains its own menu and toolbar.

Many commands have keyboard shortcuts, shown next to the Command in the drop-down menu and also in online Help.

The menu items (commands) are shown as

.Command.

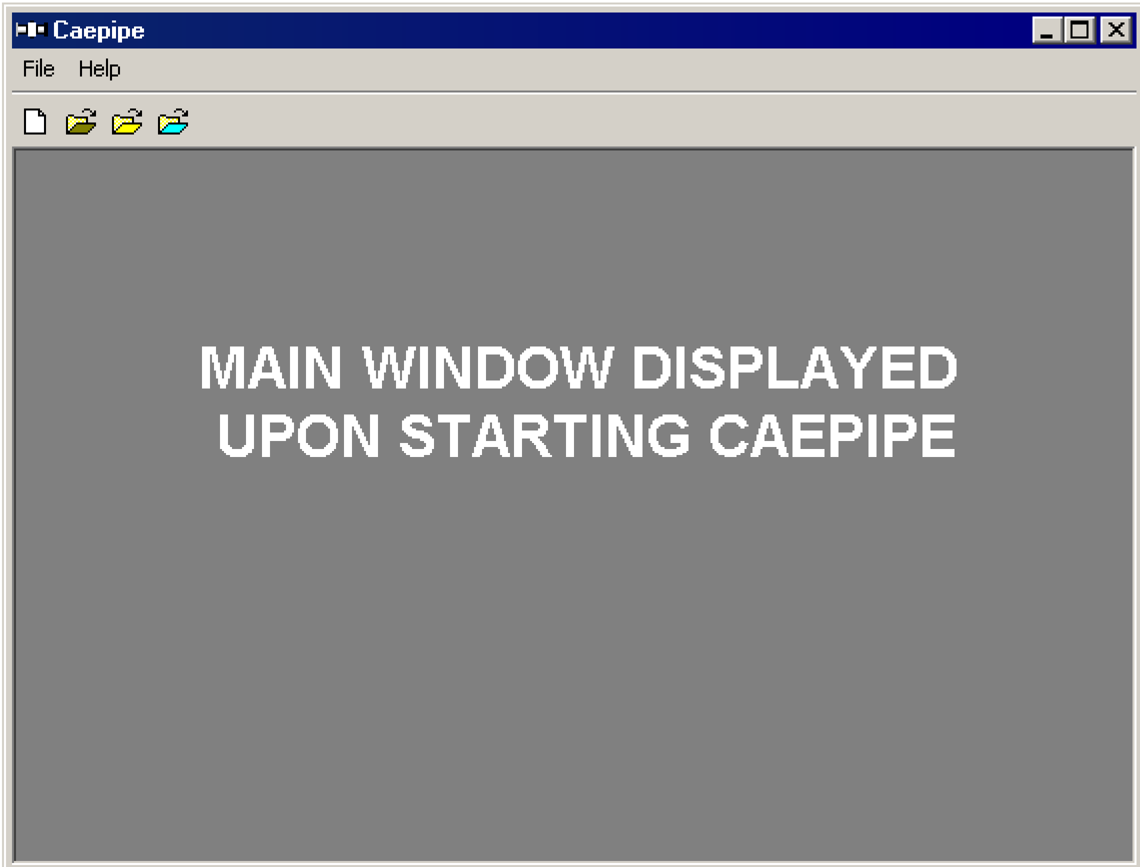
For the dialogs, the tabs are shown as

Tab

and the buttons are shown as

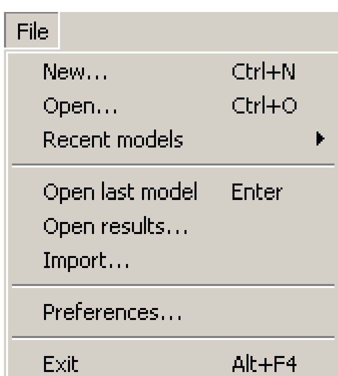
Button

The Main window is the first window you encounter when you start CAEPIPE.



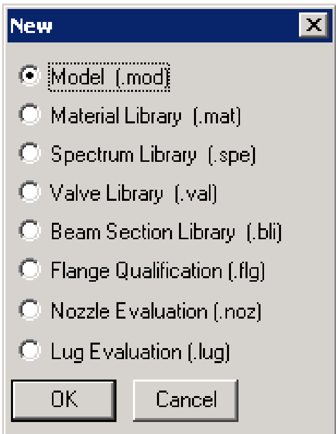
File Menu

This menu contains commands to open files and set user preferences.



.New (Ctrl+N).

This command opens a New File type dialog window. You can create any of the following file types associated with CAEPIPE from this window.



Model (.mod) - Creates a blank layout in CAEPIPE for a new piping model file.

Material Library (.mat) - Creates a user-defined material library.

Spectrum Library (.spe) - Creates a user-defined spectrum library.

Valve Library (.val) - Creates a user-defined valve library.

Beam SectionLibrary (.bli) - Creates a user-defined beam section library

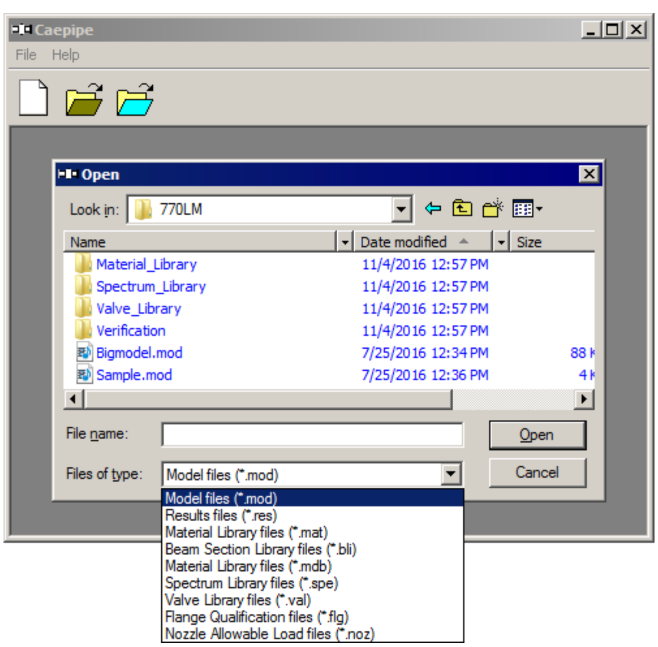
Flange Qualification (.flg) - Creates a new file for flange qualification.

Nozzle Evaluation (.noz) - Creates a new file for nozzle evaluation. Refer to Technical Reference Manual for details.

Lug Evaluation (.lug) - Creates a new file for lug evaluation. Refer to Technical Reference Manual for details.

.Open (Ctrl+O).

From the Open command you can open any number of CAEPIPE compatible files.



Model files (*.mod) - Opens CAEPIPE Model files
Results files (*.res) - Opens CAEPIPE Results files
Material Library files (*.mat) - Opens CAEPIPE Material Library files
Beam Section Library files (*.bli) - Opens CAEPIPE Beam Section Library files
Material Library files (*.mdb) - Opens CAEPIPE old Material Library files created using 4.x versions of CAEPIPE. If any modification is done to a .mdb file, the modified version is always saved in .mat format.
Spectrum Library files (*.spe) - Opens CAEPIPE Spectrum Library files
Valve Library files (*.val) - Opens CAEPIPE Valve Library files
Flange Qualification files (*.flg) - Opens Flange Qualification files
Nozzle Evaluation files (*.noz) - Opens Nozzle Evaluation files
LugEvaluation files (*.lug) - Opens Lug Evaluation files

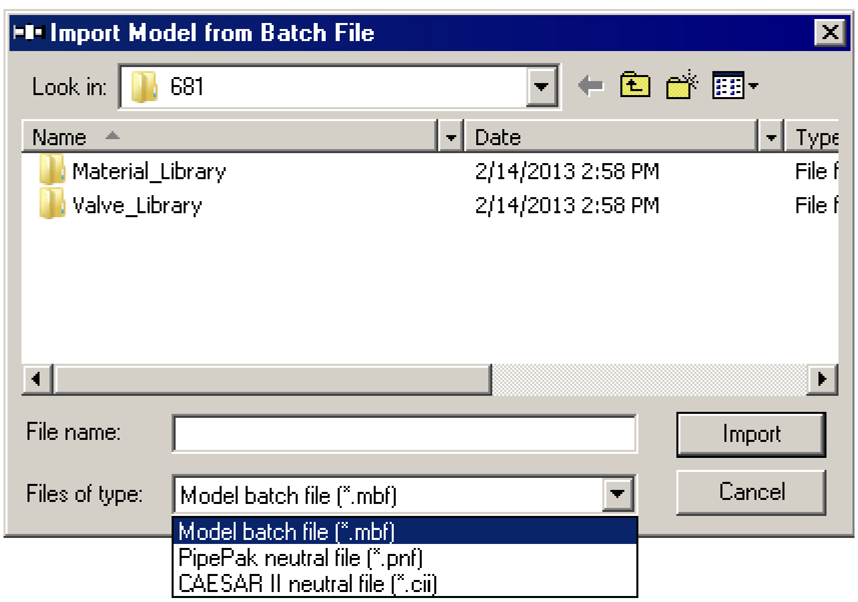
.Open last model (Enter).

This command can be useful when you need to open the same model repeatedly. Simply pressing Enter from the Main window will open the model for you. There is no need to go through the Open File dialog.

.Import.

You can import a CAEPIPE neutral file (.mbf), an Algor (Autodesk) PipePakneutral file (.pnf) or a COADE (Intergraph) neutral file (.cii) by using this command.

See Appendix A for details.

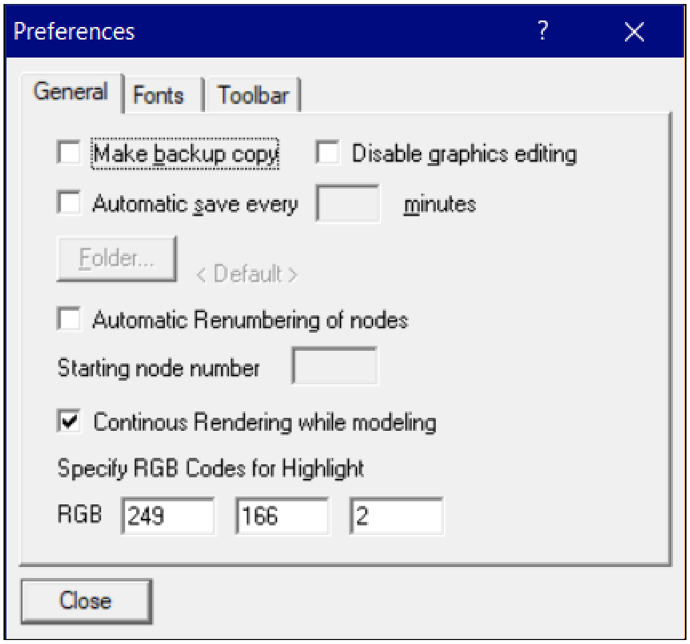


.Preferences.

Use this command to set your preferences for automatic saving and backup of the model, setting text and graphics fonts and toolbar preferences.

Preferences command has three tabs - Backup, Fonts and Toolbar.

General



Make backup copy

With this checked, every time you save your model, CAEPIPE copies the previously saved model data to another file (for backup, with a .bak extension). You can select the location where you want these files saved (click on Folder button). To open a backup file, rename the file to somename.mod before opening it in CAEPIPE.

Disable Graphics Editing

Graphics editing feature (in the Graphics window) can be turned ON/OFF from here. You may want to do this if your windows get garbled while using this feature.

Automatic Save

With this checked, and a time value entered, CAEPIPE will periodically save model data to a file named yourmodelname.sav, in the directory pointed to by Folder, or to untitled.sav.

If the software or hardware crashes, you can easily retrieve the most recently saved data. Locate the file (yourmodelname.sav) and rename the extensionfrom .savto .mod before opening it in CAEPIPE.

Folder

Select any folder to which you (as a user) have write permission to. Please make sure you check with your system administrator. CAEPIPE will try to issue you a warning if the folder you selected does not have write permission. *<Default> points to the folder that contains the model file (.MOD).*

Example: You can specify a new folder called CAEPIPE.BAK as a backup and save folder, so that CAEPIPE writes all backup files (.bak) and timed saves (.sav) to this folder.

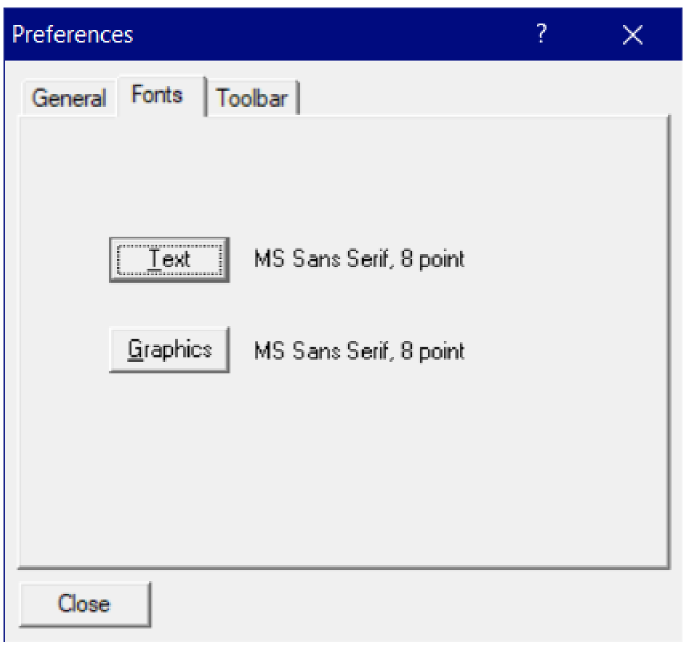
Automatic Renumbering of nodes

When you delete (an) existing node(s) from a model in Layout, CAEPIPE automatically renumbers nodes sequentially from the "Starting Node Number" you specify. So, if you do not want CAEPIPE to automatically renumber nodes when you delete any node in the Layout window, uncheck this option here. Also note that the renumbering operation upon node deletion is instantaneous, making it hard to identify which rows changed.

Continuous Rendering while modeling

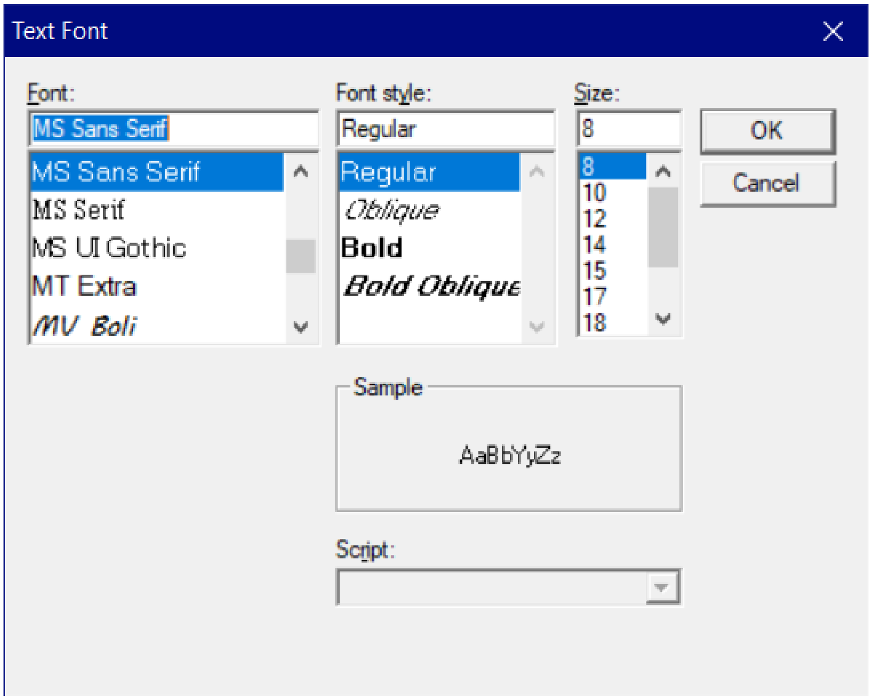
You can turn ON or OFF the continuous rendering mode for Graphics. This feature will work when the Layout Frame is active. When turned ON, the user can specify the RGB color for highlighting the current element or data by specifying the value in the fields provided.

Fonts



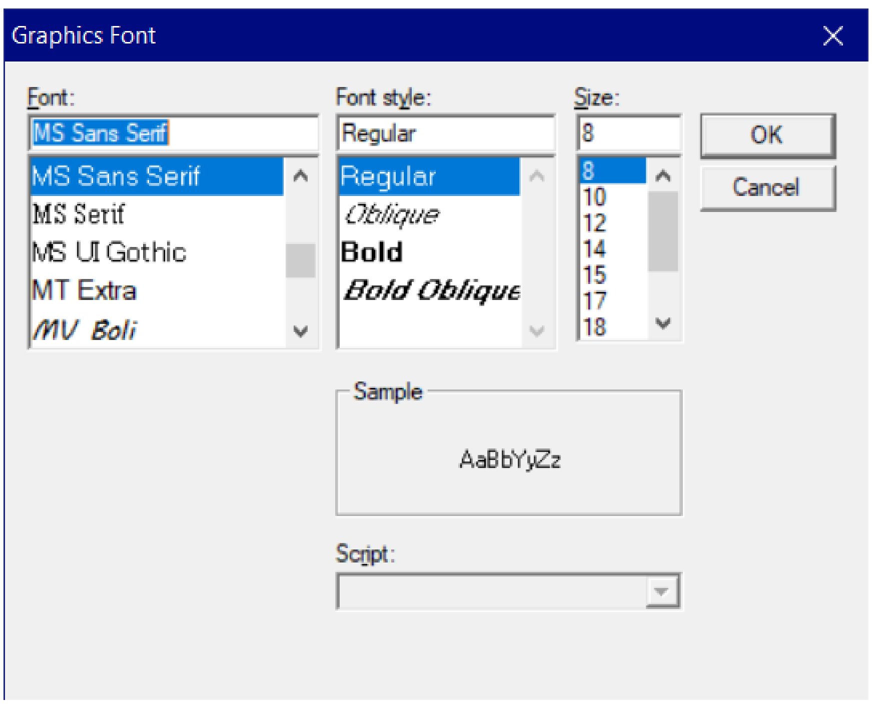
Text

Select a font face, font style and size to use inside all CAEPIPE text windows (Layout, List and Results).

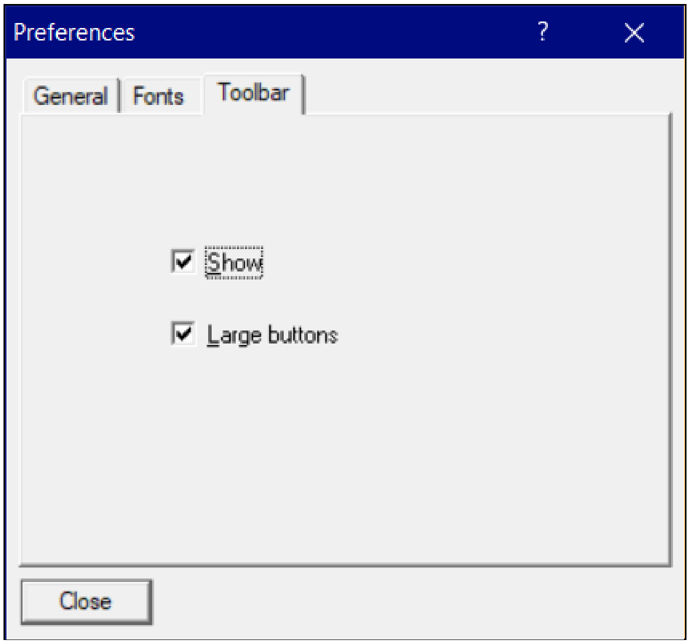


Graphics

Select a font face, font style and size to use inside the Graphics window.



Toolbar



Show

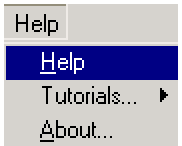
You may choose to display or not display the toolbar in CAEPIPE windows.

Large Buttons

Two sizes for toolbar buttons are available. Selecting this command will show the toolbar with large buttons.

Help Menu

From any CAEPIPE window - Main, Layout, List, Graphics or Results, you have access to the same Help menu.



.Help.

Clicking on Help command opens the CAEPIPE on-line help file.

Note:

Microsoft Windows 10 has discontinued the feature of opening .hlp file. Hence, clicking this "Help" option in Windows 10 will open the link https://support.microsoft.com/en-us/help/917607/error-opening-help-in-windows-based-programs-feature-not-included-or-help-not-supported.

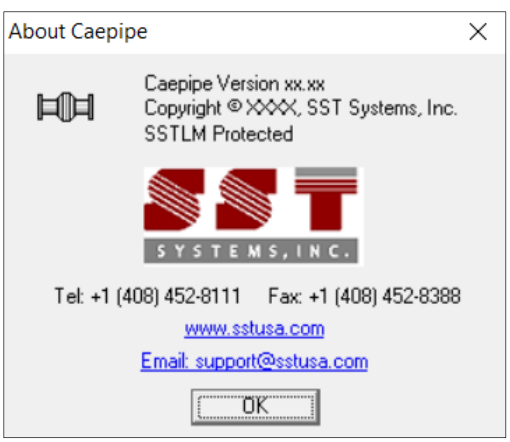
.Tutorial.

We have three tutorials that are accessible from Start Menu > CAEPIPE:

1. Basic Pipe Stress Analysis Tutorial (Basic_Pipe_Stress.pdf)
2. Tutorial 1 (Tutorial_1.pdf)
3. Tutorial 2 (Tutorial_2.pdf)

The CAEPIPE Tutorials are available separately as an Adobe Acrobat (PDF) file. If you can't locate them inside your CAEPIPE program files folder, then visit our web site www.sstusa.com to get a copy.

.About.



The About command shows the program version number and (your)hardware key serial number if the hardware key was supplied at the time of software delivery. Our support staff might ask for this information when you contact them.

Also from the "About" dialog, you can access the SST website by clicking on www.sstusa.com or send email to Technical Support by clicking on support@sstusa.com.

The text window displayed (that contains the piping layout) when you open a piping model is called the Layout window.

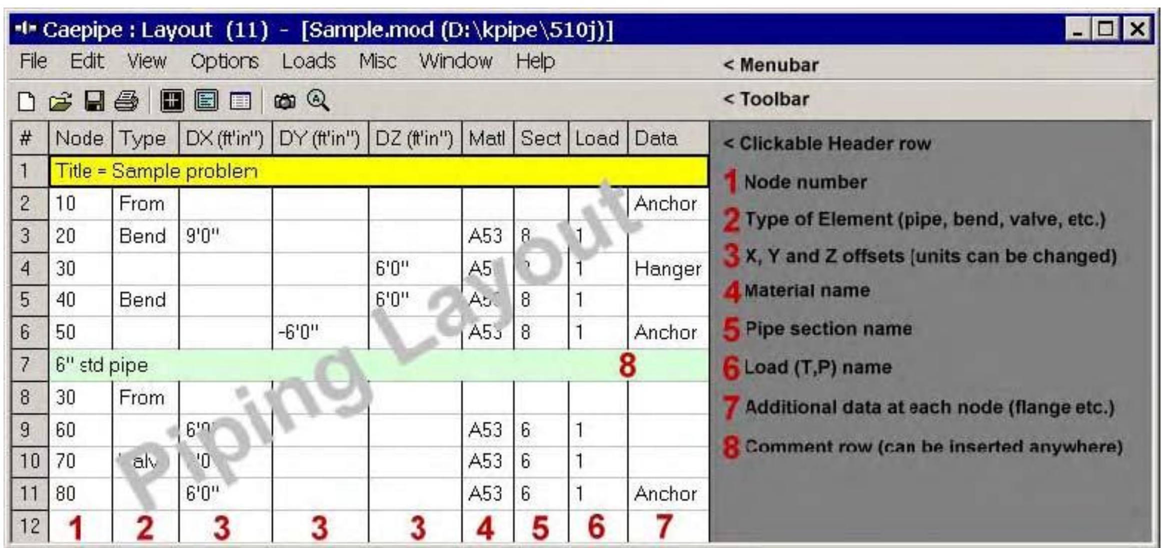
A Layout window, as the name indicates, allows you to create or modify the layout of your piping system.

Other things you can do here are:

- Specify material types, pipe sections and loads.
- Use some graphics features (zoom all, viewpoint, etc.) to view the model.
- Define analysis options such as piping code to use, reference temperature, cutoff frequency for mode extraction, as well as set units, etc.
- Define load cases (Sustained, Expansion, Response Spectrum, etc.) for analysis.

In the Layout window, the row you are working on is highlighted by the yellow highlight bar (in figure below, the highlight is on the Title row). The following illustration names the different fields in the Layout window. Presently, there are ten fields (columns) in the Layout window.

The Header row with column headings (see “Clickable Header row” in the illustration) is a special row that allows you to click on it to perform a related operation. For example, you can click on Type to show Element types.



Description of Fields (numbers in parentheses refer to numbers in the illustration above).

(Row Number)

The first field contains the row number, automatically incremented and non-editable.

(1) Node

This is the node number field where you type in node numbers as you model the piping system. The numbers should be numeric except when you are using internally generated nodes (A, B, C, D, & J) and absolute coordinate nodes.

To define an absolute coordinate of a Node, * should be prefixed with the Node number.

Nodes with suffix “A” and “B”refer to the end nodes of the curved portion of the bend called the near and far ends.

Similarly, Nodes with suffix “C” and “D” refer to the end nodes of the curved portion of the jacketed bend called the near and far ends and Node with suffix “J”refers to the end node of a jacket pipe.

To change an existing node number, click in this field and use Backspace or Delete key and type in a new node number. Press Tab or Enter and confirm the change.

You need not enter each node number. It is automatically generated when you press Tab to move to the next field. You can set an increment under the menu Options > Node increment command.

(2) Type

This is the element type field. An element (in-line) is a fitting or a component between two nodes. This could be a pipe, a bend, a valve, an expansion joint or most items listed under the Element types dialog. See a description later in this section under Element Types.

(3) DX, DY, DZ

These fields are offsets (also called relative coordinates) in the three global X, Y and Z directions. They are measured from the previous node’s location.

Offsets are components of lengths of elements (exceptions are zero-length elements like a ball and a hinge joint). For example, the offsets for a 6 feet long pipe routed 30 degrees to global X-axis (in X-Y plane) are: DX=5.196 ft. (6 * cosine 30°), DY=3 ft. (6 * sine 30°).

If the element is in a skewed direction, then calculate the X, Y and the Z components (using basic trigonometry) and input them here or use the Slope command to have CAEPIPE calculate the components.

When you specify values for the DX, DY and DZ fields for a “From” element type row, they are treated as (absolute) coordinates of the node rather than (relative) offsets from the previous node. If the model begins at the global origin (0,0,0), you do not need to specify coordinates for the first “From” node. For any other “From” row with a previously defined node number, you do not need to enter coordinates either because CAEPIPE already knows the coordinates of this node (for example, node 30 on row 8 in the previous figure).

The offsets can be positive or negative, depending on the way you route your piping.

Alternatively, you can also enter absolute coordinates of a Node directly in the layout by adding “*” following the “Node” number in the Node field instead of calculating DX, DY and DZ. For example, with 20* in the Node field, enter the absolute coordinates X, Y and Z of Node 20 under DX, DY and DZ columns.

The default number of decimals you can input for a length value is four. With an environment variable “INCREASE_DIGITS” set to “YES”, the number of digits you can input after the decimal can be increased to seven. This data can be printed other than to TXT and CSV formats.

(4) Matl (Material)

Type in the material name of the material, you want to use for this element and subsequent elements until you want to change to another material for another set of elements. See Find and Replace command under Layout window > Edit menu, for editing options.

(5) Sect (Section)

Type in the section name of the pipe section you want to use for this element and subsequent elements until the pipe section changes. See Find and Replace command under Layout window > Edit menu, for editing options.

(6) Load

Type in the load name of the load you want to use for this element and subsequent elements until the load (temperature and/or pressure) is modified. See Find and Replace command under Layout window > Edit menu, for editing options.

(7) Data

Use this field to enter data items at a node such as anchors, flanges, nozzles, hangers, SIFs, etc. See Data types below for how to specify an item.

(8) Comment

You can include a comment anywhere you like in the Layout window by simply pressing "c" (first) on an empty row at node column. You can use this feature to annotate the model with notes. Comments print along with the layout data so you can use this feature to document (revisions to) the model.

A specialized form of a comment can be entered to input a Hydrostatic test load by pressing "h" first on an empty row at node column.

A hydrostatic test is a way in which pipelines are tested for strength and leaks. The test involves filling the pipe system with a test liquid, usually water, which may be dyed to aid in visual leak detection to the specified test pressure.

The Hydrostatic load is applied to the model from that row onwards until modified again (in the next figure, the Hydrostatic load is applied to model from row 8 onwards). When a Hydrostatic load is defined, CAEPIPE will automatically include a new load case "Hydrotest" under Layout window > Loads > Load cases.

5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	Hydrotest load: Spec. gravity = 1.0, Pressure = 300 (psi), Exclude insulation								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor

Menu bar

File Edit View Options Loads Misc Window Help

File menu:

- Standard file operations with a few special ones to note - Open Results (model file (.mod) must be present for this operation), Print (to file), Analyze (model), QA Block, Recently opened file list. You can open not only the CAEPIPE model files, but also material and spectrum library files, and the CAEPIPE Results files.

Edit menu:

- You can edit all types of data (elements, supports, range of rows) from here. Use Ctrl T to edit element (bend, valve), Ctrl D to edit Data (flange, anchor). You can change properties for groups of elements, split elements and generate copies of existing rows.

View menu:

- Mainly for Graphics (Viewpoint, Previous view and Zoom All). Other functions are to open the List window (Ctrl L) by selecting one of the displayed items, and search for a node number.

Options menu:

- For setting Analysis options such as Piping code, Reference Temperature, Pressure correction, cutoff frequency for mode extraction, etc., set Units (SI, Metric, English or a combination), and customize the look of CAEPIPE by choosing a display font.

Loads menu:

- Set the load cases for analysis, and specify loads for the model such as static seismic, wind, spectrum, time history, etc.

Misc menu:

- View model coordinates, define/view materials, sections, loads (also for beams), pumps, compressors and turbines, time functions and spectra.

Window menu:

- From here, you can move the focus to another window (such as Graphics or List). F2 will move focus between text and Graphics windows and F3 between open text windows. The real advantage of these hotkeys (F2 and F3) is for users who have a smaller monitor and work with maximized windows for input, list, graphics and results. In such a setting, one key press (F2 or F3) will quickly move the focus to another window (without having to either minimize the one where the focus is or switch windows through the taskbar).

Help menu:

- For on-line help and information pertaining to the remaining period of your yearly Maintenance, Enhancement and Support (ME&S) agreement with SST.

Toolbar

The toolbar presently is organized into three groups: File, Window and Graphics operations.

- File operations are New (model), Open (a model), Save (model), and Print.
- Window operations are (move focus to) Graphics window, (move focus to) List window (if open), and display a dialog of List of items to show an item in the List window.
- Graphics operations are Viewpoint and Zoom all.

In the opening window (when you first start CAEPIPE), there is a button to open the last opened model file (hotkey: Enter). This is not available from other windows.

Hotkeys You Can Use

You can use following hotkeys in the Layout window:

Tab	Move cursor from field to field (left to right)	Home	Move cursor to the first character in a field
Shift Tab	Same as Tab but right to left	End	Move cursor to the last character in a field
Page Up	Move highlight up one page	Ctrl+Home	Top of model
Page Down	Move highlight down one page	Ctrl+End	End of model
Delete/BkSp	Delete characters	Arrow keys	Move highlight one row/field at a time
Ctrl+Insert	Insert an empty row	Ctrl+X	Delete a row
Ctrl+Enter	Duplicates last layout row. Applicable only for Beam/Pipe/Jpipe	F1	Help
F2	Graphics window	F3	List window
Ctrl + Up	Increase font size in all CAEPIPE text windows (Layout, List and Results).	Ctrl + Down	Decrease font size in all CAEPIPE text windows (Layout, List and Results).

Other hotkeys are listed under menus.

Hotkey combination: CTRL+ENTER (for duplicating the last layout row) works for Beam Elements (in addition to PIPE and JPIPE).

Function Keys

F1: Opens on-line help.

F2: This key moves the focus to and from the Graphics window.

F3: This key moves focus to and from the List window.

F4: Opens the Viewpoint dialog which you can set for the Graphics window.

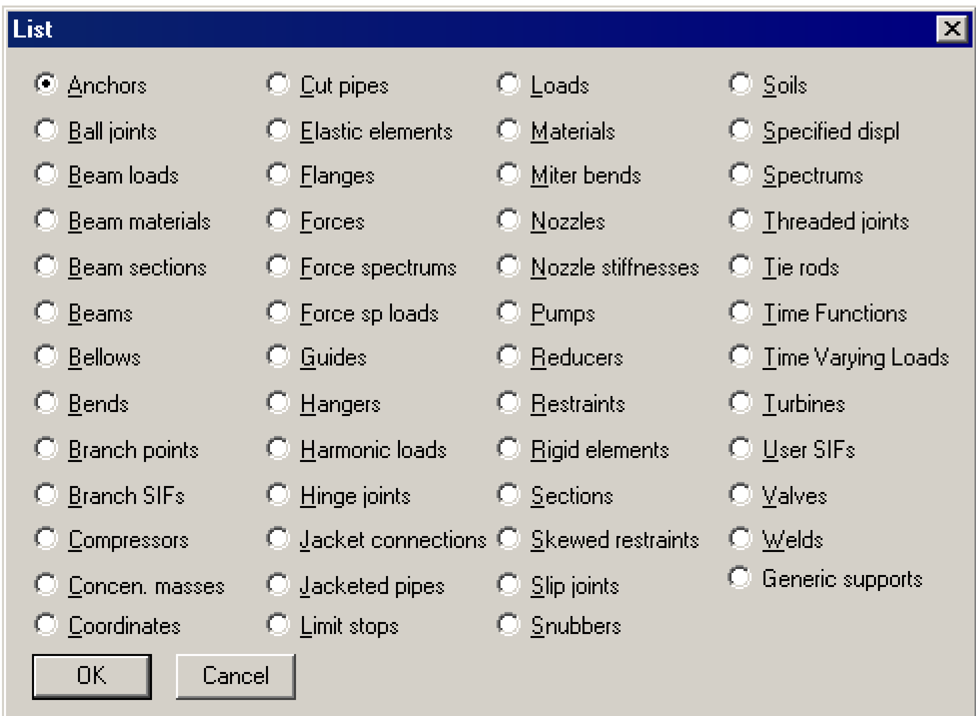
F5: This function key restores the Graphics window to the previous graphics view.

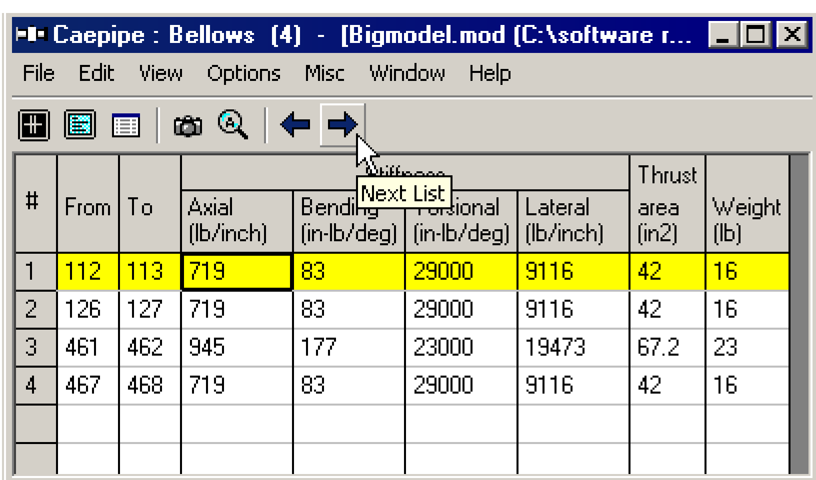
List (Ctrl+L)

List is a powerful feature that helps you see itemized lists of information in the model. All input data can be viewed through this command. For example, you can view all elements and data types specified in the model, list Coordinates of the nodes of the model, Specified displacements, Nozzle stiffnesses, Branch nodes, Threaded joints, Pumps, Turbines, Compressors, Materials, Sections and Loads, etc.

This is a useful tool to verify information after modeling is finished, for example, to check whether all bends have the correct bend radius or whether limit stops are aligned in the correct directions or how many anchors have been specified in the model and so on.

This command is "intelligent" in that the list dialog shows only those items that exist in the model. Consequently, List can display a small list or as comprehensive a list as shown in the next figure.



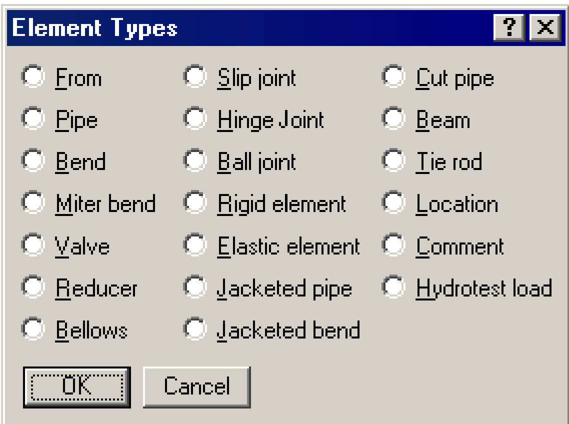


You can navigate through the different itemized lists by clicking on the two blue-colored arrows on the toolbar.

Element types

How to input an Element

To input an element (between two nodes) in a new row, either right-click in the Type field of the row or left-click on Type (in the Header row) to display the Element types dialog (or press Ctrl+Shft+T). Select the element you want from this dialog to insert into the model.



Notice the different types of elements available. Any of these can be input for "Type." Some of these are strictly not elements such as From, Location, Comment and Hydrostatic test load. All of these have keyboard shortcuts. You do not have to necessarily open this dialog to input any of these. You can type the first letter or the first few letters (example: Be for bend, M for miter, V for valve, etc.) to input an element type. The keyboard shortcuts are listed below. However, for Comment, simply press "c" (first) on an empty row, and for Hydrostatic test load, simply press "h" (first) on an empty row.

Shortcut keys for Element types

b (Tab)	Bend
ben	Bend (no Tab required, and typing "ben" moves the cursor over to the DX field.)
bea	Beam
bel	Bellows
ba	Ball joint
c	Cut pipe (Cold spring)
e	Elastic element
f	From
h	Hinge joint
j	Jacketed pipe
jb	Jacketed Bend
l (L)	Location
m	Miter Bend
P	Pipe (you don't have to type this because a Pipe is the default element type)
r (Tab)	Reducer
re	Reducer
ri	Rigid element
s	Slip joint

v Valve
t Tierod

Once input, each element name appears under the Type field. The only exception is the Pipe element which is indicated by a blank. For example, see the first annotated figure in this section. The field under Type is blank on the row that contains Node 30 signifying a pipe.

How to change the Element type

If you want to change a Valve element to an Elastic element, then you have to do the following: Move the highlight bar to the row that contains the Valve (or click once on Valve). Delete the word "Valve" (Backspace) and press "E", confirm the change;

or

Assuming the highlight is on the Valve row, click on Type (in the header row) to display the Element types dialog and then select Elastic element from it. The Valve changes to an Elastic element after you confirm the change.

How to change Element properties

From the keyboard: Press Ctrl+T (from anywhere on the row) to edit the element. Note: A pipe has no editable property, and Ctrl+T does nothing.

Mouse: Right clicking on the element gives you applicable choices. For example, for a Bend, you are shown Edit Bend and List Bends as the choices.

Notes:

1. A pipe element is the default element type. So, leave the element type blank for that row.
2. An existing element type (say, a bend) cannot be changed to a Location type.
3. To change the element type, you can erase (use Backspace or Delete key) the existing element and specify a new one (using accelerator key), or usemouse to specify another element by clicking on Type in the header row. Or, use Ctrl+X to delete the entire row.

Data types

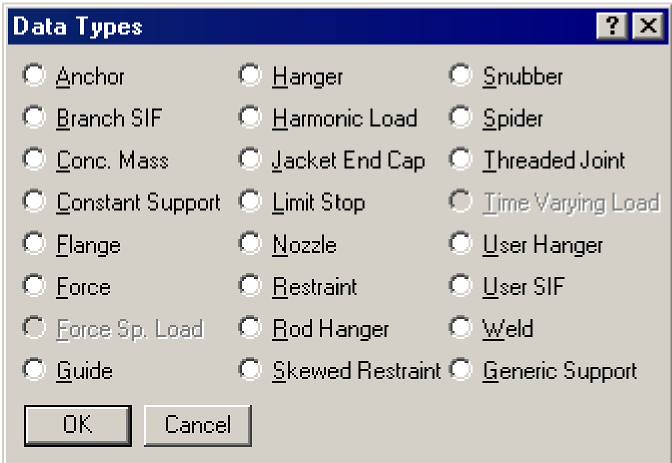
How to input a Data item

When you want to input a support, a restraint, a flange, an SIF, etc., at a node, use the mouse or the keyboard (several key accelerators are available to make your job easy) to open the Data types dialog and select an item from it.

Keyboard: Press Ctrl+Shift+D to open the Data types dialog to select an item, or, type one of the following keys (under Shortcut keys) that will automatically enter the respective Data type (press Ctrl D to position the cursor in this field):

Mouse: To open the Data types dialog,

- Left click on Data (in the header row), or
- Right click in the Data column, or
- Select Misc menu >> Data Types.



Shortcut keys for Data types

a	Anchor (rigid)
br	Branch SIF (Welding tee, Branch connection, etc.)
c (Enter)	Concentrated Mass
conc	Concentrated Mass
cons	Constant Support
f (Enter)	Flange
fl	Flange
fo	Force
gu	Guide
ge	Generic Support
h (Enter)	Hanger (to be designed)
han	Hanger (to be designed)
har	Harmonic load
j	Jacket end cap
l (L)	Limit stop
n	Nozzle
r (Enter)	Restraint (2-way), see also X, Y, Z below
ro	Rod hanger
s (Enter)	Skewed restraint
sk	Skewed restraint
sn	Snubber
sp	Spider
th	Threaded joint
t (Enter)	Time varying load
ti	Time varying load
u (Enter)	User hanger
user s	User SIF
w	Weld
x	X restraint (2-way)
y	Y restraint (2-way)
z	Z restraint (2-way)

Notice the different types available. For example, press “a” in the Data field to input an Anchor (you do not have to go through the long-winded mouse method of clicking on Data in the header row to display the Data types dialog, and then selecting Anchor from it).

How to change the Data type

Assume that you want to change an Anchor to a Nozzle. Before you change an existing Data type (Anchor), you have to delete it.

To delete, click on Anchor, delete the word “Anchor” (backspace) or right click on the Anchor, and select “Delete Anchor”. Once deleted, then type “n” (for Nozzle).

Alternately, with the cursor placed on the row for which you need to change the Anchor, click on Data on the header row, and pick Nozzle and confirm the change.

How to edit the Data item’s properties

To edit properties for anchors, flanges, etc.

Keyboard: Press Ctrl+D (from anywhere on the row) to edit the item. Note: A Threaded Joint has no editable property, and Ctrl D does nothing.

Mouse: Right clicking on the item gives you applicable choices. Example: For an Anchor, you are shown Edit Anchor.

Notes:

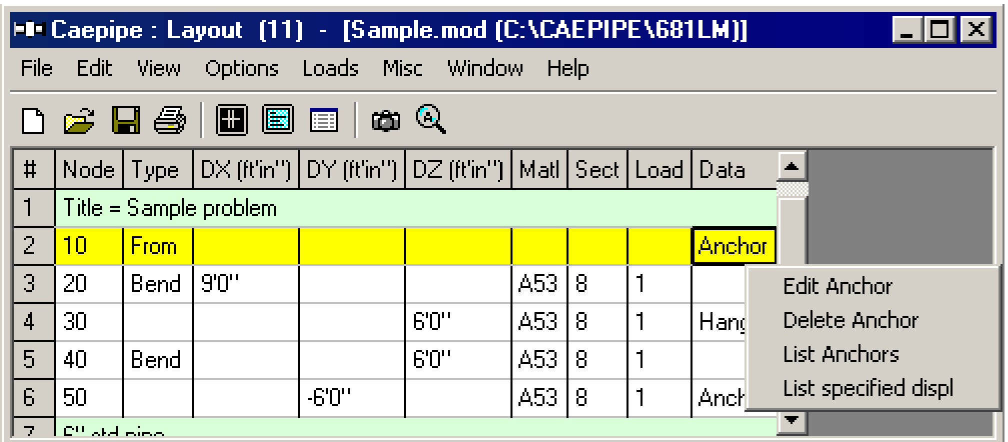
1. You cannot specify a data item for a Bend, a Miter Bend or a Jacketed Bend, as the node number is a tangent intersection point, and does not exist physically on the element. Instead, create an intermediate node for the Bend or Jacketed Bend and then specify a data item at the new node using Location type. You cannot create an intermediate node for a miter bend.

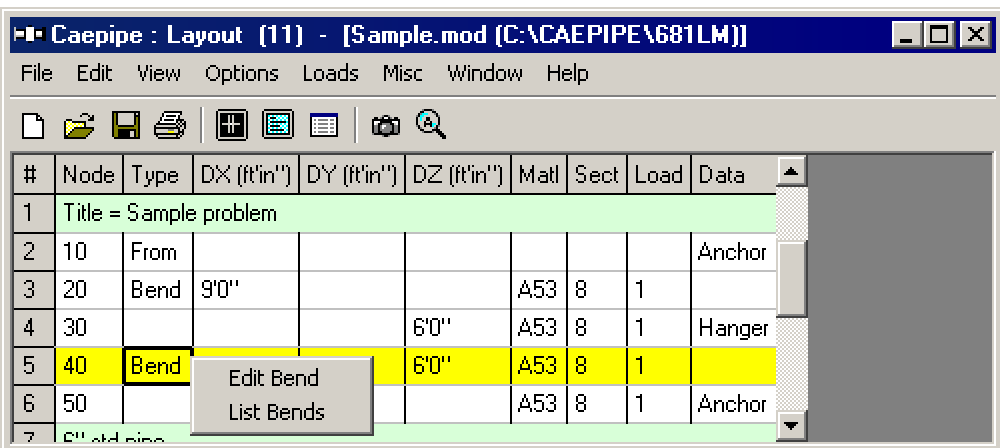
2a. You can delete an existing data item at anytime:
- Use the Delete or the Backspace key to erase the item and press Enter.
- Right click on the item and select Delete Item.

2b. Alternately, to change the data type, you can erase (use Backspace or Delete key) the existing item and type a new one (using accelerator), or use the mouse to specify another data item by clicking on Data in the header row.

Context Menus

Context menus (right click menus) can be found everywhere in CAEPIPE. For example, when you right click on an Anchor, or a Bend, you are shown the following context menus. You can perform any listed action that more than often is a shortcut (e.g., Delete Anchor or List Anchors).





Right clicking on DX/DY/DZ shows you, for example, Insert, Delete (rows), Split (an element) and (specify) Slope for an element. Right click in the Matl, Sect or Load column to see the respective context menu.

Export data from CAEPIPE

CAEPIPE has the in-built ability to Output (Export) model and/or results to the below stated formats at this time.

1. Output model and results data to CSV (Comma Separated Values) and TEXT file format through File > Print > Print to file. For further details, see sections titled "Print Model" and "Print Results" in this manual.

2. Output model and results data to PDF file format using a PDF utility such as "Microsoft Print to PDF", "doPdf", etc. through File > Print. For further details, see sections titled "Print Model" and "Print Results" in this manual.

3. Output model data to MBF (Model Batch File) through File > Export to MBF… For further details, see section titled "Import/Export" in Appendix A of this manual.

4. Output model data to PCF (Piping Component File) format through Layout Frame > File > Export to PCF. For further details, see section titled "Import/Export" in Appendix A of this manual.

5. Output model data to 3D Plant Design software PDMS, E3D and CADMATIC through File > Export to 3D Plant Design. For further details, see section titled "Export to 3D Plant Design" of this manual.

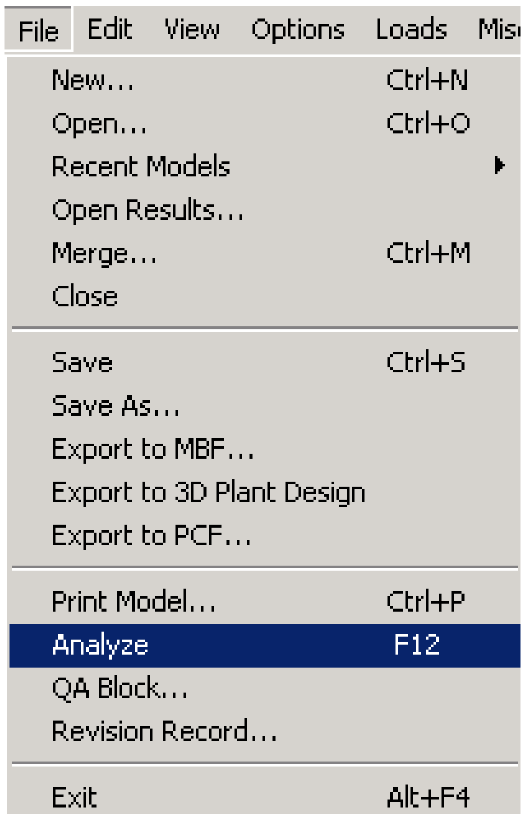
6. Output Material library data to an ASCII file. For further details, see section titled "Export Material Library" in Appendix A of this manual.

7. Output Time History results to CSV format through Results Frame > File > Export. For further details, see section titled "Export" in "Results Window File Menu" of this manual.

8. Output Hanger Report to LICAD software format (via an ASCII file .cli) through Results Frame > File > Export. For further details, see section titled "Export" in "Results Window File Menu" of this manual.

9. Output Deformed Geometry to 3D Plant Design software PDMS, E3D and CADMATIC while being in Displacements results through Results Frame > File > Export to 3D Plant Design. For further details, see section titled "Export to 3D Plant Design" in "Results Window File Menu" of this manual.

File Menu



This menu contains commands for standard file operations with a few special ones to note:

.Open (Ctrl+O).

You can open the following CAEPIPE file types from the Open menu.

1. Model files (*.mod)
2. Results files (*.res)
3. Material Library files (*.mat)
4. Beam Section Library files (*.bli)
5. Material Library files (*.mdb) [old format]
6. Spectrum Library files (*.spe)
7. Valve Library files (*.val)
8. Flange Qualificationfiles (*.flg)
9. Nozzle Evaluation files (.noz)
10. Lug Evaluation files (.lug)

For a description of each file type listed here, see subsection titled "File" under Section "Menu" above in this document.

.Open Results.

This command opens the Results files (*.res). The corresponding model file (.mod) must be present in the same directory as the results file (.res). For your convenience, the Open Results dialog shows the results filename for the currently opened model if it is already analyzed; if not, the name of the last opened results file is filled in the name field.

Note:

Results file produced out of a specific CAEPIPE Version can be opened ONLY using that version of CAEPIPE / CAEPIPE Review / CAEPIPE Evaluation. For example, Results file produced out of CAEPIPE Version 10.30 can be opened only using CAEPIPE / CAEPIPE Review / CAEPIPE Evaluation Version 10.30. In other words, Results file produced out of CAEPIPE Versions earlier than 10.30 cannot be opened using CAEPIPE / CAEPIPE Review / CAEPIPE Evaluation Version 10.30.

.Merge.

You can merge two or more piping models into one file using this command. This feature merges Materials, Sections, Loads, Beam Materials, Beam Sections, Beam Loads, Pumps, Compressors, Turbines, Specrums, Spectrum Levels and Elements from the selected model into the currently openedmodel. The process, however, skips Analysis Options, Load Cases, Wind Loads, Time History and Force Spectrums from the selected model during merging. Also, CAEPIPE checks for duplicate nodes and provides an option for renumbering the elements. Pressing “Yes” will renumber and merge the model; “No” will merge without renumbering, and you will have to manually identify and remove duplicates and/or renumber.

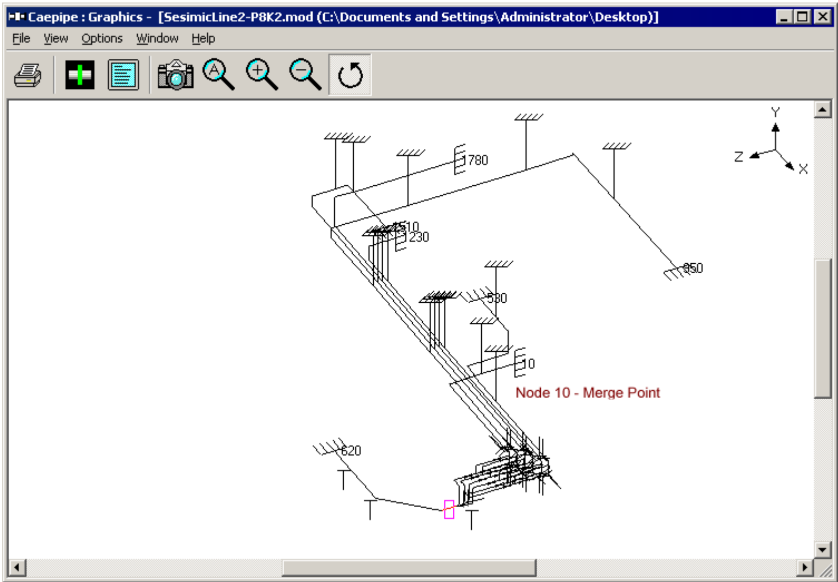
In addition to the above, CAEPIPE will do the following when a model with Spectrum and Spectrum level is defined.

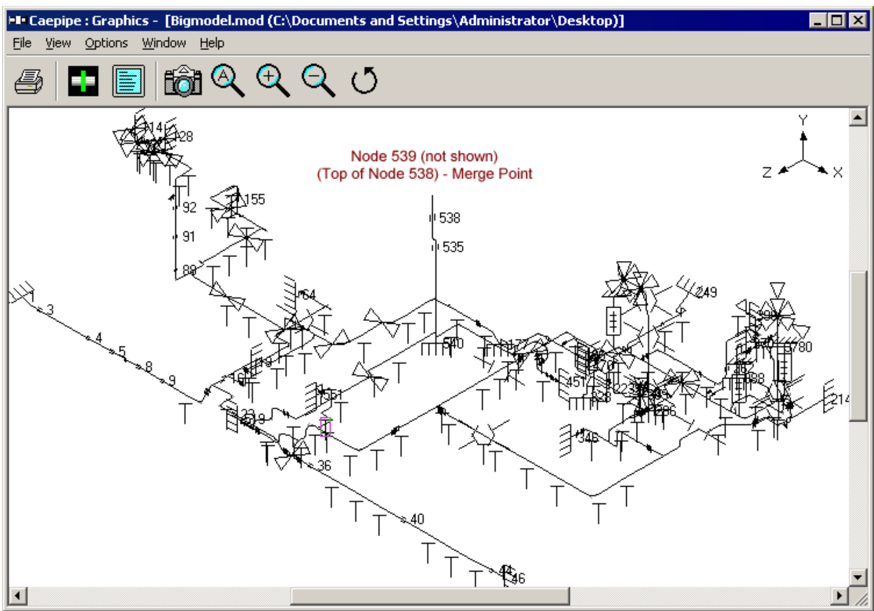
1. Rename the Spectrum automatically when the Spectrum name is already defined in the source model. When renaming is done, CAEPIPE will automatically update the Spectrum level data with the newly assigned name.
2. Rename the Spectrum Level name when the Spectrum Level name is already defined in the source model. When renaming is done, CAEPIPE will automatically update the level tag assigned at support with the updated tag.
3. Assign fist spectrum level tag name to all supports that has BLANK level tag.

Refer example under “From” in Technical Reference Manual for a suggestion on how to model separate files for merging later.

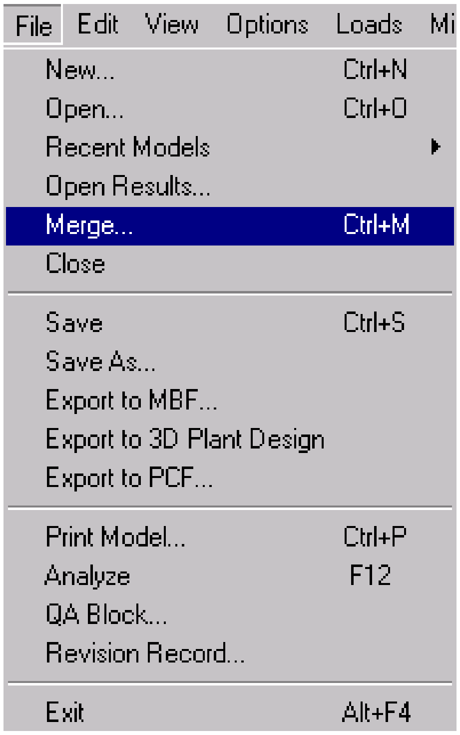
Example:

Here are two models with the note showing where the common merge point is (node 10 in the first model and node 539 in the second model).

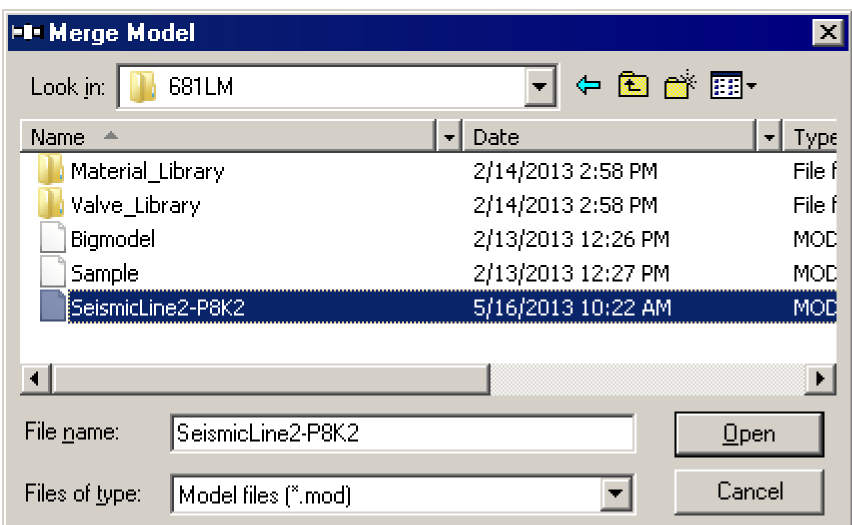




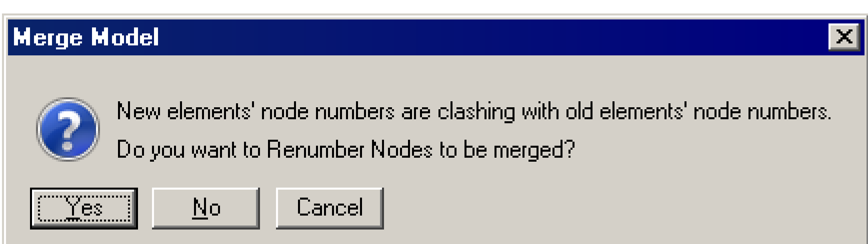
Open Bigmodel.mod first. Use menu File > Merge to start the process.



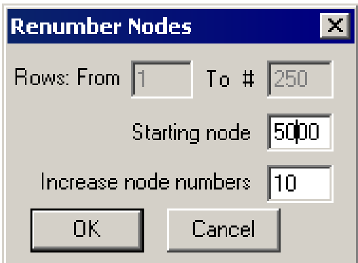
Select the second model as shown below and click on Open.



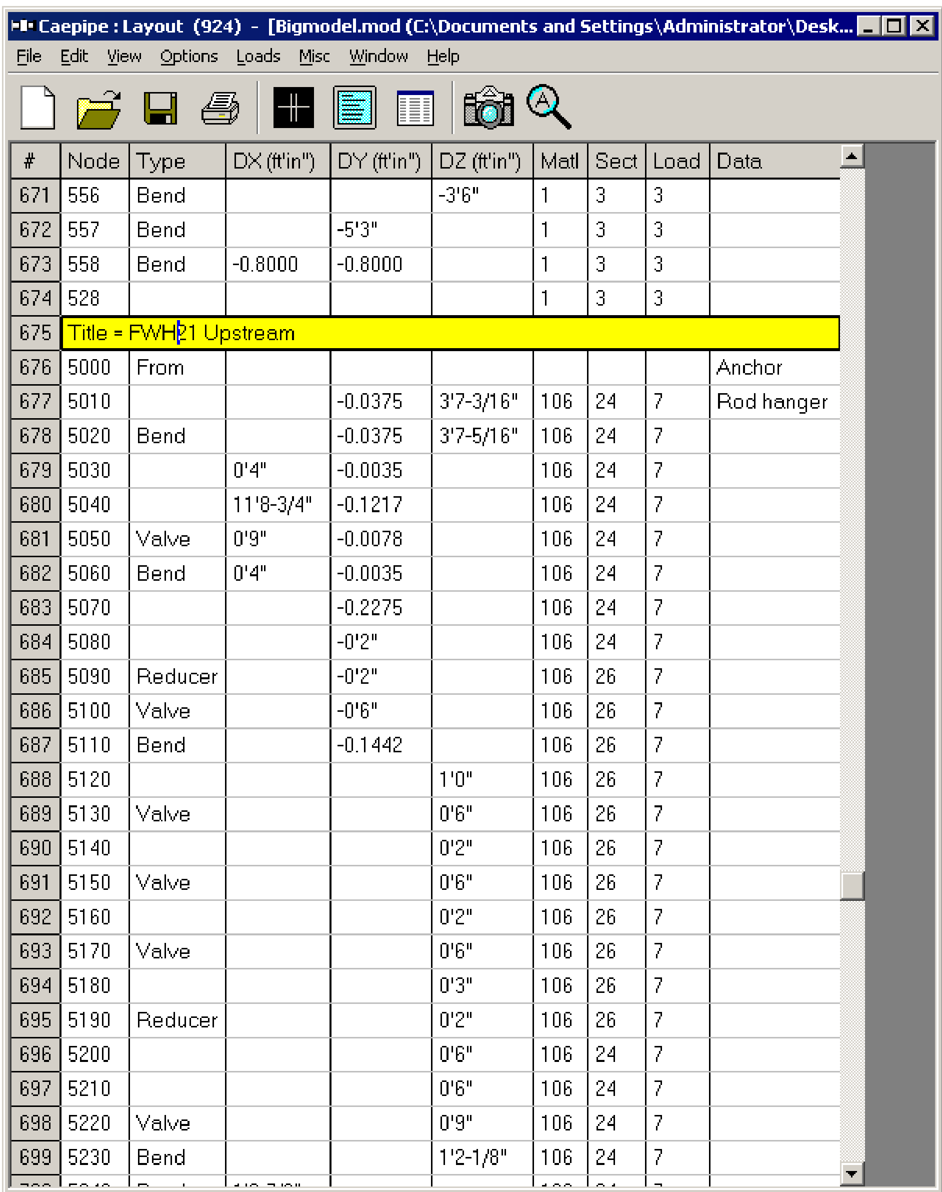
To avoid duplication of node numbers, names of materials, sections, loads, etc., click on Yes as shown below.



Choose a starting node number which is clearly not used in the first model. CAEPIPE will use the starting node number to number the nodes sequentially using the increment shown.



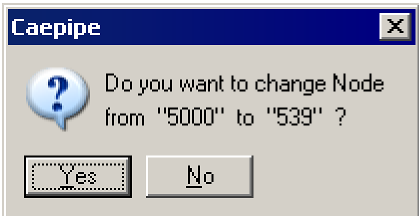
Models are merged now. The second model starts from row 675 (the row with the yellow highlight).



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
671	556	Bend			-3'6"	1	3	3	
672	557	Bend		-5'3"		1	3	3	
673	558	Bend	-0.8000	-0.8000		1	3	3	
674	528					1	3	3	
675	Title = FWH21 Upstream								
676	5000	From							Anchor
677	5010			-0.0375	3'7-3/16"	106	24	7	Rod hanger
678	5020	Bend		-0.0375	3'7-5/16"	106	24	7	
679	5030		0'4"	-0.0035		106	24	7	
680	5040		11'8-3/4"	-0.1217		106	24	7	
681	5050	Valve	0'9"	-0.0078		106	24	7	
682	5060	Bend	0'4"	-0.0035		106	24	7	
683	5070			-0.2275		106	24	7	
684	5080			-0'2"		106	24	7	
685	5090	Reducer		-0'2"		106	26	7	
686	5100	Valve		-0'6"		106	26	7	
687	5110	Bend		-0.1442		106	26	7	
688	5120				1'0"	106	26	7	
689	5130	Valve			0'6"	106	26	7	
690	5140				0'2"	106	26	7	
691	5150	Valve			0'6"	106	26	7	
692	5160				0'2"	106	26	7	
693	5170	Valve			0'6"	106	26	7	
694	5180				0'3"	106	26	7	
695	5190	Reducer			0'2"	106	26	7	
696	5200				0'6"	106	24	7	
697	5210				0'6"	106	24	7	
698	5220	Valve			0'9"	106	24	7	
699	5230	Bend			1'2-1/8"	106	24	7	

Notice that node 10 in the first model and node 5000 in the second model start from the same global origin (0,0,0).

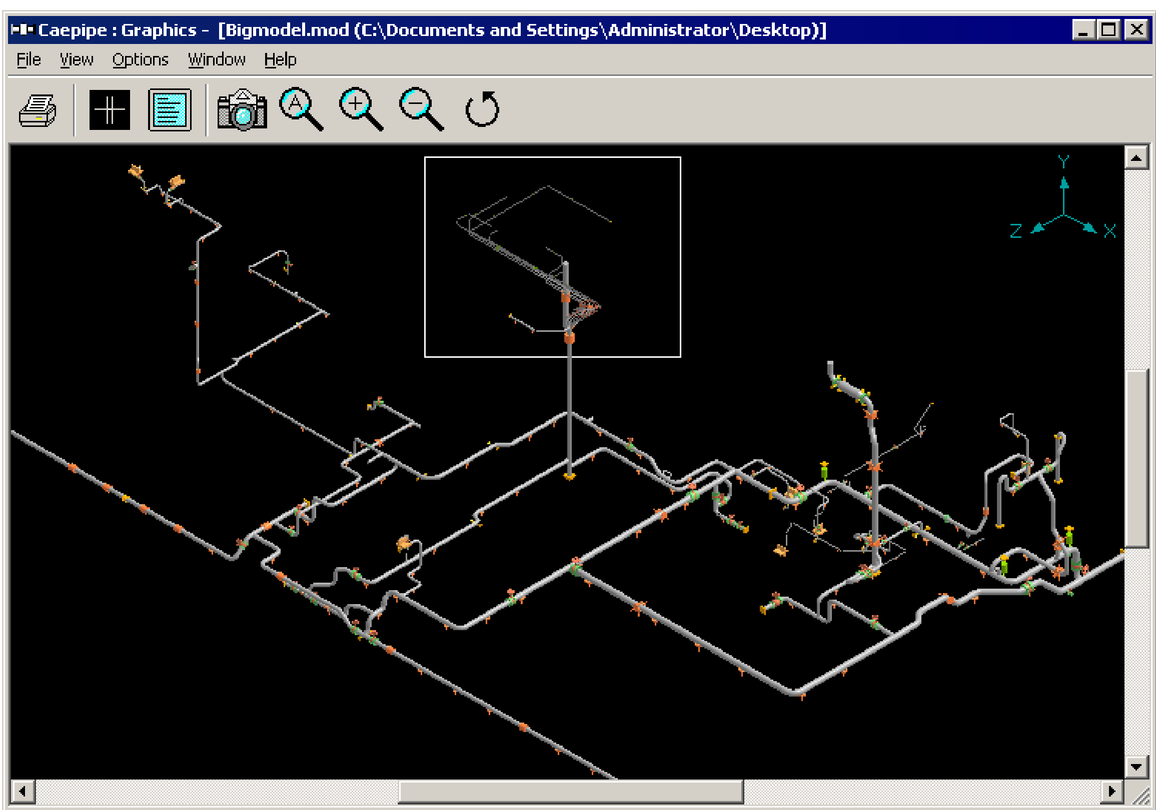
So, you need to change node 5000 to the connecting node number in the first model (i.e., node 539). Upon doing so, you are asked



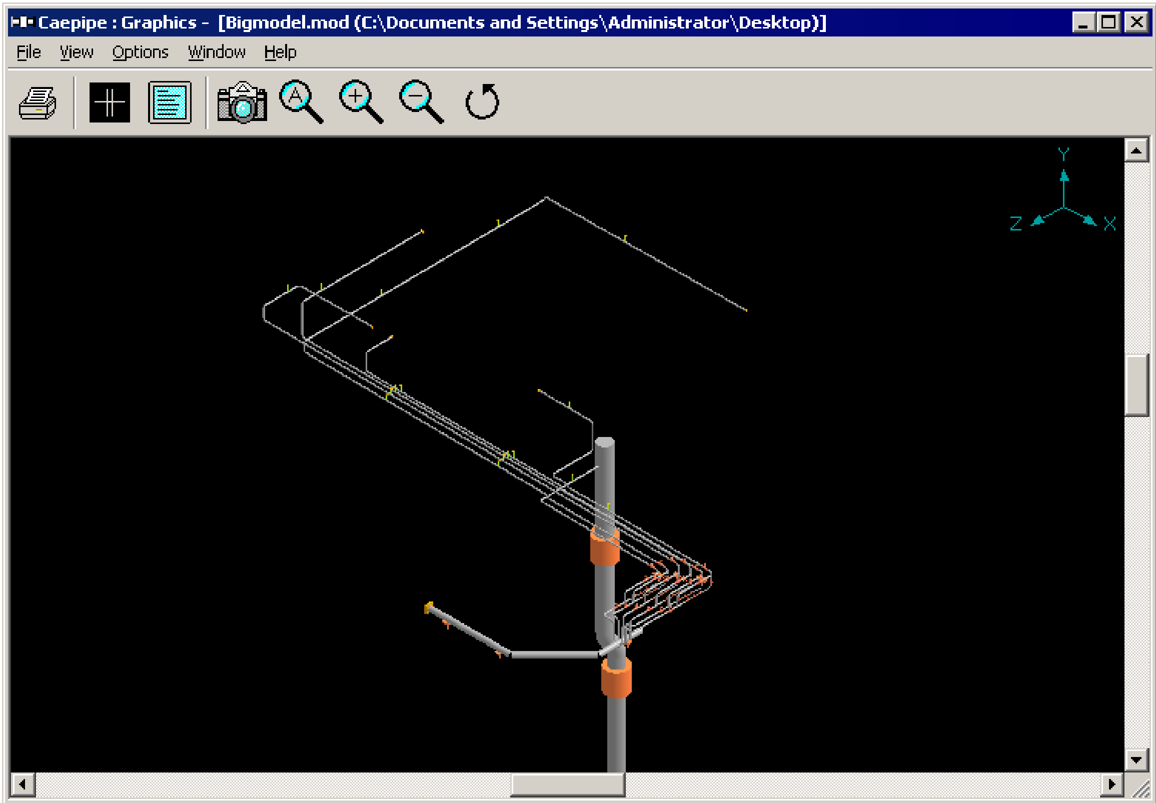
Click on "Yes" to make the change. Also, remove the anchor from the data column on row 676.

672	557	Bend		-5'3"		1	3	3	
673	558	Bend	-0.8000	-0.8000		1	3	3	
674	528					1	3	3	
675	Title = FWH21 Upstream								
676	539	From							
677	5010			-0.0375	3'7-3/16"	106	24	7	Rod hanger
678	5020	Bend		-0.0375	3'7-5/16"	106	24	7	
679	5030		0'4"	-0.0035		106	24	7	
680	5040		11'8-3/4"	-0.1217		106	24	7	

The combined model is shown in the graphic view below (with a box – for illustration only - around the first model).

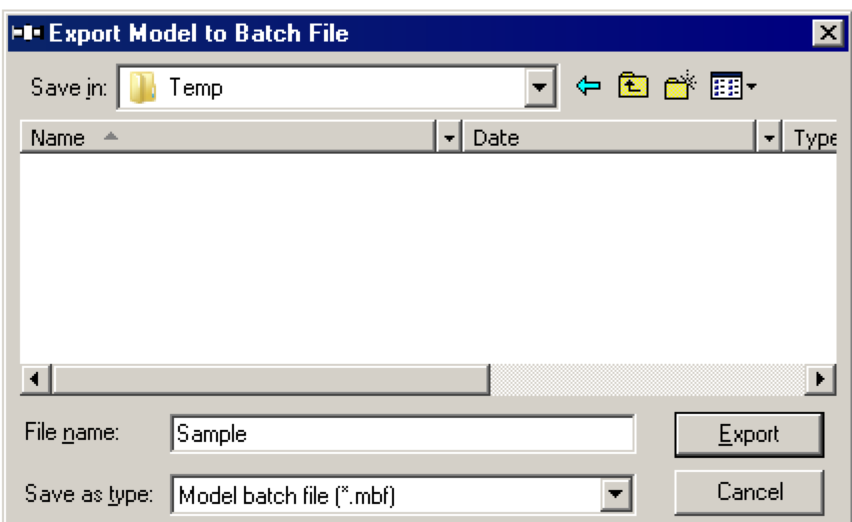


Zoomed image of the connecting node is shown below.



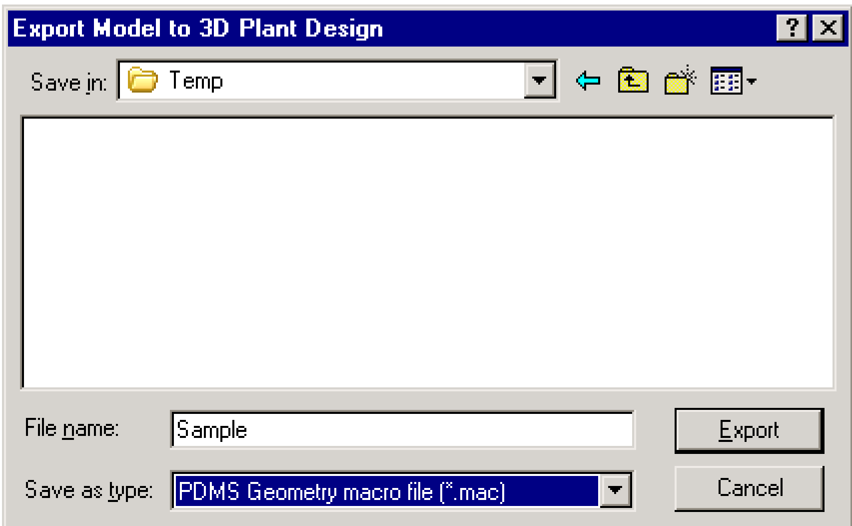
.Export to .mbf.

Presently, you can export the model data to CAEPIPE's neutral file format (called a model batch file, .mbf). See Appendix A for format details.



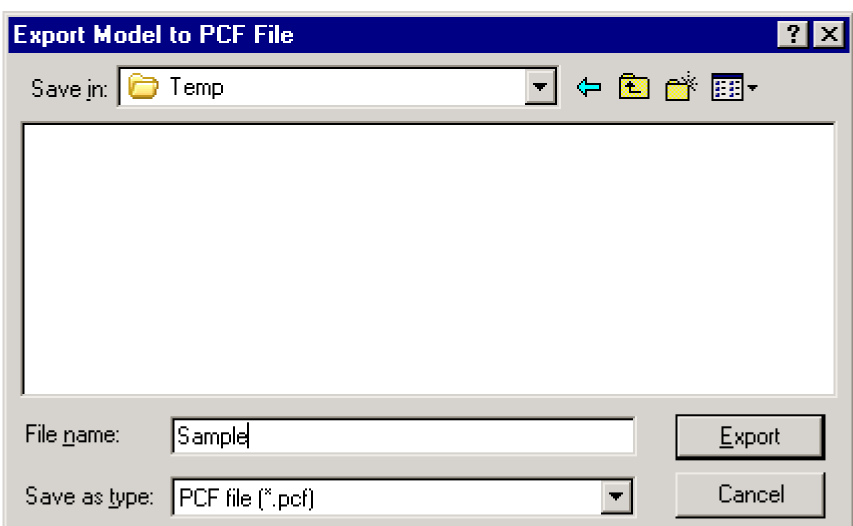
.Export to 3D Plant Design.

This feature exports the 3D CAEPIPE model to PDMS (.mac file) or CADMATIC (.3dd file) as structural components (only dumb graphics without attributes).



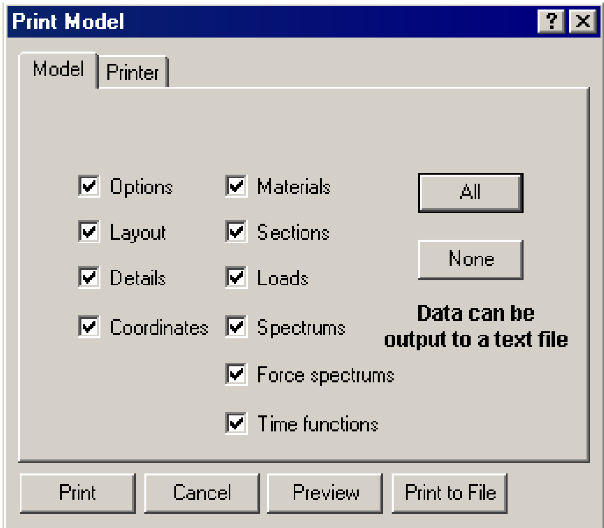
.Export to PCF.

CAEPIPE can export model data from inside the Layout window to a PCF file with the extension .pcf (piping component file). To export a model, select the menu command File > Export to PCF… from the Layout window. See section titled PCF Export in Appendix Afor more details.

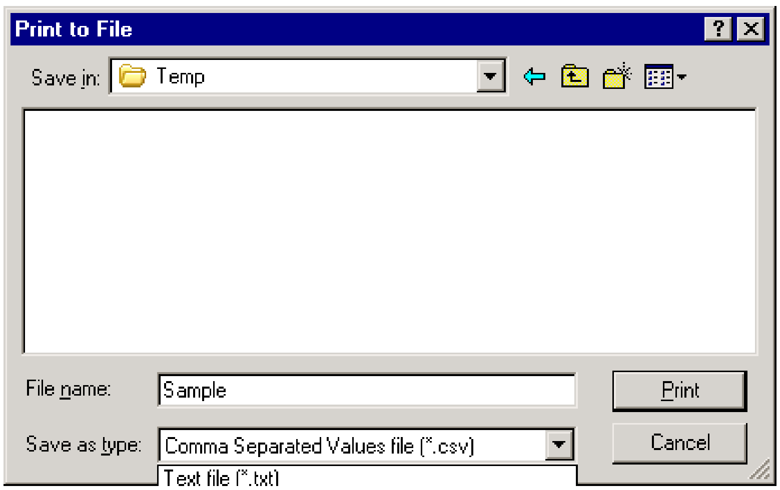


.Print Model (Ctrl+P).

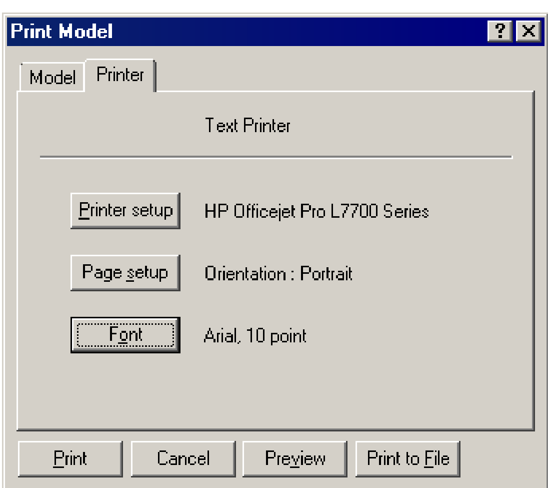
The Print command, flexible and customizable, is specific to each window. Here in Layout, you can print the model input data only (under the Model tab)to a printer, a PDF file (with a free PDF utility or Adobe® Acrobat installed), and change printer settings (under the Printer tab). You can customize the look of your report by changing the font, page sizes or margins.



Data can be output to either a text file or a CSV (comma separated values) file that can be opened in a spreadsheet software (e.g., MS-Excel).



You could change the selected printer and customize other settings in the Printer tab (shown next).



.Save (Ctrl+S).

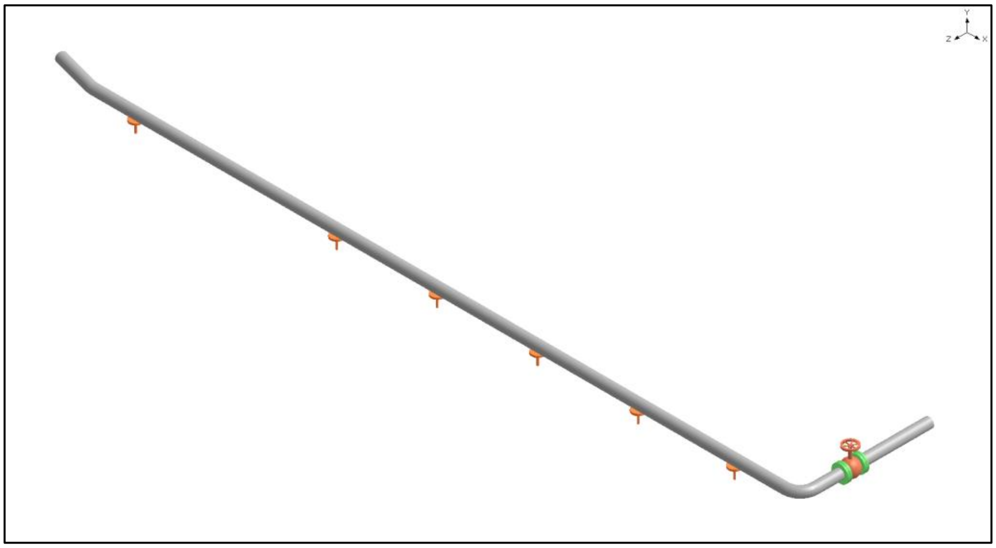
This command saves model data to a .mod file format. It will also issue warning(s) if errors are found in the layout data.

.Analyze (F12).

Once you are done modeling, use this command to analyze the model. At the end of analysis, you are asked whether you want to "View results?" Click on OK to see results or Cancel to return to the main window.

Shortcut "F12" will work only when the command "Analyze" is active in the "File" menu.

CAEPIPE may occasionally issue the message "Stiffness not positive definite". This error message appears when the model has a free rigid body movement in one or more degrees of freedom. To avoid this error message, make sure that any portion of the pipe stress model does not have free rigid body motion in any of the six (6) degrees of freedom (3 translational and 3 rotational).



For example, the Branch line shown in the snapshot above allows free rigid body motion for the whole branch line in at least one degree of freedom. Hence, CAEPIPE will issue the error message as given above when the model is analyzed.

A quick fix is to add a support or modify an existing support such that it prevents the free body motion.

For example, the free body motion of the branch line shown above can be prevented in two ways as given below.

a. Replace the Limit Stop located at the beginning of the Branch line with an Anchor.

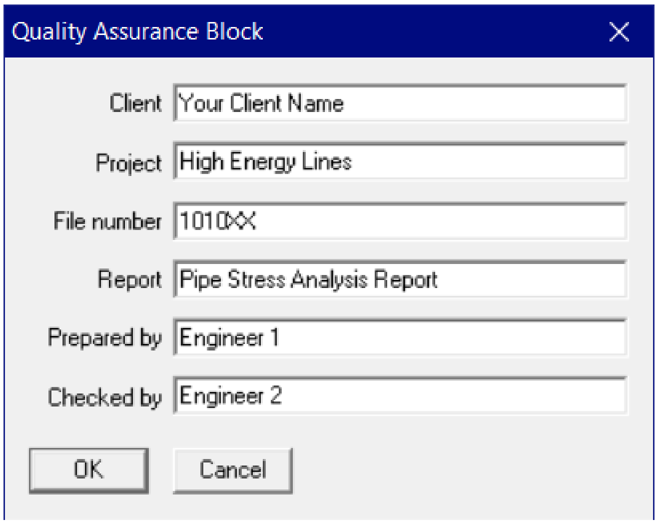
b. Replace the Limit Stops located at the Start and End of branch with Guides.

Note:

Command "Analyze" in the "File" menu will become inactive when the model is not edited and ".res" file is available in the directory where the stress model ".mod" is stored.

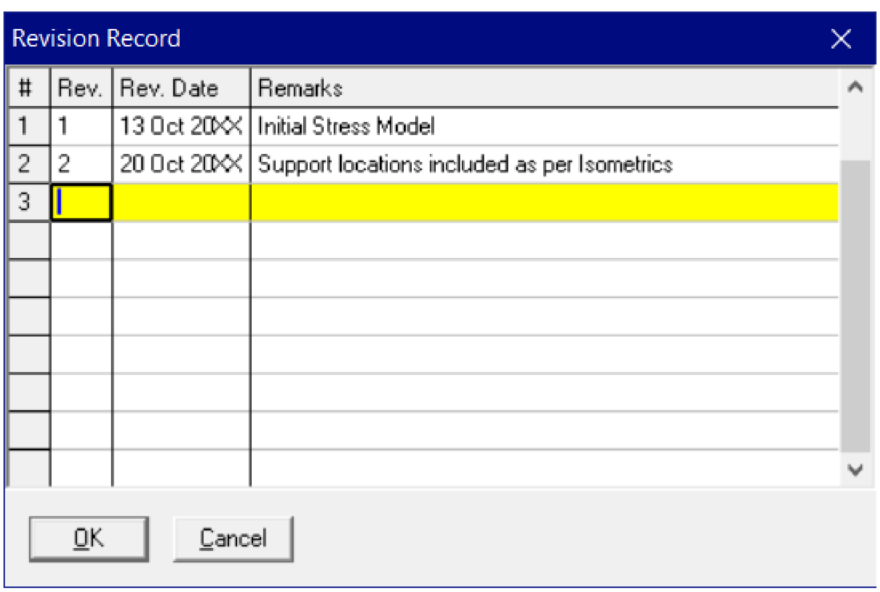
.QA Block.

Type in project-specific information here, printable only from the Results window. You cannot change this information after analysis in the Results window. If you like to modify any information here, do it from the Layout window. You will then need to reanalyze the model because CAEPIPE will have deleted the existing results file for this model since the old data *may* no longer apply.

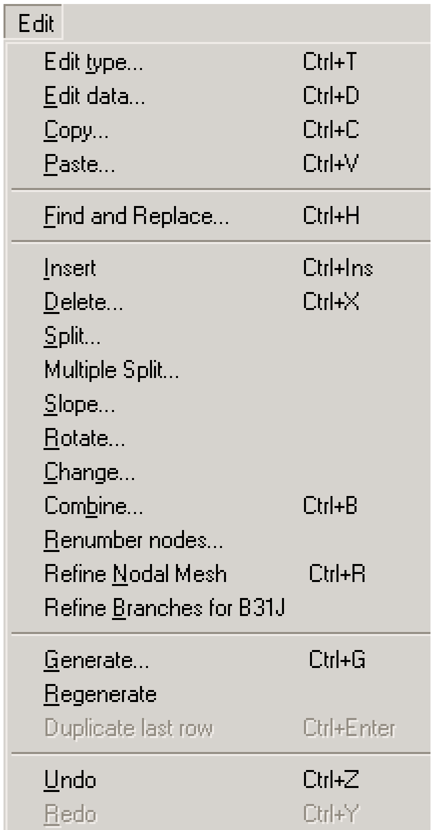


.Revision Record.

Store and print the details of the revisions carried out in this stress model for future reference.



Edit Menu



Using this menu, you can edit properties for elements (under Type column) and data items (under Data column), insert and delete rows, split an element, rotate and renumber parts of the model, change a few properties like friction coefficient for a range of rows, and generate copies of existing rows.

.Edit type (Ctrl+T).

Double click on the element name in the Type column or press Ctrl+T to edit properties (if applicable) of an element (bend, valve, etc.). The appropriate dialog is shown.

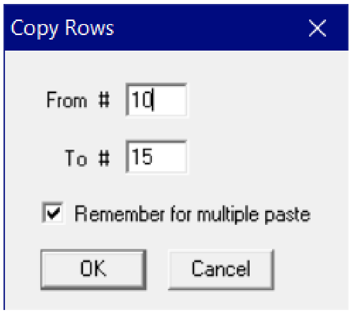
.Edit data (Ctrl+D).

Double click on the Data item or press Ctrl+D to edit properties (if applicable) of the item (flange, anchor, etc.). The appropriate dialog is shown.

.Copy (Ctrl+C).

You can copy a specific item on a row such as Material, Data or Type column entries, or an entire row or a block of rows, and Paste all, or individual parts of the row. To copy a desired row, move the cursor to that row and press Ctrl+C (or select Edit menu > Copy command). The copy command will copy all information on that row irrespective of the specific cursor location within that row.

Copy of multiple rows can be done through Layout Window > Edit > Copy (and NOT through Ctrl+C). This feature can be performed ONLY in Layout window and NOT in List windows such as Materials, Loads, Section properties, Flange, Valve, etc. Since this is a block operation, you need to specify (a range of) row number(s). Of course, you could accept the default row shown (which copies only one row).



Turning ON the option "Remember for multiple paste" will remember the rows copied for pasting the elements multiple times.

Copy command available through "List window > Edit > Copy" will remember the information from a row copied for multiple pasting ONLY for the Element types namely Valve, Bellow, Slip Joint, Hinge Joint, Ball Joint and Tie Rod, and for the Data types namely Flange, Guide, Limit Stop and Skewed Restraint.

.Paste (Ctrl+V).

Once you have copied a row through Ctrl+C, move your cursor to the location where you want to paste the copied information and press Ctrl+V (or select Edit menu > Paste). If you want to paste the entire row, move the cursor to the Node column of the row where you will be pasting. To paste an individual part of a copied row, move the cursor to the specific column within the row where you wish to paste the specific datum. The copied information is available for only one operation. Also, note that when an entire row is pasted, it will overwrite the existing row (if present). See Example 1 for further details. To update / refresh the Graphics window after paste, save the model and use the command "Redraw"available through Graphics window > View.

If you have copied a segment consisting of multiple rows through Edit menu > Copy (and NOT through Ctrl+C), move your cursor to the location where you want to paste the copied information and press Ctrl+V (or select Edit menu > Paste). See Example 2 below for further details.

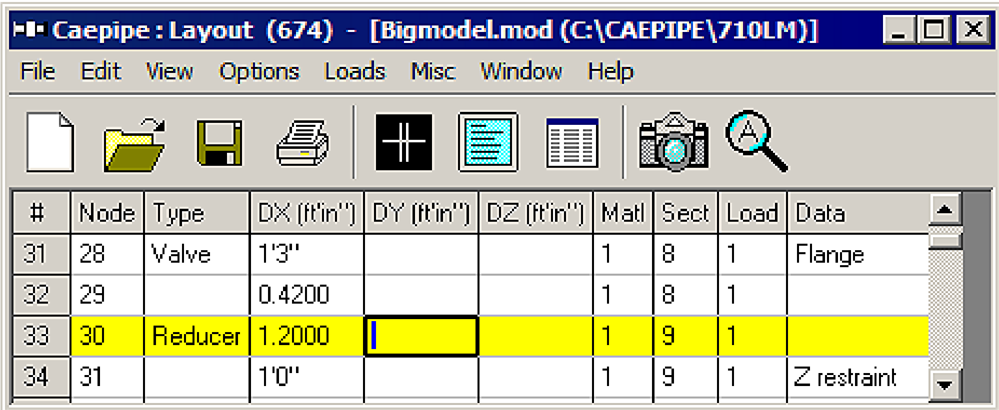
Note:

If you wish to copy and paste a row multiple times, then invoke Copy command through "Edit > Copy" and enter the From # and To # as the desired row # and turn ON the option "Remember for multiple paste".

Node numbers for the elements pasted will be generated based on "Node increment" defined through Layout Window > Options > Node increment.

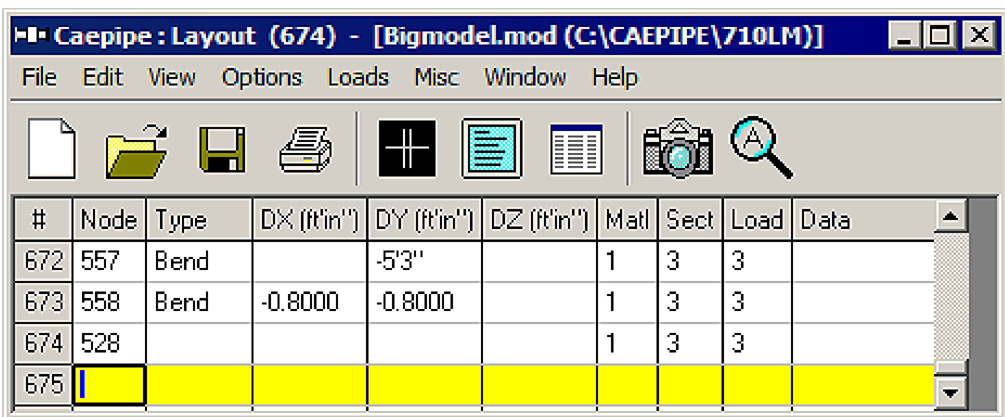
Node number for "Location" element is assigned the same node number as the preceeding element whose type is other than "Comment".

Example 1: Copy and Paste a Single Row



To copy the row at #33 (30, Reducer, 1.2ft, , ,1, 9, 1,), you may position the cursor anywhere on that row before you invoke the Copy command. But, to paste ALL of this information, you must *position the cursor under the Node column of a (new) row*.

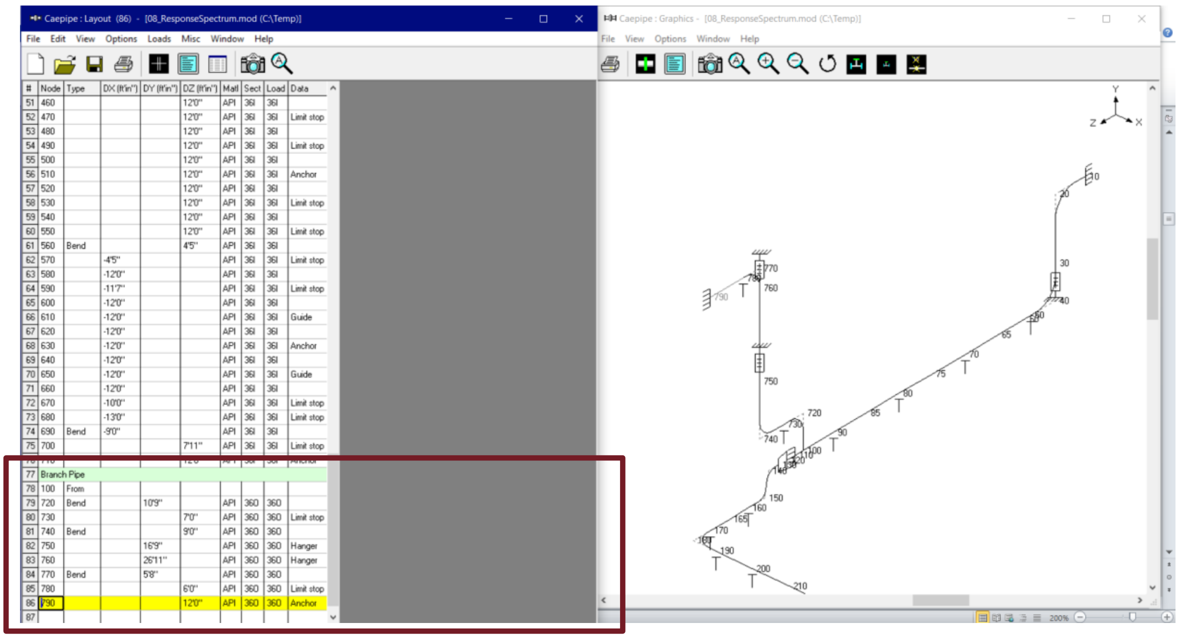
If you want to paste, however, only the type 'Reducer' then place cursor under Type, only for DX of 1.2', place cursor under DX, and so on.



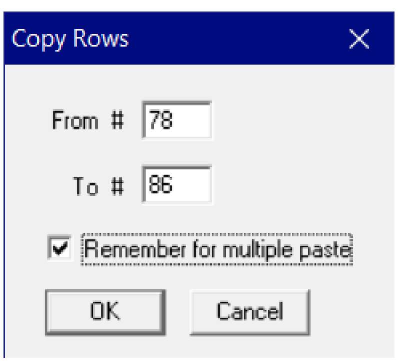
Example 2: Copy and Paste a group of elements (multiple rows) including their Data in Layout

This example shows how to Copy a branch pipe starting at Node 100 and ending at Node 790 and Paste the same at Nodes 65 and 75. Sample model used in this example can be downloaded from here.

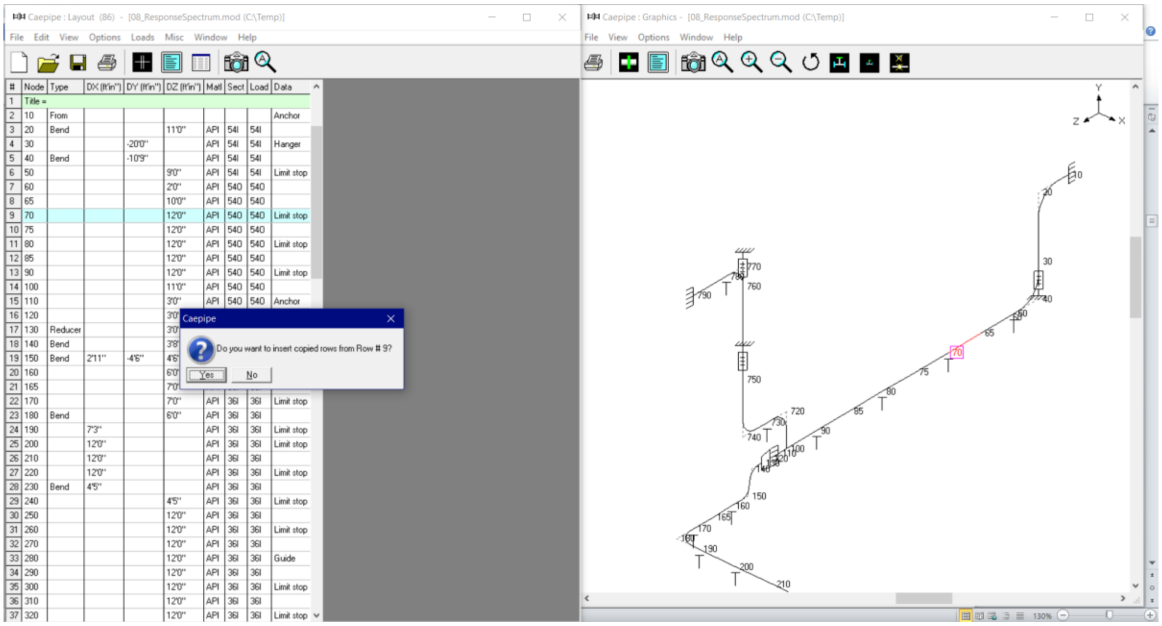
1. Navigate in the layout window where the brach pipe layout information is defined as shown in the figure below.



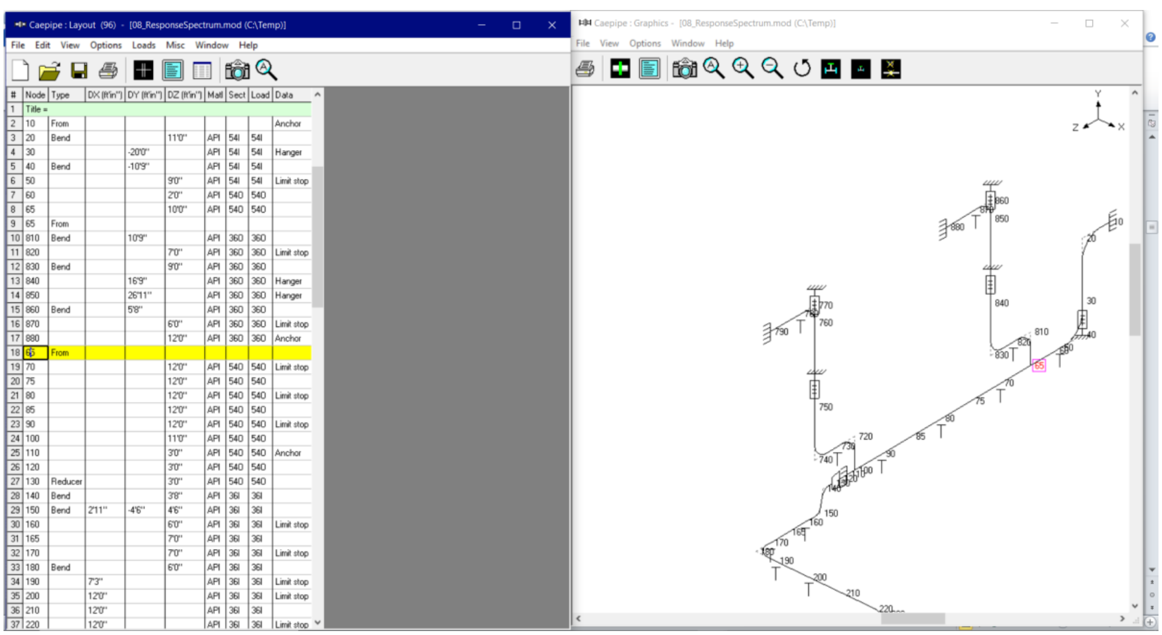
2. Invoke the Copy command through Layout Window > Edit (and NOT using the shortcut Ctrl+C). Enter the From # as 78 and To # as 86. Turn ON the option "Remember for multiple paste" to paste the segment multiple times in the layout.



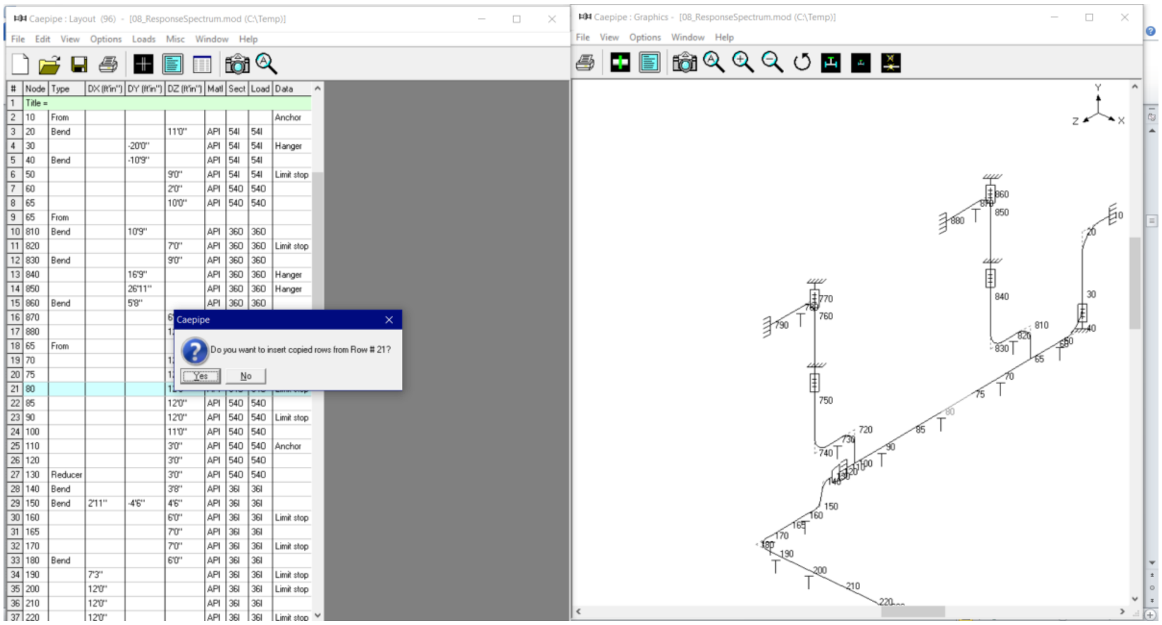
3. To Paste the branch pipe copied at Node 65, position the cursor at Row # 9 (after Node 65 definition) and press Ctrl+V. When done, you will see a message as shown below as you are inserting the copied rows in the middle of the layout. Press "Yes" to accept.



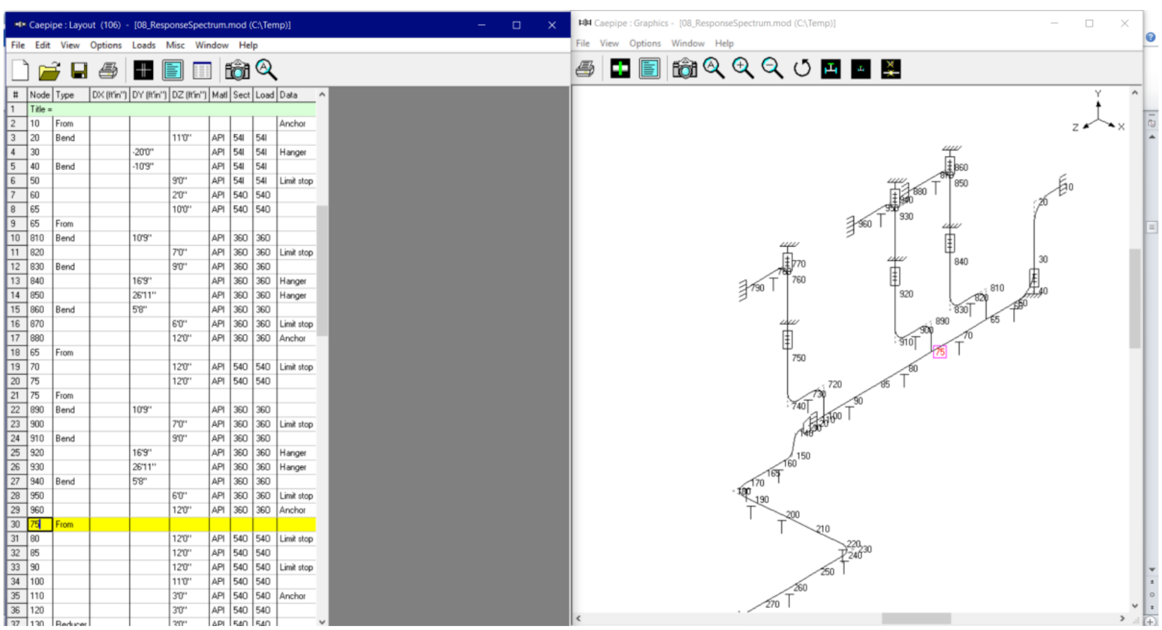
4. In the figure shown below, you will observe that the new branch is inserted with Node numbers from 800 through 880 (rows 9 through 17). Now, to position the branch pipe at Node 65 properly, change the Node Number at row # 9 as 65 and insert a new row at row # 18 with Node number as 65 and Element type as "From" at both row # 9 and row # 18 as shown below.



5. Similarly, to Paste the branch pipe at Node 75, position the cursor at row # 21 (after Node 75 definition) and press Ctrl+V. Accept by pressing “Yes”.



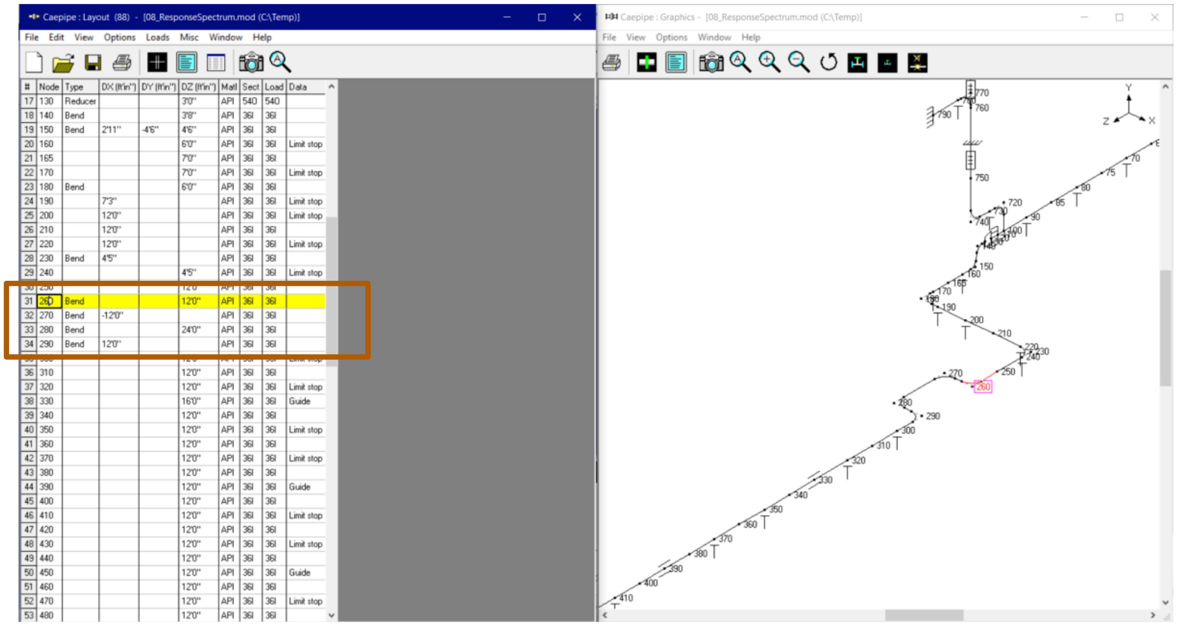
6. Now, to position the new branch at Node 75, change the Node Number at row 21 as 75 and insert a new row at row # 30 with Node number as 75 and Element type as “From” for both row # 21 and row # 30 as shown below.



Example 3: Copy and Paste a Loop multiple times in Layout

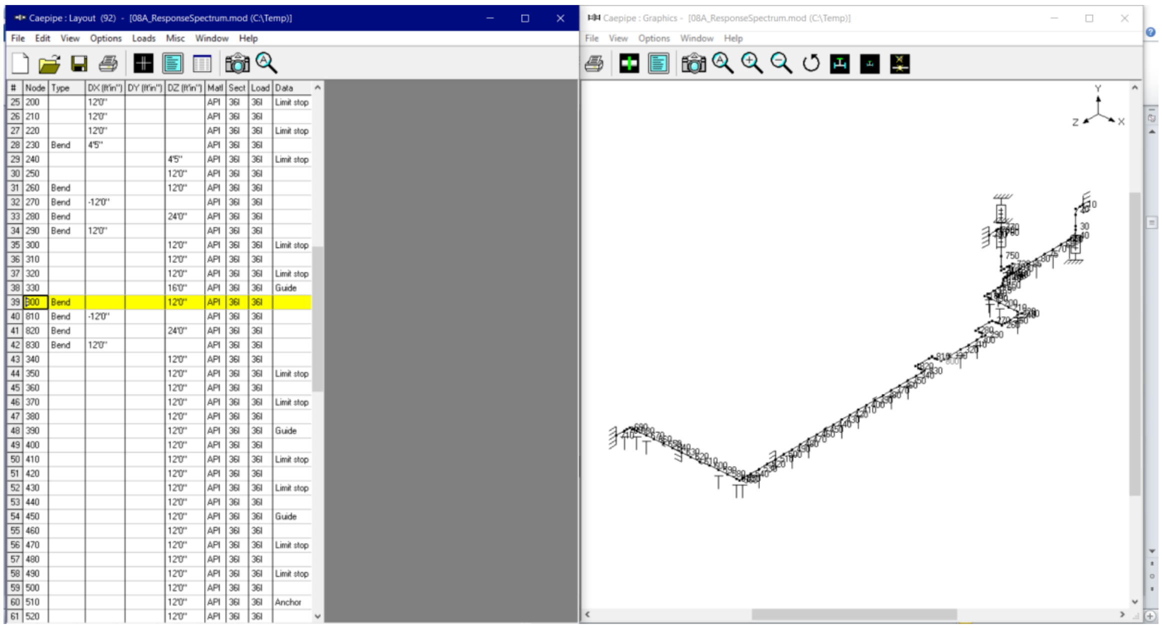
This example shows how to add a Loop to the existing layout and Copy & Paste the same loop multiple times (say at Node 340 and at Node 400). Sample model used in this example can be downloaded from here.

1. Edit the existing layout and add a loop at Node 260 (row 31) as shown below.

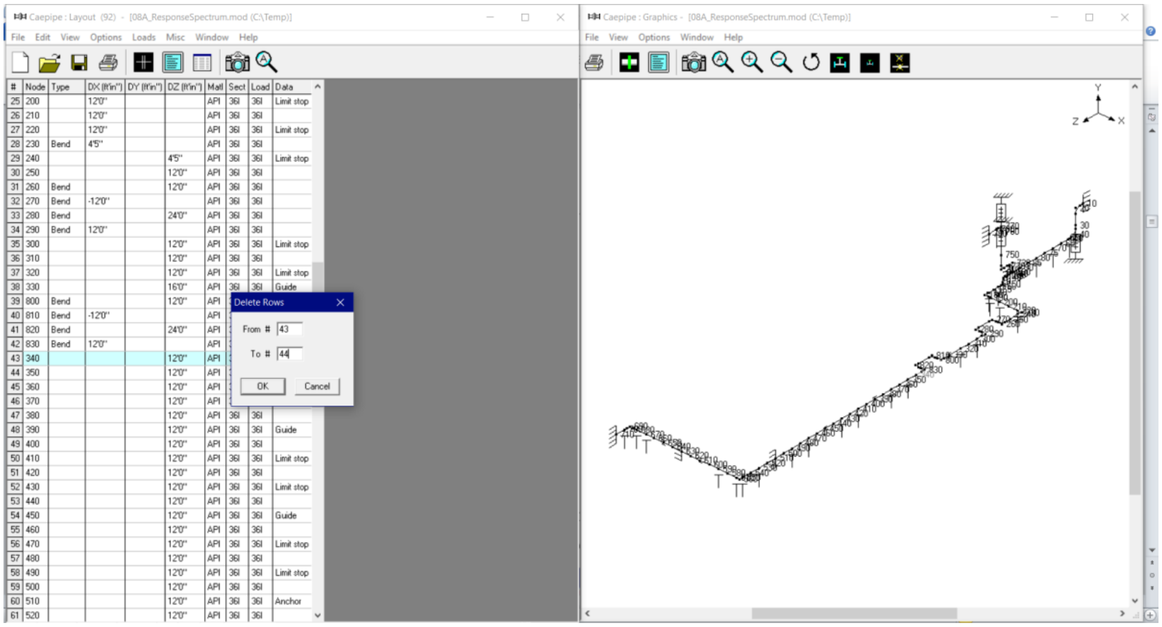


2. Invoke Copy command through Edit menu > Copy. Enter the From # as 31 and To # as 34. Turn ON the option "Remember for multiple paste".

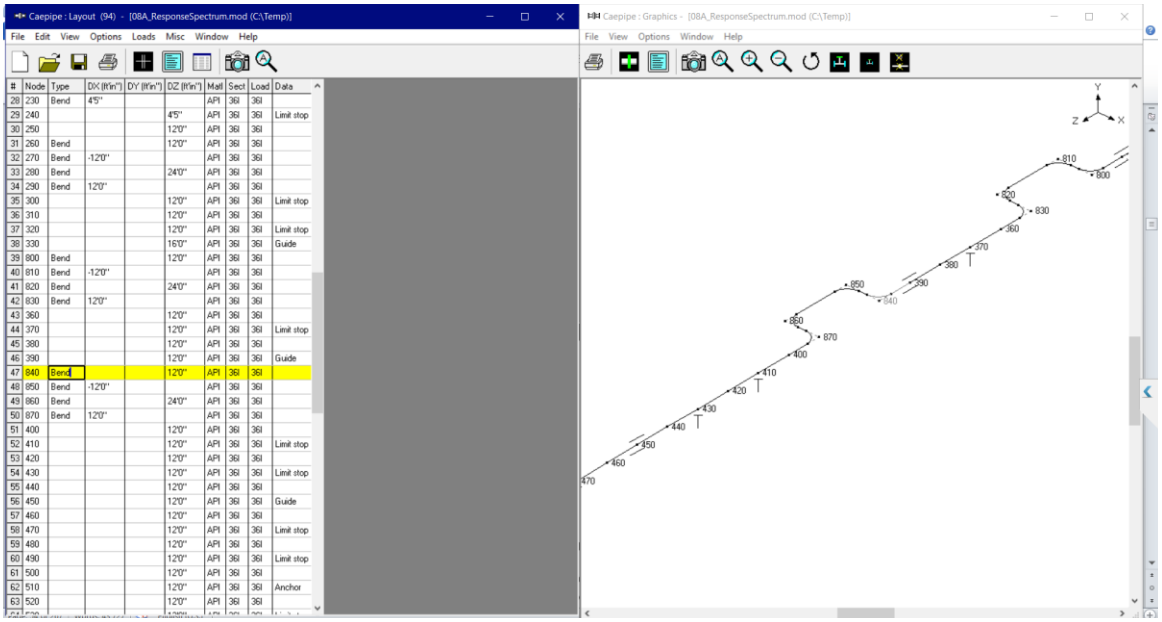
3. Now, to insert the loop at Node 340, position the cursor at row 39 and press "Ctrl+V".



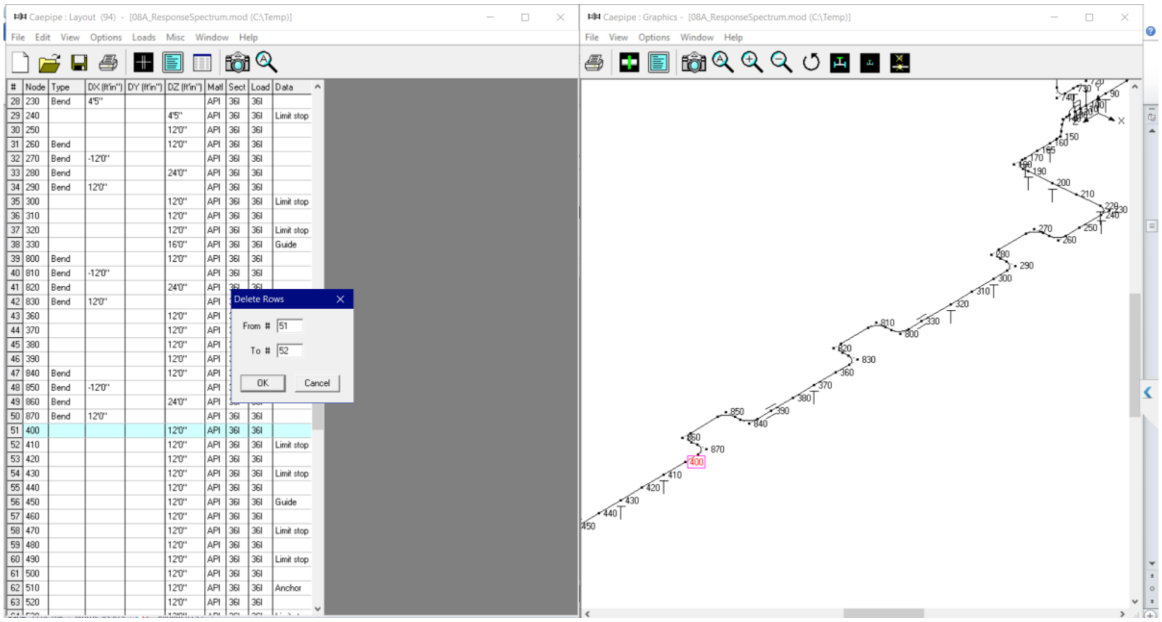
4. Delete the rows 33 and 34 (Nodes 340 and 350) as they are already replaced with loop through Edit > Delete as shown below.



5. Similarly, to copy the same loop at Node 400, position the cursor at row # 47 (Node 400) and press "Ctrl+V".

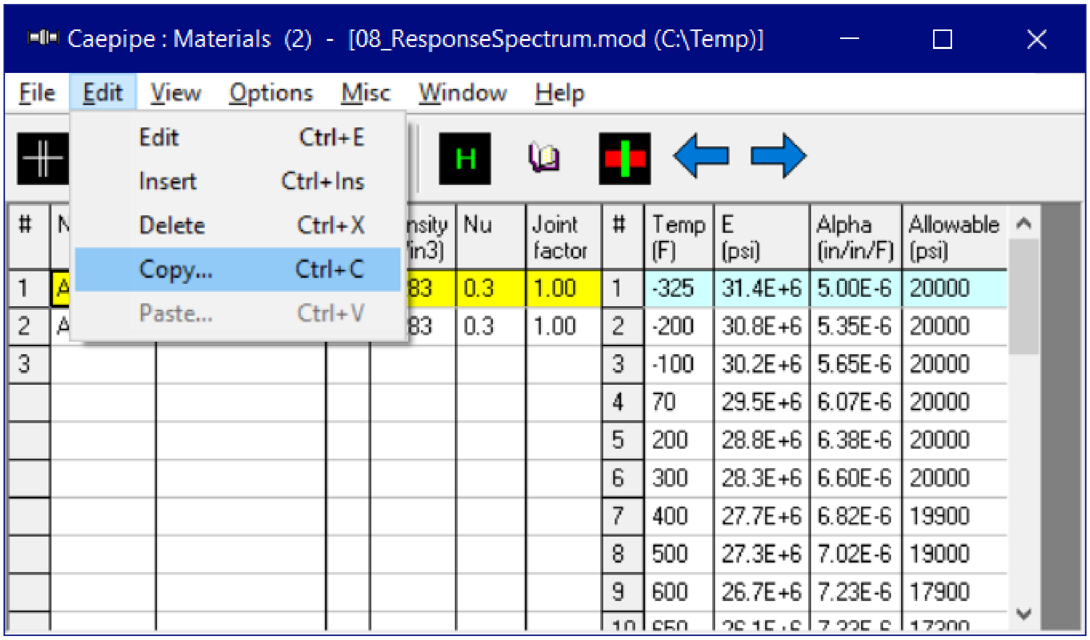


6. Delete the rows 51 and 52 (Nodes 400 and 410) as they are already replaced with loop through Edit > Delete as shown below.

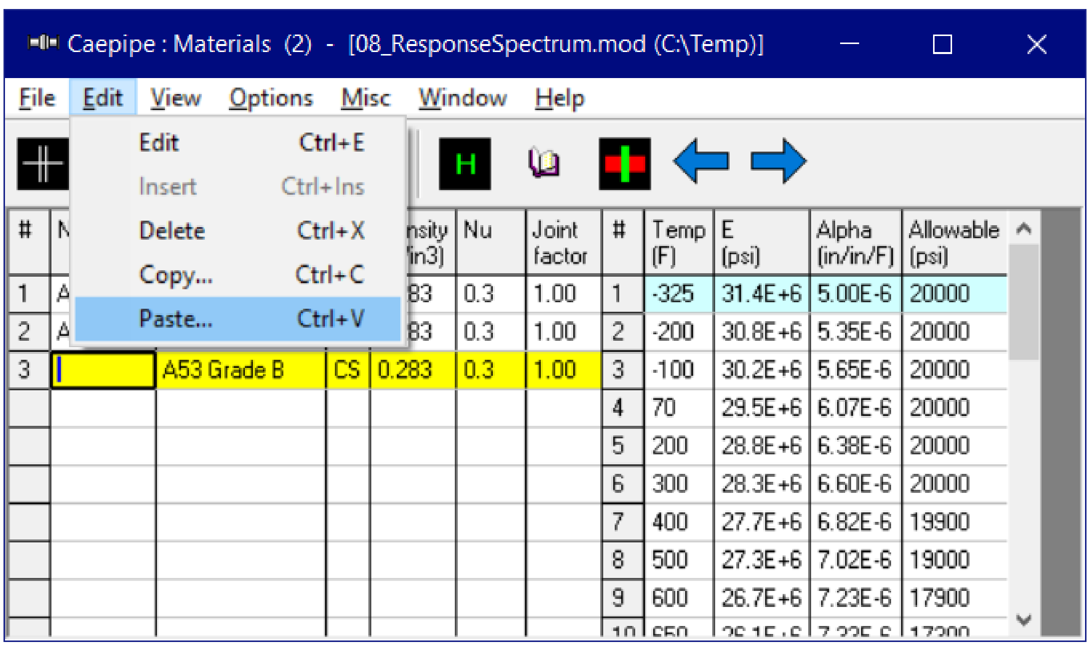


Example 4: Copy and Paste a Section / Material / Load via List Window

To copy a Section / Material / Load, position the cursor in the required row and invoke the Copy command through List window > Edit > Copy or press "Ctrl+C".

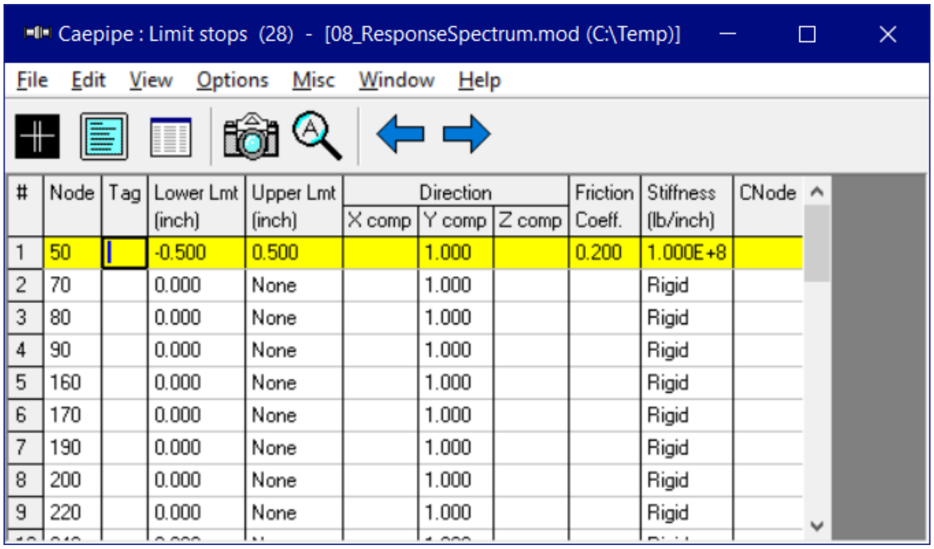


Now, to paste the row copied, you must position the cursor at an empty row where you wish to copy and invoke command through Layout Window > Edit > Paste or press "Ctrl+V". When done, enter the desired material reference name (up to 5 characters) in the column Name.



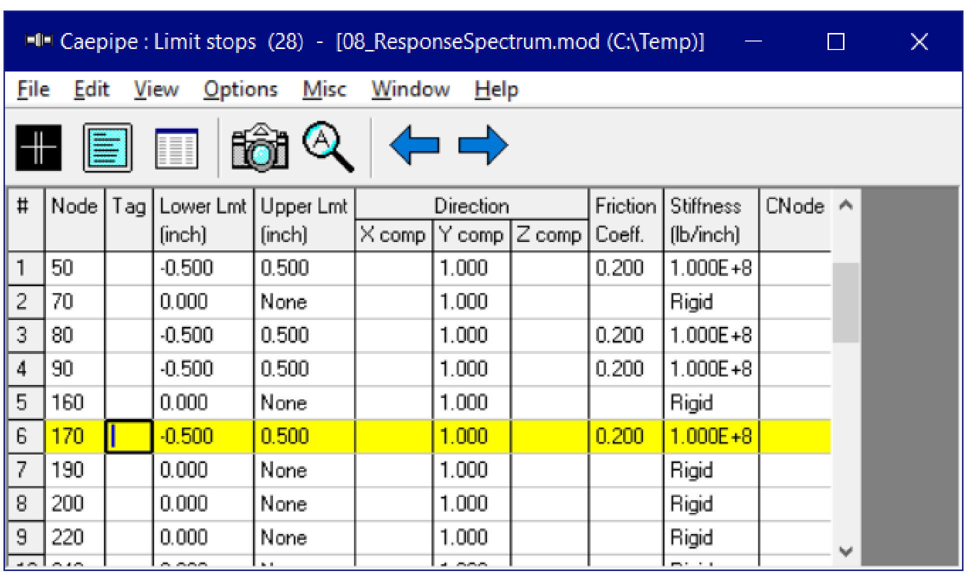
Example 5: Copy and Paste a row in List Window multiple times

For Element types Valve, Bellow, Slip Joint, Hinge Joint, Ball Joint and Tie Rod and Data types Flange, Guide, Limit Stop and Skewed Restraint, Copy command available through "List window > Edit > Copy" will remember the data copied from a row for pasting the same multiple times. As an example, below figure shows how to Copy and Paste a Limit Stop data multiple times in a List window.



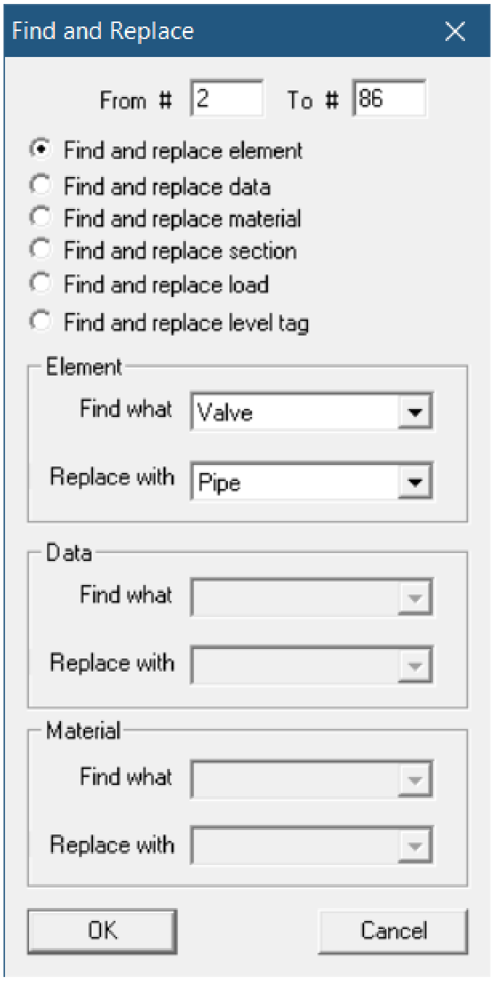
To Copy data of a Limit Stop at Node 50, from the List window for Limit Stop, position the cursor at row # 1 and invoke the Copy command through List window > Edit or press "Ctrl+C".

Now, to paste the Limit Stop data copied multiple times, position the cursor at the required rows (row # 3, row # 4, row #6) and invoke Paste command through List window > Edit or press "Ctrl+V". See figure given below for details.

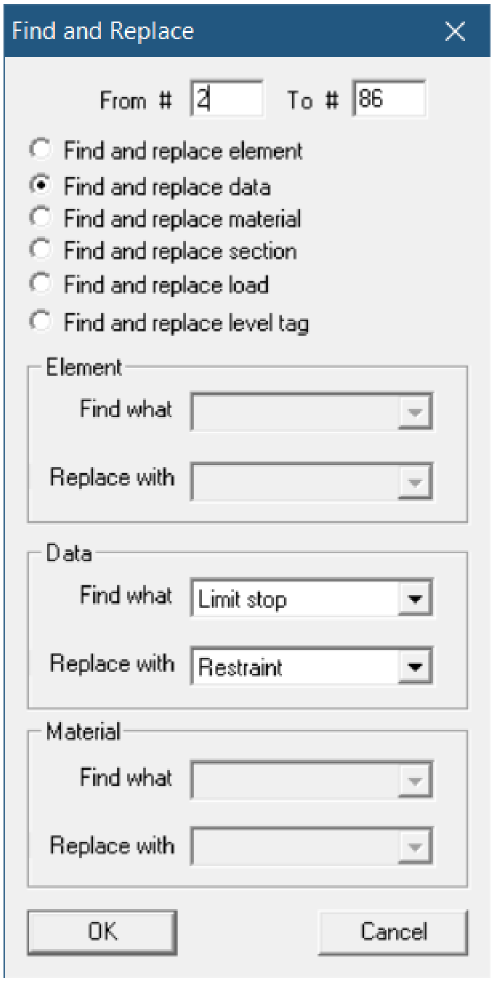


.Find and Replace (Ctrl+H).

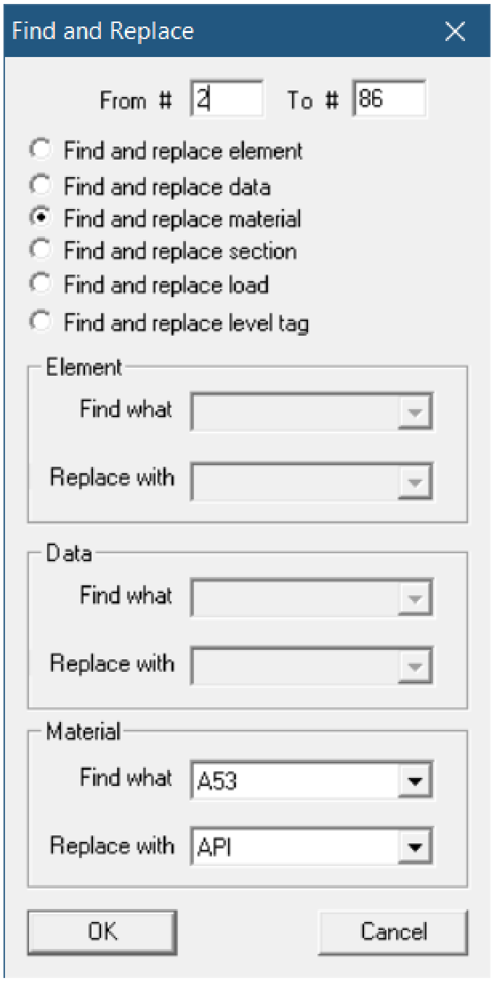
Find and replace Element / Data types, Material, Section, and Load by selecting the Find and Replace command from the Edit menu or press Ctrl+H. The Find and Replace dialog box is shown below.



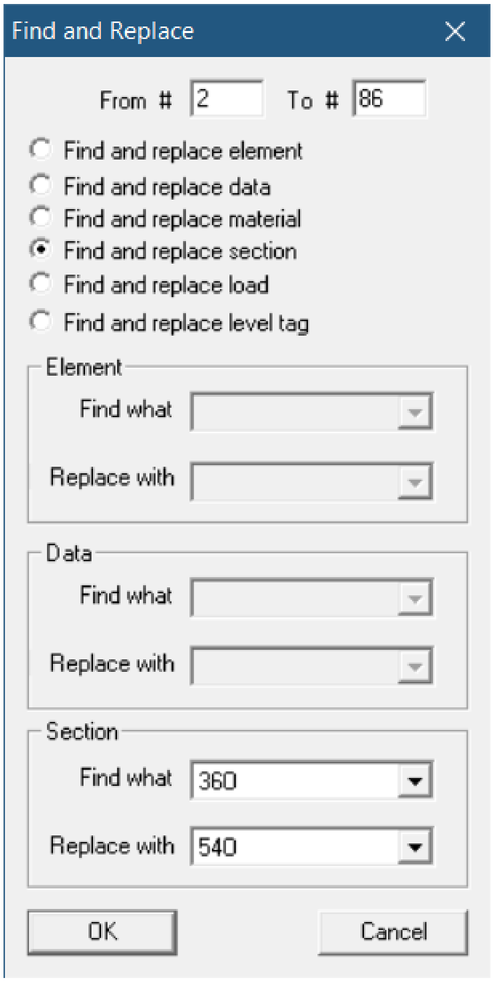
In the shown dialog above, all Valves will be replaced with pipes between rows 2 and 35. Use this command, for example, to change all LR elbows to SR elbows. Note that while replacing element(s), you cannot leave the "Replace with" field blank.



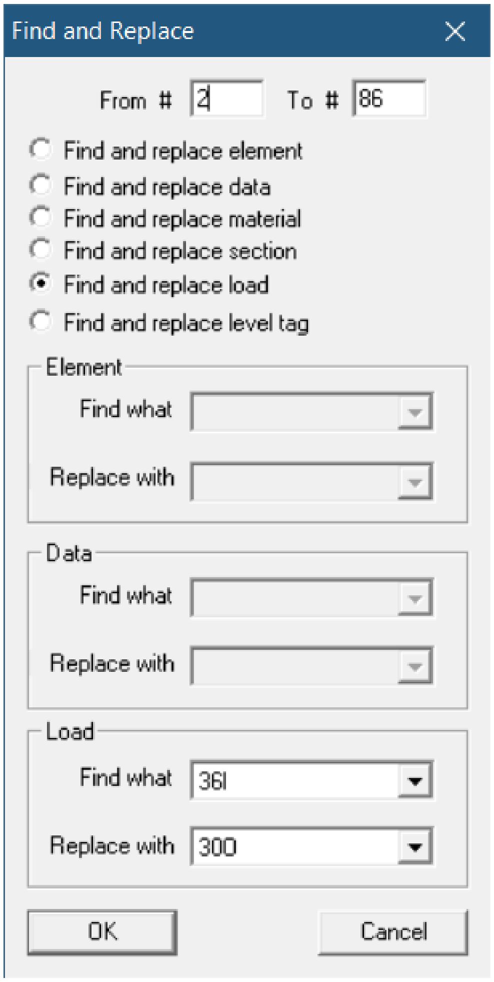
You could change to different support types while evaluating different "what-if" conditions, or *remove* a specific support type from the model (select "blank" in the Replace with field).



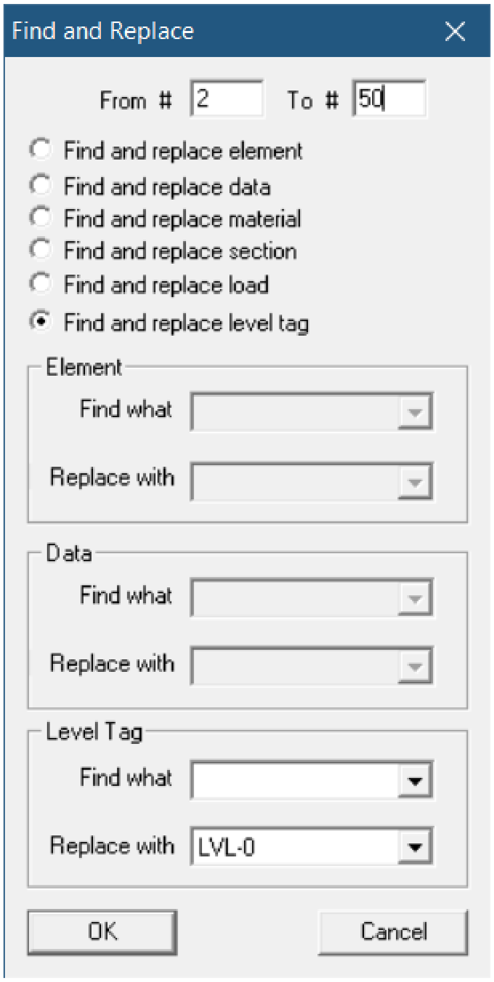
You could change to a different material type while evaluating different "what-if" conditions.



You could change to a different section while evaluating different "what-if" conditions.



You could change to a different load while evaluating different "what-if" conditions.



You can assign Level Tag to supports when the model has spectrum with multiple spectrum level defined. For example, dialog shown above will find all supports that has no level tag defined (BLANK) in the range 2 to 50 and replace with level tag "LVL-0". Similarly, leaving the Replace field BLANK with Find what field defined will remove level tag assigned to all supports.

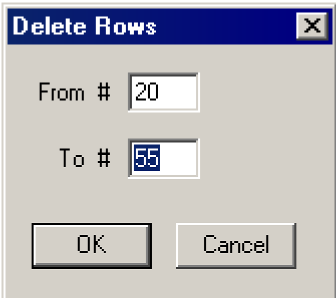
.Insert (Ctrl+Ins).

Use Ctrl+Ins to insert an empty row above the present highlighted row in the Layout window. This command is also available from the context menu, by right-clicking in the Offsets (DX, DY, DZ) columns.

.Delete (Ctrl+X).

Use Ctrl+X on a row to delete it. To delete a range of rows, you must select the Delete command from the Edit menu, and then type the row numbers to set the range for deletion. If you want to delete till the end of the model, either type in a big number for "To #" such as 9999 or type in the (total) number (of rows) shown in the title bar in parentheses.

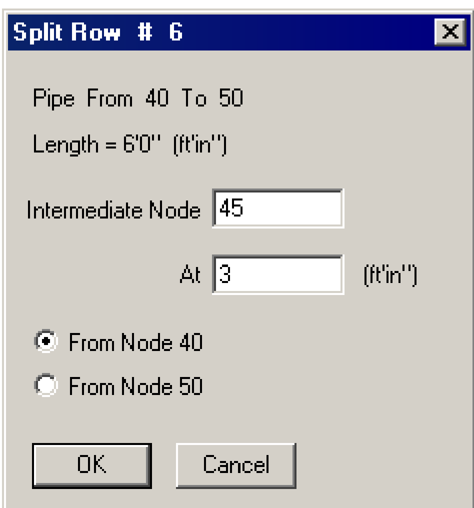
This command is also available from the context menu (by right clicking), in the Offsets (DX, DY, DZ) columns.



.Split.

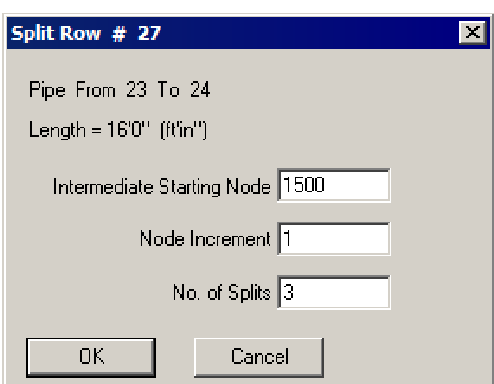
Use this command to split a straight element or a curved section of a bend into two parts. This can be useful to create nodes for supports, applying forces at intermediate locations or introducing intermediate mass points. To split curved sections of bends (including jacketed bends), specify intermediate nodes at desired angles in the bend dialog. This command is also available from the context menu by right clicking on the Offsets (DX, DY, DZ) columns.

When you have a long sloping pipe, you can create one long element with the slope built into it and then use the Split command to break it down into smaller elements. Or, of course, you could use the Slope command.



.Multiple Split.

CAEPIPE can split an element into multiple segments.

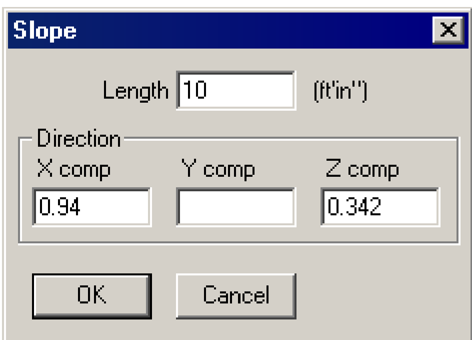


Assume that you wanted to split an existing element into 3 *equal* segments. This command will let you define the intermediate starting node # (with an increment) and the number of splits you want (see dialog above). If a non-pipe element (such as a bend, valve, reducer, etc.) is split, multiple pipe elements and anon-pipe element will be generated as a result of this operation.

.Slope.

You can specify the slope of the line you want to model for an element, in terms of its direction cosines (DCs).

This command calculates the offsets automatically for a sloping element (that does not align with one of the major global axes). When a pipe slopes (or is routedat an angle) with respect to the global axes, it becomes necessary to calculate the offsets (DX, DY and DZ) using those faithful sine and cosine functions. But, therein lies the problem. Sometimes, calculations get tricky and time-consuming.



Let us see a few examples.

Example 1: A 20° line in the XZ plane, 10 feet long

Method 1:

Calculate the offsets:

DX = 10 cosine (20) = 9.4 (ft.), DZ = 10 sine (20) = 3.42 (ft.). Simple!

Method 2:

Find direction cosines (DCs) for the line (vector). Direction cosines are simply cosines of the angles the vector makes with the global X-, Y- and Z-axes. In this case, A=20° (X), B=90° (Y), C=70° (Z). Cosines of these angles are: X comp=0.94, Y comp=0, Z comp=0.342.

Now, position the cursor on the sloping element row, right click under DX/DY/DZ. Select Slope from the menu, enter these numbers into the X comp, Y comp and the Z comp fields of the Slope dialog that opens, and type the length, 10 ft. Press Enter. CAEPIPE calculates the respective offsets using the DCs you input.

Once you input the slope, follow this pipe with other elements down the line by inputting different lengths while you retain the same direction cosines.

Example 2: A 40° line in the X(-Y) plane

Method 2:

Find direction cosines, if the line is at 40 deg. to X-axis in X(-Y) plane, it makes 50° with the (-Y) axis. So, the DCs are: cos (40°) = 0.766, cos (50°) = 0.643, So, X comp = 0.766, Y comp = -0.643, Z comp = 0.0. Type these DCs and a length into the Slope dialog. Press Enter.

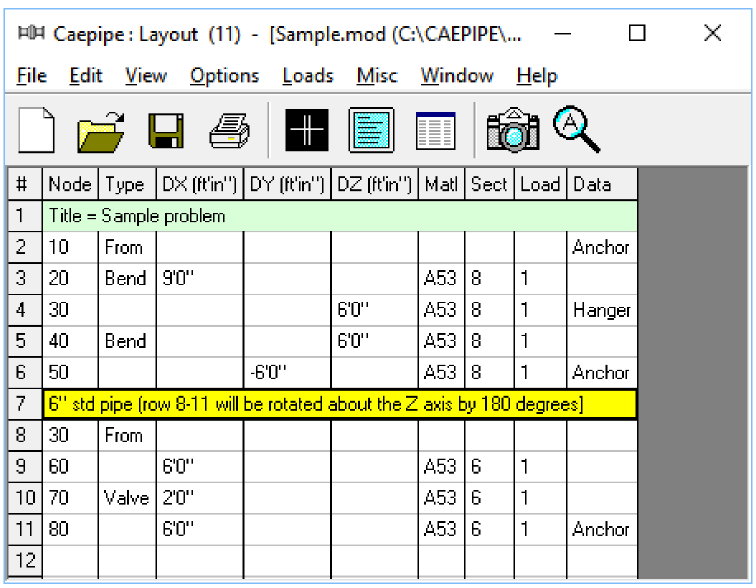
More information about Direction cosines is available under the section on Direction in the Technical Reference Manual.

.Rotate.

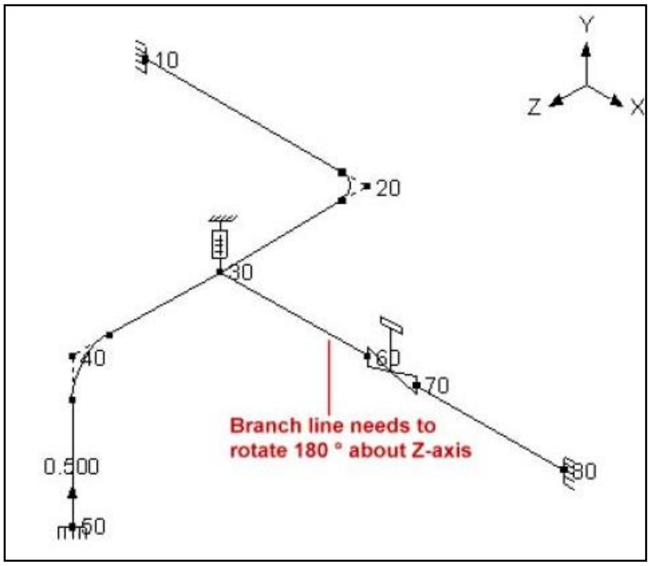
You can rotate a model or a part of it about the global axes. CAEPIPE will adjust the offsets for new orientation.

Example

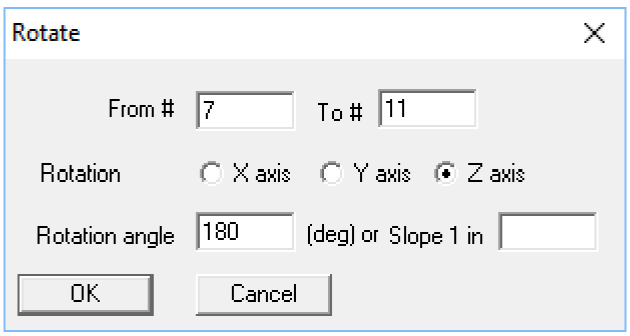
Let us rotate the branch line of the familiar Sample model. The branch line is between nodes 30 to 80 (rows 8 to 11 in the Layout window). Say, we wanted to route this branch line in the –X direction (i.e., it needs to rotate 180°).



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe (row 8-11 will be rotated about the Z axis by 180 degrees)								
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									



Select Rotate from the Edit menu and type in the shown values.



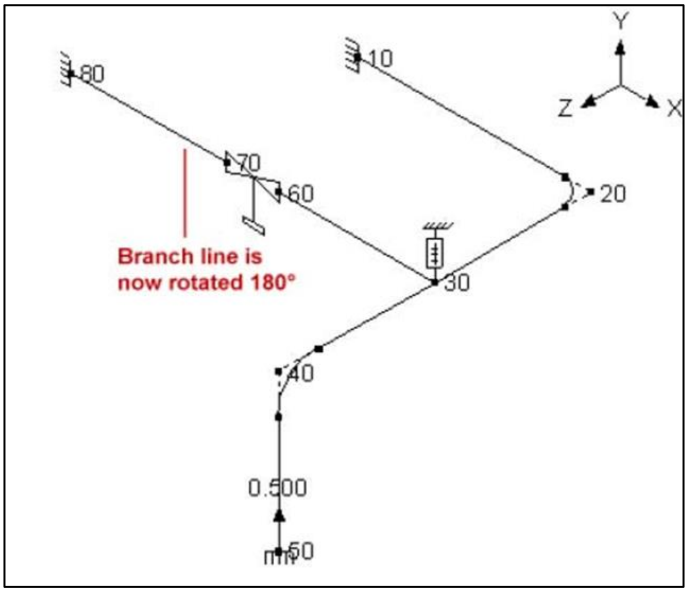
Notice the rotated offsets (now they are in the -X direction).

Note: The valve additional weight is shown in the negative direction because of the rotation. To make it positive, edit the Valve (Ctrl+T) properties and change its DY offset from -18 to 18 inches.

Caepipe : Layout (11) - [Sample.mod (C:\CAEPIPE\...

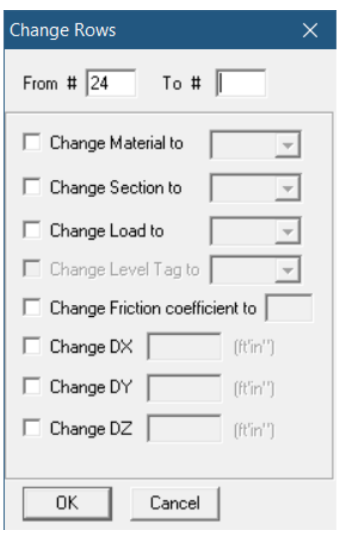
File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe (row 8-11 will be rotated about the Z axis by 180 degrees)								
8	30	From							
9	60		-6'0"			A53	6	1	
10	70	Valve	-2'0"			A53	6	1	
11	80		-6'0"			A53	6	1	Anchor
12									



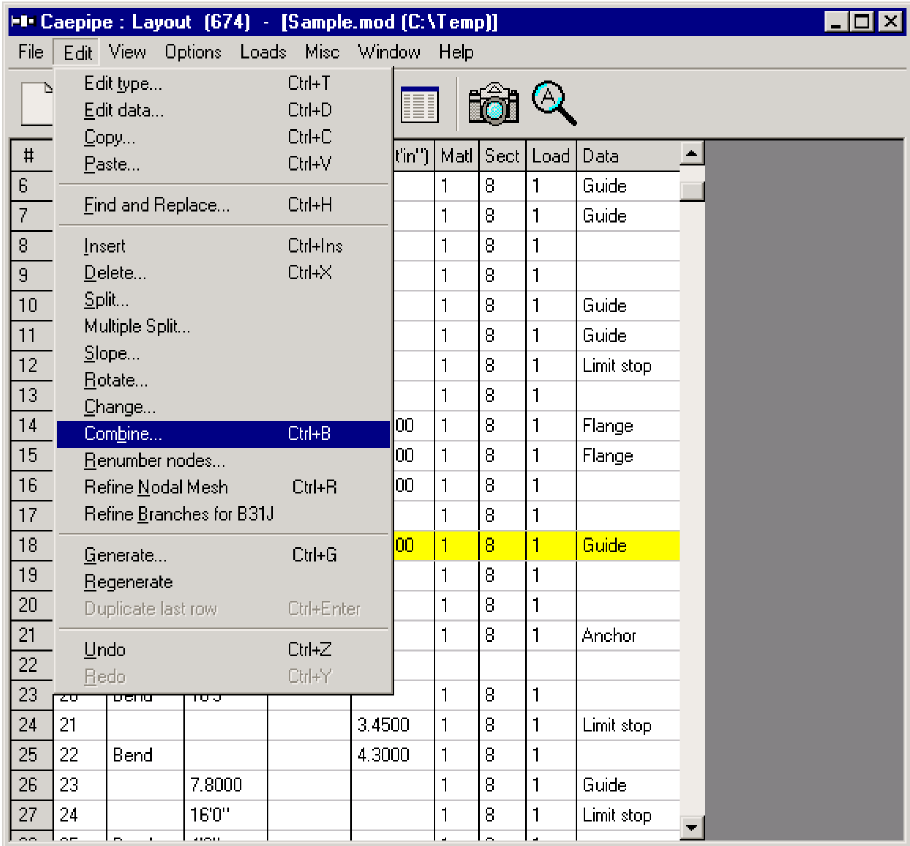
.Change.

Use this command for block operations such as changing friction coefficient for all guides and limit stops within a range of rows, or changing the material, section or load for a range of rows or level tags for all supports (available only when spectrum and spectrum level defined) or span length (i.e., DX, DY and/or DZ between adjacent nodes) for a range of rows. You are asked for the range when you select this command from the Edit menu.

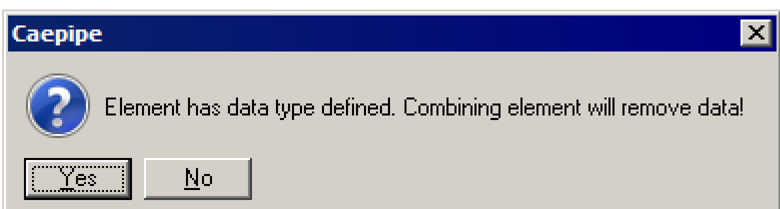


.Combine (Ctrl+B).

You can combine current PIPE/BEND elements with previous PIPE elements.



If you have a Data type (such as a support) defined on one of the "to-be-combined" rows, CAEPIPE will alert you to the fact that you will lose that data type definition during the Combine process.



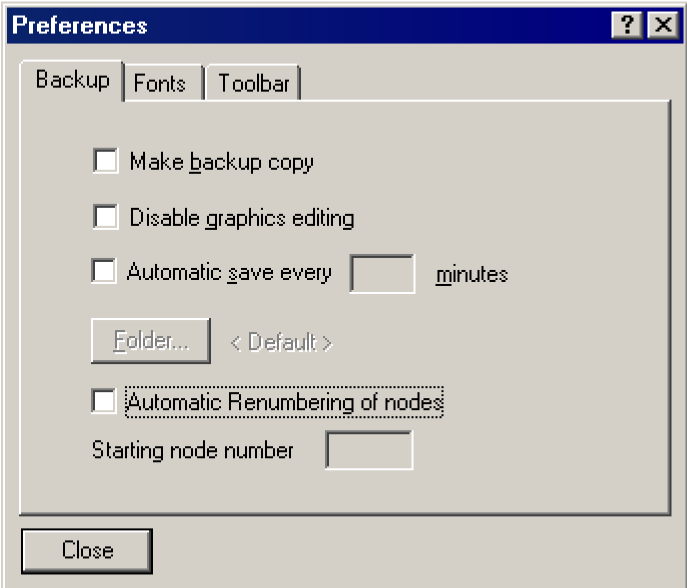
If you accidentally combine two elements, you can always UNDO the operation.

.Renumber Nodes.

You can renumber existing node numbers in your model using this handy feature;really useful when you have a big model and want to adopt a consistent node numbering system after multiple edits. You can renumber the whole or parts of the modelkeeping in mind that the greatest node number in a model cannot exceed 99,999.

Renumbering a segment of a piping system also renumbers all those nodes of the segment that appear as "Connected to" nodes in the remaining segments of the piping system.

Before using this feature, please make a copy of your model and work on the copy. Note: This feature is unrelated to "Automatic Renumbering of Nodes" found in the main window>Preferences.

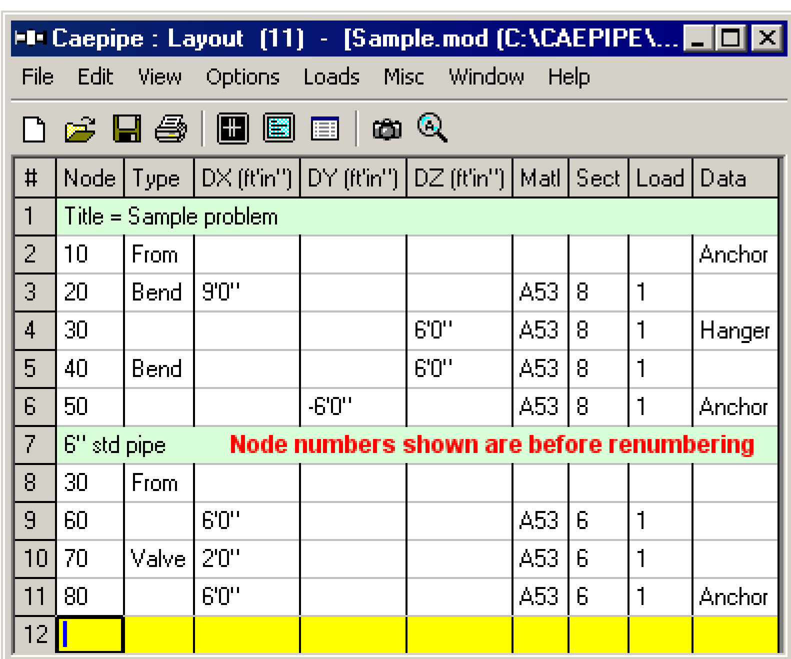


Automatic Renumbering in the Main window > Preferences

When this feature is turned on, deleting row(s)in Layout triggers an automatic (and instantaneous) renumbering operation, while the topic being discussed here allows you to renumber selectively for a range of rows.

Example

Assume that for the CAEPIPE Sample model, you wanted to change the node numbers for the header to begin from 100, and the branch line to begin from 1000. This can be easily done as shown next:

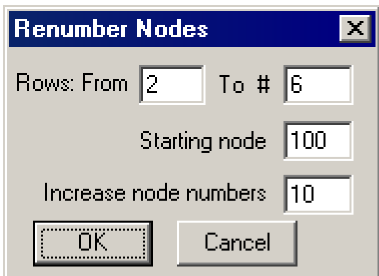


#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Sample problem								
2	10	From							Anchor
3	20	Bend	9'0"			A53	8	1	
4	30				6'0"	A53	8	1	Hanger
5	40	Bend			6'0"	A53	8	1	
6	50			-6'0"		A53	8	1	Anchor
7	6" std pipe	Node numbers shown are before renumbering							
8	30	From							
9	60		6'0"			A53	6	1	
10	70	Valve	2'0"			A53	6	1	
11	80		6'0"			A53	6	1	Anchor
12									

1. Changing header node numbers:

Select the menu Edit > Renumber nodes. The Renumber Nodes dialog will open. In the dialog, type in:

- 2 and 6 for From Row# and To Row#,
- 100 for Starting node number and
- 10 for Increase node numbers by.

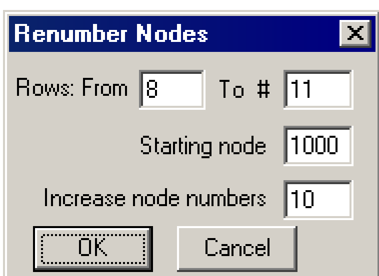


Press Enter (or click on OK) and CAEPIPE changes the node numbers.

2. Changing branch line numbering:

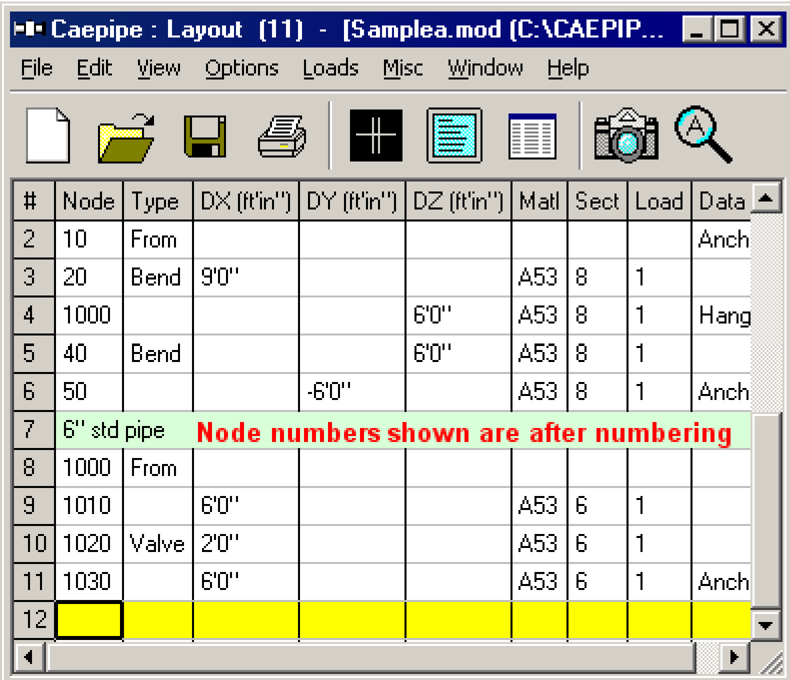
Again, use the menu Edit > Renumber nodes, in the Renumber Nodes dialog, type in

- 8 and 11 for From Row# and To Row#,
- 1000 for Starting node number and
- 10 for Increase node numbers by.



Press Enter (or click on OK) and CAEPIPE changes the node numbers.

The final layout looks like the following. Note that CAEPIPE tracks all occurrences of a specific node number (e.g., 1000, the hanger node, occurs on two rows 4 and 8) throughout the model so you do not have to remember to change every occurrence of the same node number in multiple places in your model.



Other reasons you may want to use the Renumbering feature:

1. In a big model, multiple edits sometimes bring about confusing node numbering. Use this feature to organize the numbering system.

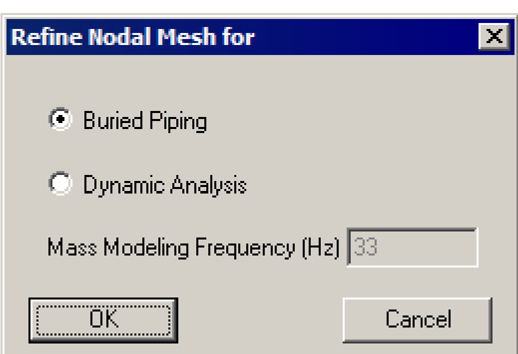
2. You may need to follow a guideline for numbering just to make parts of the piping system easily recognizable; all branch lines could get node numbers greater than 1000, all bypass lines could get node numbers greater than 2000, and so on.

.Refine Nodal Mesh.

This powerful command allows you to refine the Nodal Mesh for buried piping as well as for Dynamic Analysis.

Refine Nodal Mesh for Buried Piping

See section "Buried Piping" in the Technical Reference Manual for more details on "Refine Nodal Mesh for Buried Piping".



Refine Nodal Mesh for Dynamic Analysis

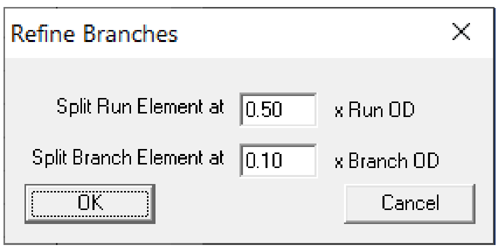
The purpose of this feature is to ensure that there are a sufficient number of mass points for an accurate dynamic model for the dynamic loading under consideration.

Intermediate mass points along a span are generated based on the mass modeling frequency defined by the user. See section “Refinement of Nodal Mesh for Dynamic Analysis” in the Technical Reference Manual for more details.

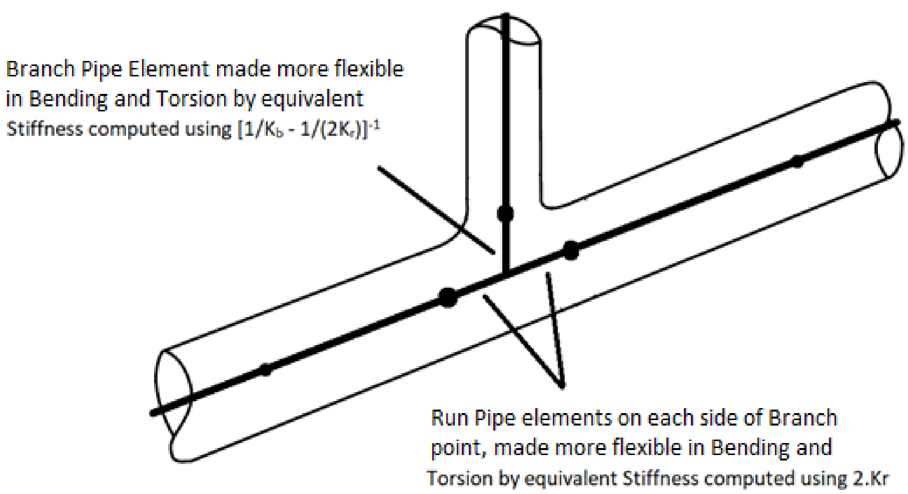
.Refine Branches for B31J.

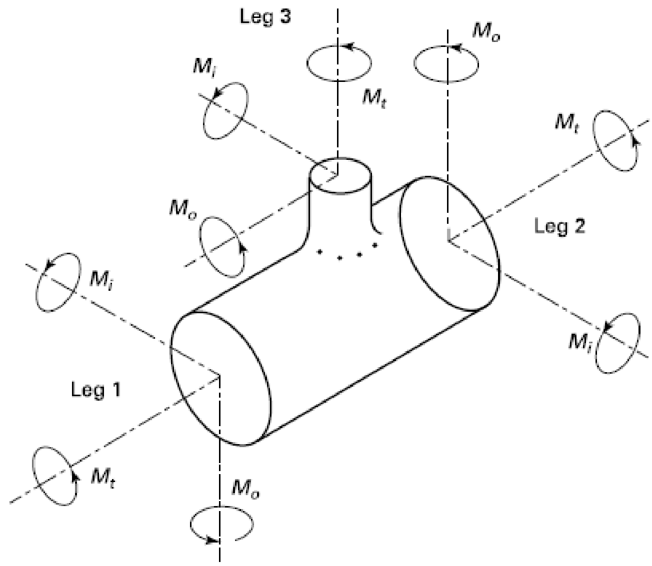
This command is active and available only if the “Piping Code” selected for Analysis is “ASME B31.1/ASME B31.3/ASME B31.4/ASME B31.5/ASME B31.8/ASME B31.9/ ASME B31.12” with the option “Use B31J for SIFs and Flexibility Factors” turned ON.

The purpose of this feature is to refine the elements connected to the Branch SIF node to have additional node points to compute and apply Flexibility Factors for Run and Branch Pipe as specified in ASME B31J.



a. Two (2) nodes on Run Pipe (one on either side of the Branch SIF node) at a distance equal to Run Split Factor x Run Pipe OD, where the Run Split Factor can be input by the user.
b. One (1) node on Branch Pipe at a distance equal to Branch Split Factor x Branch Pipe ODfrom the Branch SIF node, where the Branch Split Factor can be input by the user.





Moment	Flexibility Factor, k	Stiffness, in.-lb/rad (N·mm/rad)	Stiffness, in.-lb/rad (N·mm/rad)	
M_{i3} (Leg 3)	k_{ib}	M_{ib} / θ_{ib}	$(E)(I_b)/(k_{ib}\ d)$	K_b
M_{o3} (Leg 3)	k_{ob}	M_{ob} / θ_{ob}	$(E)(I_b)/(k_{ob}\ d)$	
M_{t3} (Leg 3)	k_{tb}	M_{tb} / θ_{tb}	$(E)(I_b)/(k_{tb}\ d)$	
$M_{i1,2}$ (Legs 1, 2)	k_{ir}	M_{ir} / θ_{ir}	$(E)(I_r)/(k_{ir}\ D)$	K_r
$M_{o1,2}$ (Legs 1, 2)	k_{or}	M_{or} / θ_{or}	$(E)(I_r)/(k_{or}\ D)$	
$M_{t1,2}$ (Legs 1, 2)	k_{tr}	M_{tr} / θ_{tr}	$(E)(I_r)/(k_{tr}\ D)$	

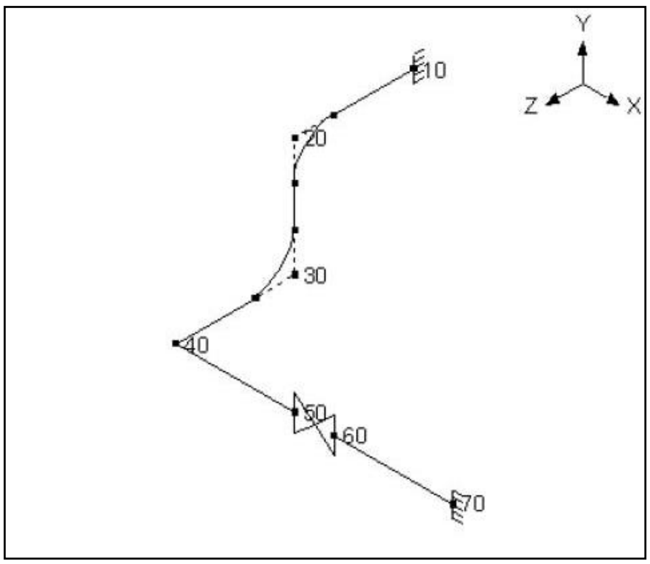
.Generate (Ctrl+G).

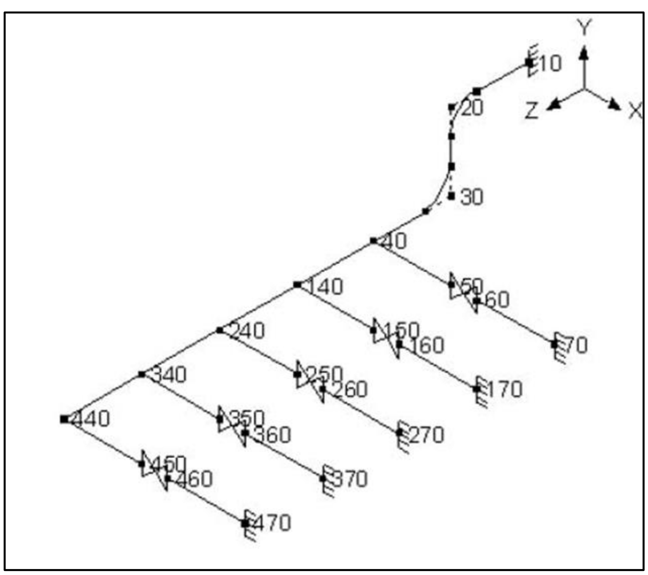
This powerful command allows you to generate copies of the whole or parts of the model while automatically taking care of node numbering and offsets. Its power is best illustrated by an example.

During "Generate" command execution, CAEPIPE issues a warning and stops when it encounters a Node number that already exists.

Example 1:

Assume that we want to go from the model shown in the first figure to the one shown in the second figure. There are two ways to reach the result. The first is by manually modeling every line like shown in the second figure. The second is the faster way using the Generate command, as you will see.

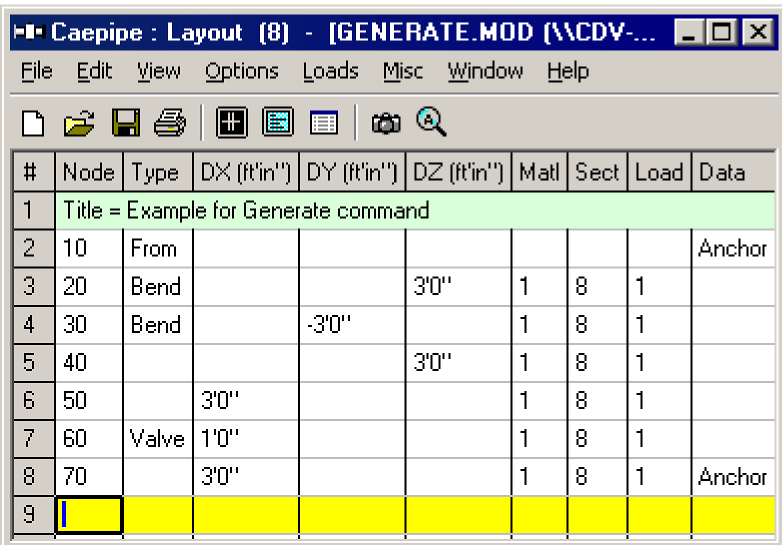




Summary of using the Generate command for the example:

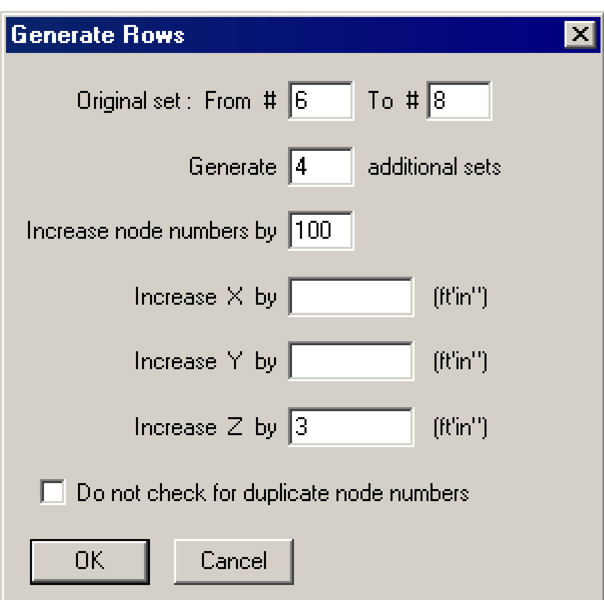
1. First identify the part of the model that repeats and model it. In this case, it is the part comprising of nodes 40 to 70 (rows 5 to 8).
2. Next, use the Generate command to create four additional branches, each starting with say 140, 240, 340 and 440 (these node numbers could be any numbers).
3. Then, create one connecting element (from node 40 to 140).
4. Further generate three additional connecting elements.

That is it. A number of figures follow to depict the above process pictorially.



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example for Generate command								
2	10	From							Anchor
3	20	Bend			3'0"	1	8	1	
4	30	Bend		-3'0"		1	8	1	
5	40				3'0"	1	8	1	
6	50		3'0"			1	8	1	
7	60	Valve	1'0"			1	8	1	
8	70		3'0"			1	8	1	Anchor
9									

The above window shows the layout for the starting point of the model. Use the Generate command.



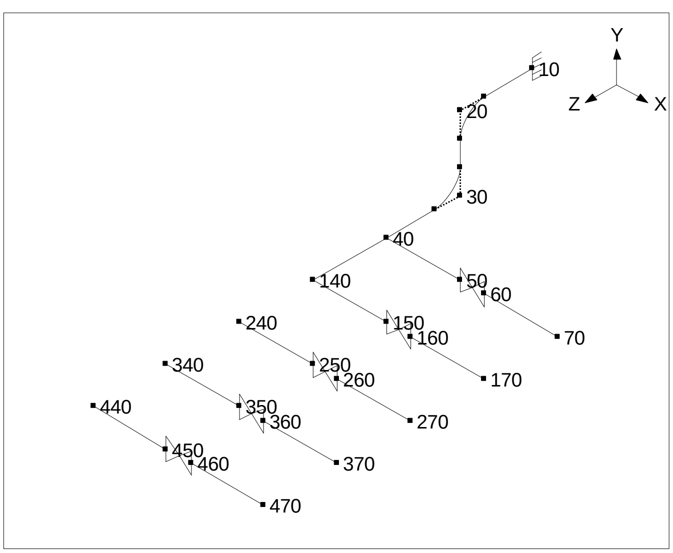
Notice the four sets that are generated (only the first one marked in the following figure).

Caepipe : Layout (32) - [GENERATE.MOD (C:\Users\Shp...

File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example for Generate command								
2	10	From							Anchor
3	20	Bend			3'0"	1	8	1	
4	30	Bend		-3'0"		1	8	1	
5	40				3'0"	1	8	1	
6	50		3'0"			1	8	1	
7	60	Valve	1'0"			1	8	1	
8	70		3'0"			1	8	1	Anchor
9	140	From		-3'0"	9'0"				
10	150		3'0"			1	8	1	
11	160	Valve	1'0"	} First set		1	8	1	
12	170		3'0"			1	8	1	Anchor
13	240	From		-3'0"	12'0"				
14	250		3'0"			1	8	1	
15	260	Valve	1'0"			1	8	1	
16	270		3'0"			1	8	1	Anchor
17	340	From		-3'0"	15'0"				
18	350		3'0"			1	8	1	
19	360	Valve	1'0"			1	8	1	
20	370		3'0"			1	8	1	Anchor
21	440	From		-3'0"	18'0"				
22	450		3'0"			1	8	1	
23	460	Valve	1'0"			1	8	1	
24	470		3'0"			1	8	1	Anchor

Look at the graphics that corresponds to the above Layout window.

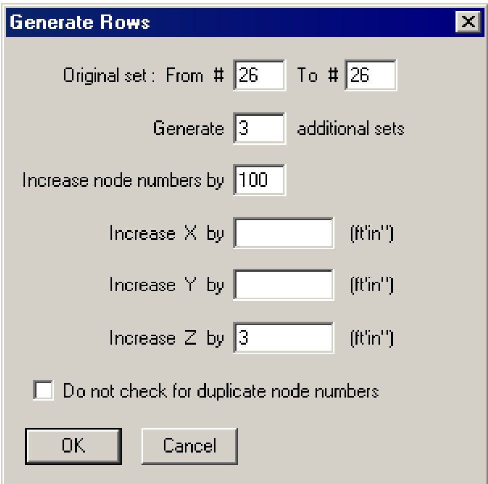


Next, create one connecting element between 40 and 140, and generate the remaining three.

24	470		3'0"			1	8	1	Anchor
25	40	From							
26	140					1	8	1	

} First connecting element

Type 40 on an empty row, Tab to Type, press"F" for "From," press Enter (rows 25 and 26 above). Now, generate the rest.

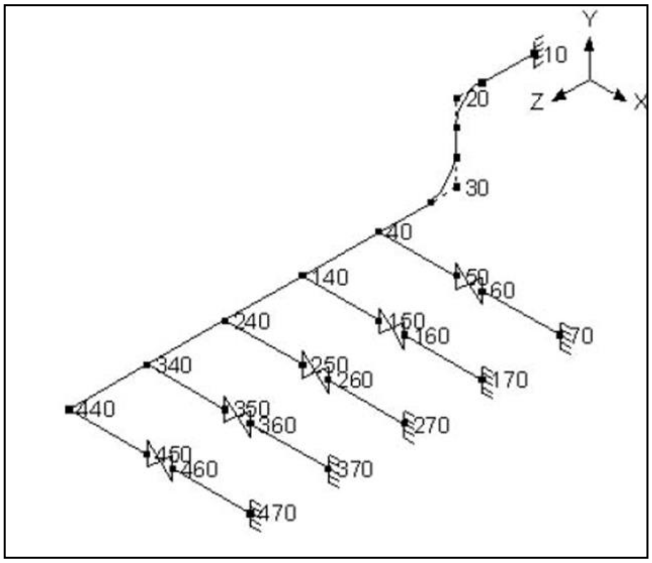


Caepipe : Layout (29) - [GENERATE.MOD (C:\Use...

File Edit View Options Loads Misc Window Help

#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example for Generate command								
2	10	From							Anchor
3	20	Bend			3'0"	1	8	1	
4	30	Bend		-3'0"		1	8	1	
5	40				3'0"	1	8	1	
6	50		3'0"			1	8	1	
7	60	Valve	1'0"			1	8	1	
8	70		3'0"			1	8	1	Anchor
9	140	From		-3'0"	9'0"				
10	150		3'0"			1	8	1	
11	160	Valve	1'0"			1	8	1	
12	170		3'0"			1	8	1	Anchor
13	240	From		-3'0"	12'0"				
14	250		3'0"			1	8	1	
15	260	Valve	1'0"			1	8	1	
16	270		3'0"			1	8	1	Anchor
17	340	From		-3'0"	15'0"				
18	350		3'0"			1	8	1	
19	360	Valve	1'0"			1	8	1	
20	370		3'0"			1	8	1	Anchor
21	440	From		-3'0"	18'0"				
22	450		3'0"			1	8	1	
23	460	Valve	1'0"			1	8	1	
24	470		3'0"			1	8	1	Anchor
25	40	From							
26	140					1	8	1	
27	240					1	8	1	
28	340					1	8	1	
29	440					1	8	1	
30									

Full model layout



Final model as shown is the desired result. So, as you saw, Generate command allowed us to create four branch lines from one existing line in three operations.

Example 2:

See example under Beam section in the Technical Reference Manual (Example 1: Pipe Rack using Beams)

.Regenerate.

Sometimes, owing to multiple edits, the model may be in an indeterminate stage. If you have made many edits, use this command to regenerate the model which will reinterpret the input and bring the model to a consistent state.

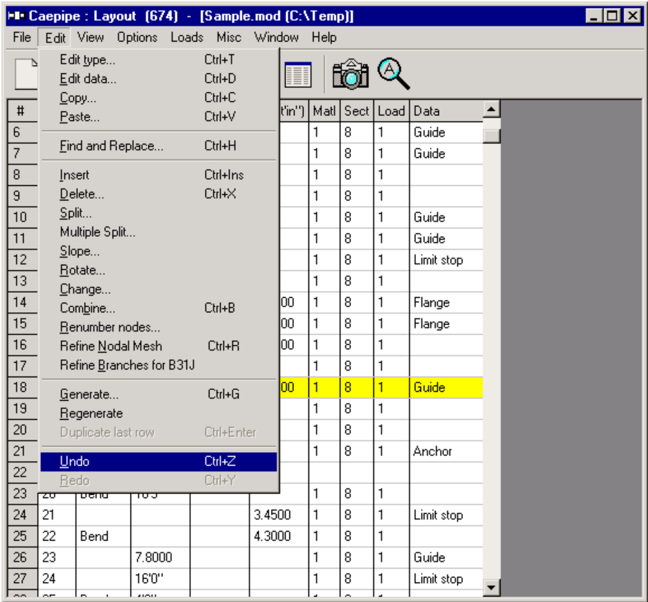
.Duplicate Last Row (Ctrl+Enter).

This is another handy feature that increases your productivity. For rows that repeat themselves (except for node number), this is a time saver. This is useful in modeling pipe racks or straight runs of pipe. Simply input all data on to a row and press Ctrl+Enter to duplicate this row on to the next with the node number automatically incremented.

CAEPIPE cannot duplicate a Location/Comment/Hydrotest row, nor can it duplicate a row between two existing rows. This command works only on new empty rows at the end of a model. If you press Ctrl+Enter right after a Location/Comment/Hydrotest row, CAEPIPE will duplicate the row right before the Location/Comment/Hydrotest row.

.Undo (Ctrl+Z).

CAEPIPE can perform an Undo operation over 1000 times (1024)! This operation includes most user-input/edit actions through the Layout, List and Graphics windows.



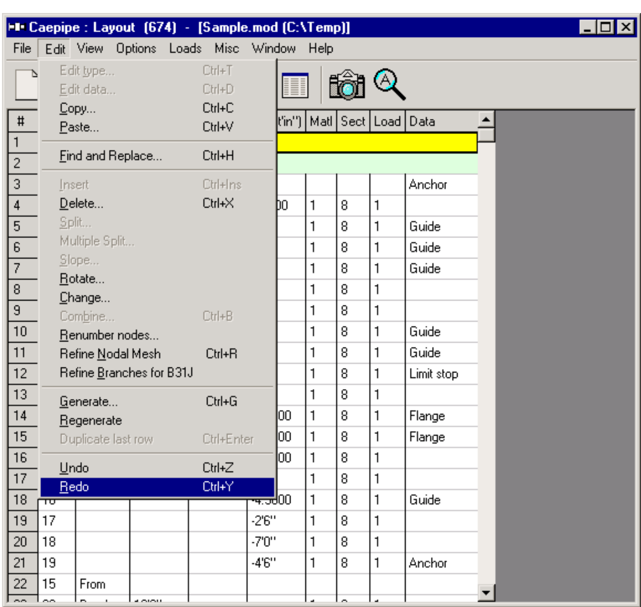
Example 1: If you change your mind after individually deleting rows 10-14 and the first three guides (see figure above), you could use the Undo command (Ctrl+Z) eight times to undo all of these operations.

Example 2: If you were to change a few of the analysis options (such as piping code, # of thermal loads and Bourdon effect through the menus), you could undo these changes by using this command three times, even though these changes are not made visible.

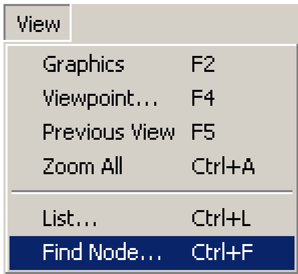
.Redo (Ctrl+Y).

Conversely, CAEPIPE can perform a Redo operation over a 1000 times (1024), which can include most user-input/edit actions through Layout, List and Graphics windows, including recently performed Undo operations.

For example 1 under Undo: You could use the Redo operation to delete what you undid in the above example! In other words, delete rows 10-14 and the first three guides.



View Menu

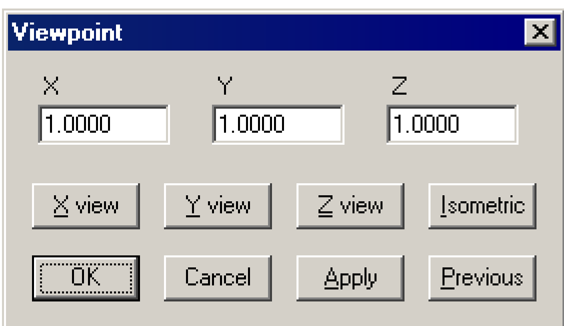


A few commands for the Graphics window can be carried out from here.

.Graphics (F2).

Use this command to move focus to the Graphics window. If the window is not open, then F2 will open the Graphics window and move focus to it. Pressing F2 again moves focus back to the Text window.

.Viewpoint (F4).



Use this command to set the graphics viewpoint. Several useful buttons inside the dialog allow you to change viewpoint to a preset one. For example, if you want to see the "plan" view (Y-vertical), click on "Y view" button.

.Previous View (F5).

Use this command to display the previously viewed graphics (zoom level and image area) in the Graphics window.

.Zoom All (Ctrl+A).

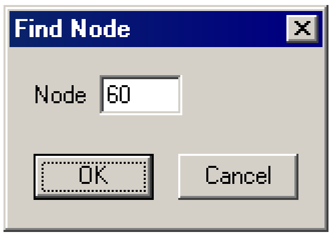
Use this command to view the whole model inside the Graphics window. Note that this command does not move focus to the Graphics window but brings the whole model graphics into view.

.List (Ctrl+L).

This is one of many CAEPIPE's unique features. This command allows you to view itemized lists of every input data (pipes, bends, materials, valves, spectra, etc.) in a separate window. See more detailed explanation under the section titled "Layout Window" in this manual.

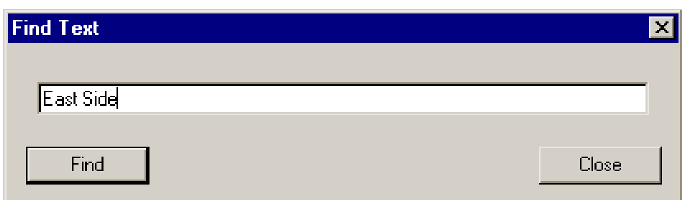
.Find Node (Ctrl+F).

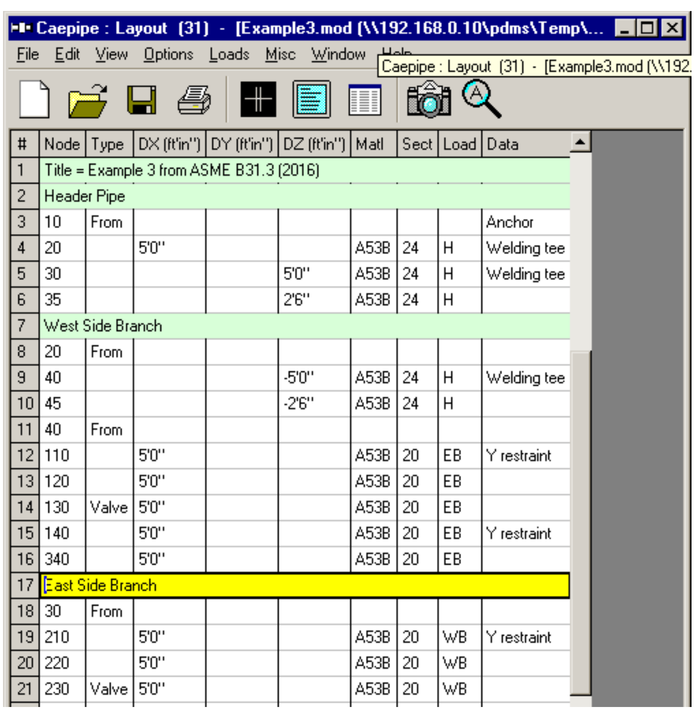
Use this command to search for a node number inside this window. You may also search for jacket and bend end nodes (e.g., 20A, 30C, 160J), *if* they are referenced on the layout screen.



.Find Text(Ctrl+Shift+F).

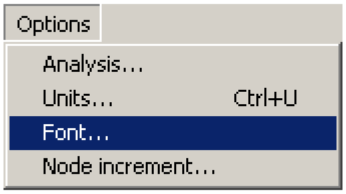
You can quickly search for every occurrence of a specific word or phrase from “Comment” rows of CAEPIPE using this command. In the dialog box shown, type the word or phrase that you want to find from “Comment” rows. Any instance of the found text is highlighted in the layout window.





To repeat “Find” the last word or phrase (after closing Find Text window), use “Ctrl+F3”.

Options Menu



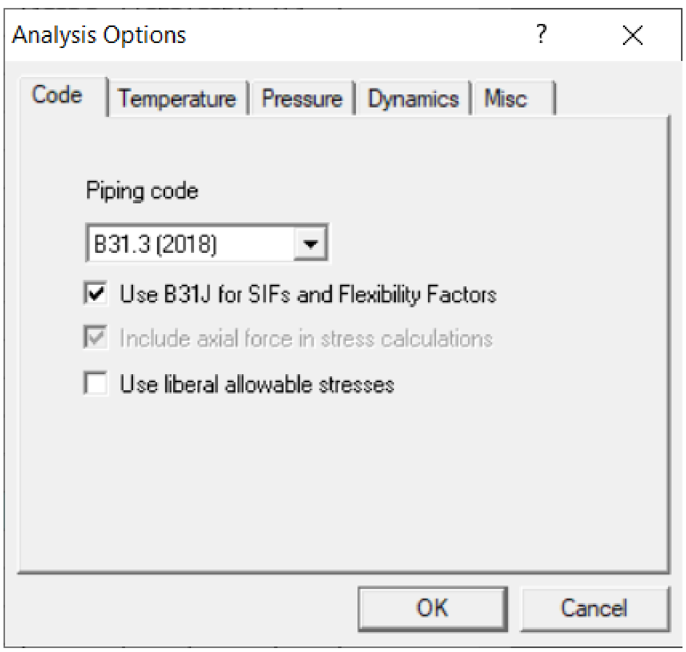
This menu allows you to specify analysis options, units, font and the node number increment (for automatic generation of the next node number while inputting a model).

.Analysis.

The Analysis Options dialog is shown. Here you can specify analysis options related to piping codes, temperature, pressure, dynamic analysis, etc., in various tabs of the dialog.

Code

On this tab you can choose the piping code and also set options for that piping code.



Piping Code

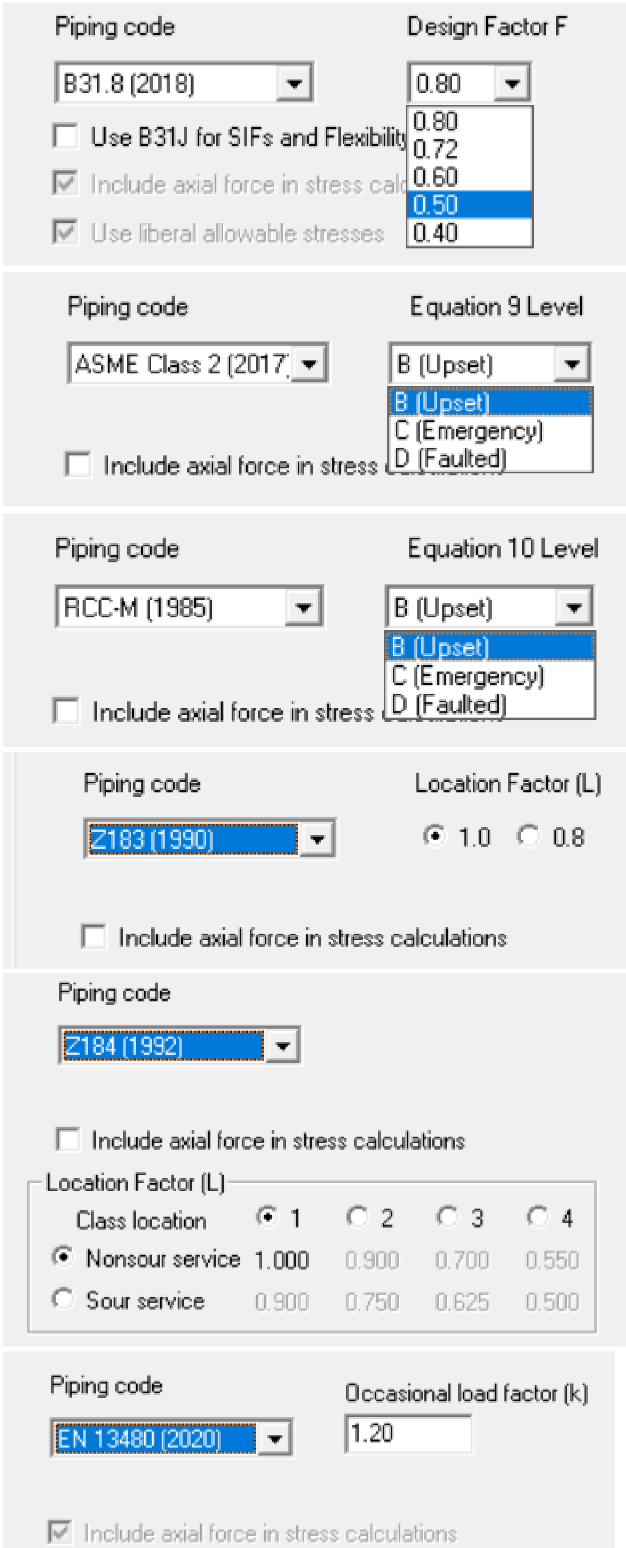
Select a piping code from the "Piping code" drop-down combo box. The following codes are currently available:

- None
- B31.1
- B31.1 (1967)
- B31.3
- B31.4
- B31.5
- B31.8
- B31.9
- B31.12
- ASME Section III Class 2 (1980, 1986, 1992, 2015 and 2017)
- ASME Section III Class 3 (2017)
- BS 806 (British)
- IGEM
- EN 13480 (European)
- Norwegian (1983 and 1990)
- RCC-M (French)
- CODETI (French)
- Stoomwezen (Dutch)
- Swedish
- Z183 (Canadian)
- Z184 (Canadian)
- Z662 (Canadian)

Notes:

For piping code = NONE, Rotating Equipment Qualification and Dynamic Analysis (Harmonic Analysis, Time History Analysis and Force Spectrum Analysis) are disabled in CAEPIPE. Hence, changing the analysis code to NONE from other code will delete all data related to Rotating Equipment, Harmonic load, Time History and Force Spectrum from the stress model without any warning.

- When the selected piping code is "None," a "Static analysis" load case, which includes weight, pressure, thermal, cold spring, static seismic and wind loads, all applied at the same time, is available for analysis. No dynamic loads (time history, response spectrum, etc.) are included. For"Piping code = None" and a non-FRP material, the in-plane and out-of-plane Stress Intensification Factors (SIF) for a 90° bend/elbow are taken to be the same as those computed using ANSI B31.3. For a bend with bend angle less than 90°, the in-plane and out-of-plane SIFsarecomputed using the procedure given in ASME/BPVC Section III, Division1, Case N-319-2 "Alternate Procedure for Evaluation of Stresses in Butt Welding Elbows in Class 1 Piping."
- For some piping codes, you need to select an additional option (such as Design Factor for B31.8, equation level for ASME and RCC-M, Location Factor for Z183 and Z184, Occasional load factor for EN13480). These are shown below.



Use B31J for SIFs and Flexibility Factors

This option visible only if the Piping Code selected for Analysis is ASME B31.1/ASME B31.3/ASME B31.4/ASME B31.5/ASME B31.8/ASME B31.9/ASME B31.12.

By checking this box, you can instruct CAEPIPE to compute Stress Intensification Factors (SIFs) and Flexibility Factors (FFs) as per ASME B31J.

When this option is not checked, then CAEPIPE will compute SIFs and FFs as per Appendix D of the ASME code selected for Analysis.

For further details on implementation, see CAEPIPE Code Compliance Manual.

Include axial force in stress calculations

By checking or un-checking this box, you can include or exclude axial force in stress calculations (F/A term). This option applies to all stresses (Sustained, Occasional and Expansion) and is provided since many piping codes do not clearly mandate the axial term's inclusion. When included, CAEPIPE includes the axial term F/A (where F=axial load, A=Pipe cross-sectional area)in itscalculation of all stress equationssuch as S_L, $S_O$andS_E.

Note that this option may be turned on depending on the code requirements.

Use Liberal Allowable Stresses

For B31.1, B31.1(1967), B31.3, B31.5, B31.12 and CODETI piping codes, the allowable expansion stress may be increased by a difference between the allowable sustained stress and the actual sustained stress times the stress range reduction factor. See Thermal Expansion Stress Range or equivalent under each piping code in CAEPIPE Code Compliance Manual.

A piping code committee member opines as follows.

Perhaps the term "liberal allowable" is not the best to describe the allowable. The allowable stress in B31.1 Eq. 13, i.e.,$S_A + f(S_h - S_L)$, can be used anytime. The only prerequisite is knowing the S_L stresses, which will only be the case after the supports are known or implicit. The Eq. 1 allowable stress, S_A, is the allowable stress traditionally used in the flexibility analysis when no supports other than the equipment anchors are known. The flexibility analysis is used to determine whether the layout of piping between the equipment anchors is adequate. The displacements determined in the flexibility analysis will allow the designer to devise the pipe weight supports to interfere with the flexibility of the pipe as little as possible, i.e., the designer will use rigid supports where the piping does not move much, use variable springs where the piping moves a small amount (most typically 1/4" to 4"), and use constant springs where the movement is great (again, typically over 4"). For lateral loads the same concept as used for the pipe weight supports is used for the lateral supports, i.e., if lateral displacements are small, rigid supports may be used, for larger lateral movements, gapped or shock suppressor supports are used (although shock suppressor supports require considerable maintenance attention and in the long run are usually not preferable to gapped supports).

General Note on Thermal Allowable Calculation for All Piping Codes

Cold allowable (Sc) and Hot allowable (Sh) stresses used in thermal stress calculations are calculated as follows for all piping codes:

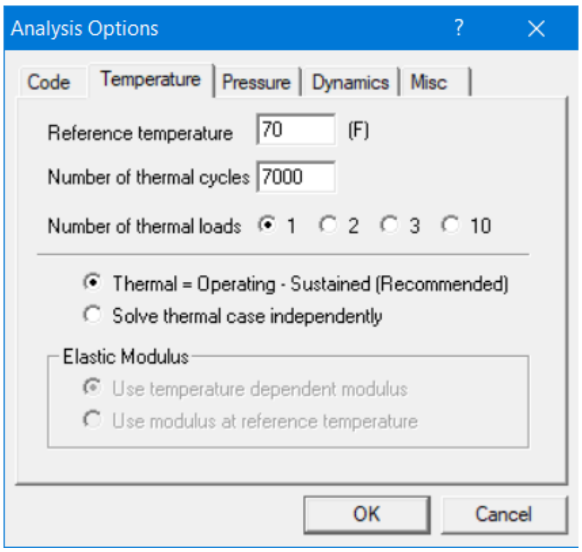
- For Expansion Cases T1 through T10, Cold allowable (S_c) is taken at Minimum (T_{ref}, T_n) and Hot allowable (S_h) is at Maximum (T_{ref}, T_n), where n = 1 to 10.

- For Thermal Ranges Tx-Ty, Cold allowable (S_c) is at Minimum (Tx, Ty) and Hot allowable (S_h) is at Maximum (Tx, Ty), where x = 1 to10 and y = 1 to 10 and x is not equal to y.

The specifics are summarized for each B31.x piping code under the title"Thickness and Section Modulus used in Weight, Pressure and Stress Calculations for ANSI B31.x Codes" of the CAEPIPE Code Compliance Manual.

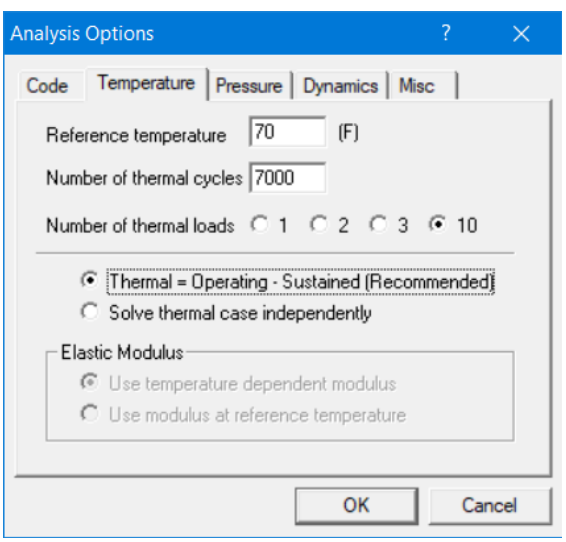
Temperature

On this tab you can set options related to thermal loads.



Reference Temperature

Input the reference temperature here, usually 70°F (21.1°C) corresponding to the installation temperature.CAEPIPE uses this temperature to lookup material properties table for the elastic modulus (see subsection titled "Elastic Modulus" below) and to calculate thermal stress ranges such as ($T1 – T_{ref}$), ($T2 – T_{ref}$), (T1 – T2) = ($T1 – T_{ref}$) – ($T2 – T_{ref}$), etc.



Number of Thermal cycles

CAEPIPE uses this number to determine the stress range reduction factor, f, which is used to reduce the allowable expansion stress range, S_A.

The reduction factor "f" used by CAEPIPE for different piping codes are described / listed in the Code Compliance Manual.

The typical equation for calculating thermal expansion stress range (e.g., for B31.1 code) is

$$S_A = f(1.25S_c + 0.25S_h)$$

where

S_c = allowable stress at cold temperature

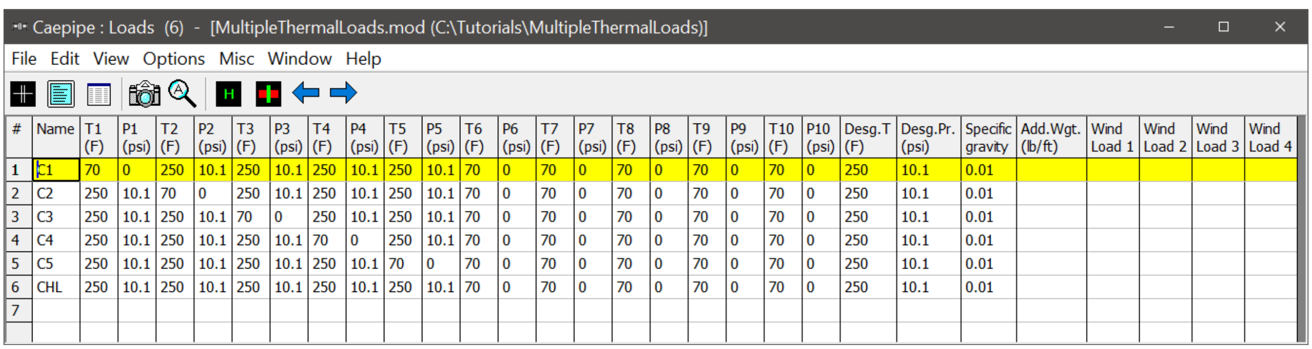
S_h = hot allowable stress

f = stress range reduction factor

Number of Thermal Loads

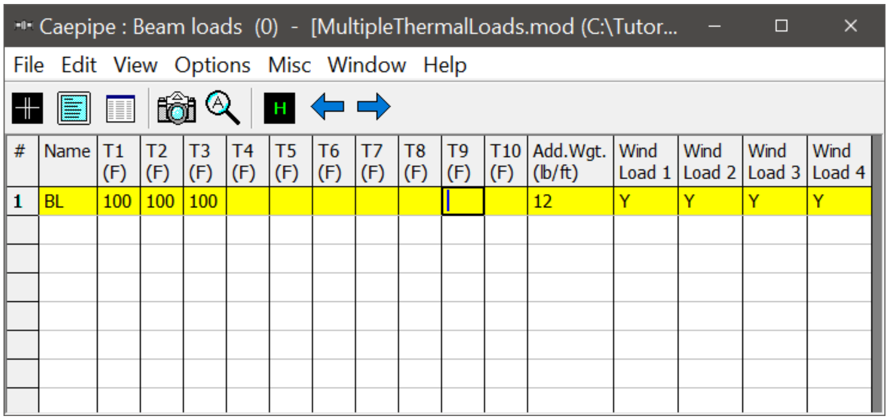
You can enter up to 10 thermal loads. This feature must be first set through Layout window > Options > Analysis > Temperature. The maximum number of thermal loadsthat one or more elements of a stress model can experience during operation should be input for the Number of Thermal Loads. Up to 10 operating temperatures may be applied as part of Load (along with 10 operating pressures) under Misc menu > Loads.

In addition to 10 thermal loads and 10 pressures, you can also enter the Design Pressure and Design Temperature through the fields Desg.T and Desg.P respectively. Refer to Section titled "Load" in the Technical Reference Manual for details on Design Temperature and Design Pressure.

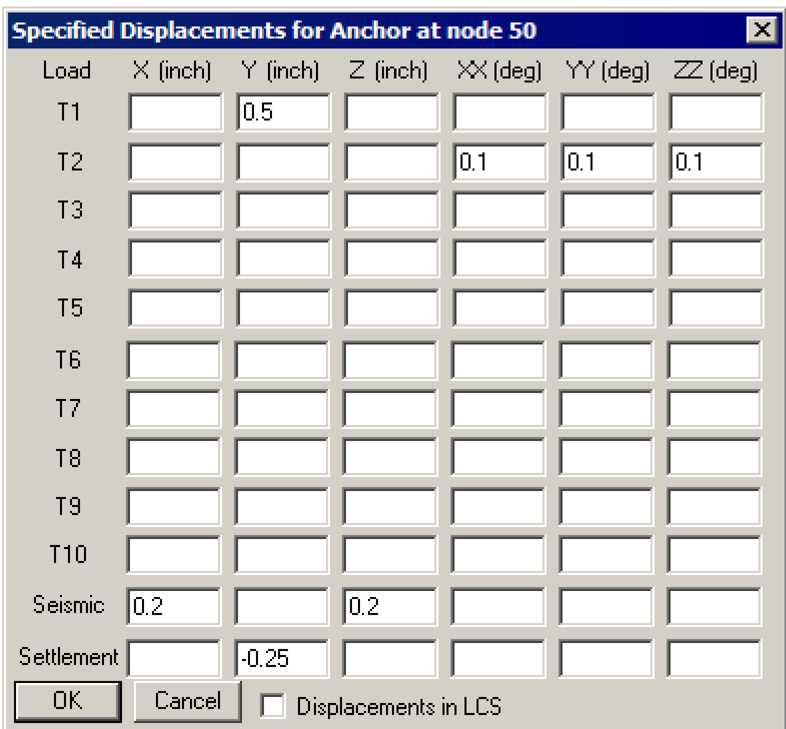


#	Name	T1 (F)	P1 (psi)	T2 (F)	P2 (psi)	T3 (F)	P3 (psi)	T4 (F)	P4 (psi)	T5 (F)	P5 (psi)	T6 (F)	P6 (psi)	T7 (F)	P7 (psi)	T8 (F)	P8 (psi)	T9 (F)	P9 (psi)	T10 (F)	P10 (psi)	Desg.T (F)	Desg.Pr. (psi)	Specific gravity	Add.Wgt. (lb/ft)	Wind Load 1	Wind Load 2	Wind Load 3	Wind Load 4
1	C1	70	0	250	10.1	250	10.1	250	10.1	250	10.1	70	0	70	0	70	0	70	0	70	0	250	10.1	0.01					
2	C2	250	10.1	70	0	250	10.1	250	10.1	250	10.1	70	0	70	0	70	0	70	0	70	0	250	10.1	0.01					
3	C3	250	10.1	250	10.1	70	0	250	10.1	250	10.1	70	0	70	0	70	0	70	0	70	0	250	10.1	0.01					
4	C4	250	10.1	250	10.1	250	10.1	70	0	250	10.1	70	0	70	0	70	0	70	0	70	0	250	10.1	0.01					
5	C5	250	10.1	250	10.1	250	10.1	250	10.1	70	0	70	0	70	0	70	0	70	0	70	0	250	10.1	0.01					
6	CHL	250	10.1	250	10.1	250	10.1	250	10.1	250	10.1	70	0	70	0	70	0	70	0	70	0	250	10.1	0.01					
7																													

This also enables you to enter 10 temperatures under Miscmenu > Beam Loads.



Additionally, up to 10 specified thermal displacements can be entered for Anchor and Nozzle data types.



Thermal = Operating – Sustained

Thermal load case results can be calculated as the difference between the Operating and the Sustained load cases, i.e., the thermal load case is not solved independently. This is the recommended procedure to solve thermal cases (T1 through T10) especially when nonlinearities (limit stops, friction, etc.) are present.

When a model has no nonlinearities, the forces, moments and displacements at nodes add up (i.e., Sustained load + Thermal load = Operating load). When a nonlinearity is present, each load case can be solved independently, i.e., the numbers are not added from Sustained (W+P) to Thermal (T1) to get Operating (W+P+T1); in other words, the results do not always add up. In the past, this used to be the only method of solving when nonlinearities were present. However, now you can use the new (recommended) option called "Thermal = Operating – Sustained", which is in line with the requirement that thermal stress range is to be calculated between two actual thermal states. Use it to solve for the thermal case as the difference between Operating and Sustained cases, which will also ensure that sustained and thermal case results add up to operating case results.

Solve Thermal case

Results for the thermal load case are obtained by solving this case independently.This option is generally not recommended. See above discussion.

Elastic Modulus

User can select one of the two options available in CAEPIPE (as given below).

- Use temperature dependent modulus
- Use modulus at reference temperature

When the option "Use temperature dependent modulus" is chosen in CAEPIPE, then CAEPIPE computes the elastic modulus at the temperature under consideration (E_{hot}) while forming the Global stiffness matrix for each load case, i.e., at temperature T1 for Operating 1, T2 for Operating 2, … T10 for Operating 10. In short, CAEPIPE uses hot modulus (E_{hot}) to compute the Displacements, Element Forces & Moments, Support Loads and Stresses when the option "Use temperature dependent modulus" is chosen.

On the other hand, when the option "Use modulus at reference temperature" is chosen in CAEPIPE, then CAEPIPE computes the elastic modulus at "Reference Temperature (T_{ref})" (hereafter referred to as E_{cold}) while forming the Global stiffness matrix for each load case. In short, CAEPIPE uses cold modulus (E_{cold}) to compute the Displacements, Element Forces & Moments, Support Loads and Stresses when the option "Use modulus at reference temperature" is chosen.

Note:

"Expansion Stresses" for Restrained portionof piping (i.e., buried portion) for the piping code Z662 are calculated using elastic modulus at reference temperature (E_{cold}) irrespective of the option (among the above two options) chosen by the user for Elastic Modulus. On the otherhand, for Restrained portion of piping, Displacements, Element Forces & Moments, Support Loads and Stresses (excepting Expansion Stresses) are computed using the elastic modulus option chosen by the user, i.e., E_{hot} or E_{cold}.

For Unrestrained portion of piping (portion not buried), for the piping code Z662, Displacements, Element Forces & Moments, Support Loads and Stresses are computed using the elastic modulus option chosen by the user, i.e., E_{hot} or E_{cold}.

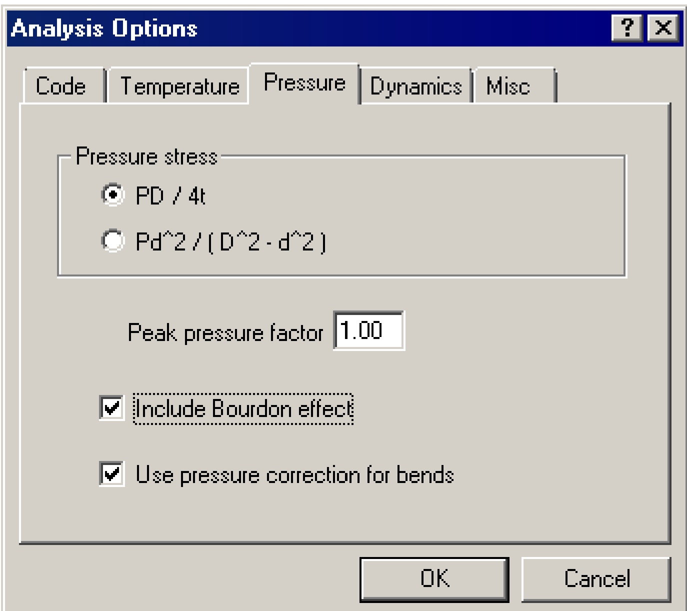
As per para. 402.2.2 of ASME B31.4 (2019) and para. 832.2 (g) of ASME B31.8 (2018), flexibility calculations shall be based on the modulus of elasticity at ambient temperature, i.e., modulus at reference temperature (E_{cold}). Hence, "Use modulus at reference temperature" (available through CAEPIPE Layout > Options > Analysis > Temperature) is set as "default" and is disabled for user to modify when B31.4 or B31.8 code is selected. In short, for both Restrained and Unrestrained portions of piping, E_{cold} is used in CAEPIPE to calculate Displacements, Element Forces & Moments, Support Loads and Stresses for B31.4 and B31.8 codes.

As per para. 519.4.5(a) of ASME B31.5 (2016) code, Bending and Torsional stresses shall be computed using the as-installed modulus of elasticity, i.e., modulus at installation temperature (E_{cold}). Hence, "Use modulus at reference temperature" (available through CAEPIPE Layout > Options > Analysis > Temperature) is set as "default" and is disabled for user to modify when ASME B31.5 code is selected.

Similarly, as per para. 12.2.7.2 of EN 13480-3 (2017), value of the modulus of elasticity (E_t) used for flexibility analysis shall be the value taken at the temperature of the piping load under consideration, i.e., modulus at temperature under consideration (E_{hot}). Hence, "Use temperature dependent modulus" (available through CAEPIPE Layout > Options > Analysis > Temperature) is set as "default" and is disabled for user to modify when EN 13480 code is selected.

Pressure

On this tab you can set options related to pressure loads.



Pressure Stress

The longitudinal pressure stress may be calculated as:

$\mathrm{pD}/4\mathrm{t}$ or $\mathrm{pd}^2/(\mathrm{D}^2 - \mathrm{d}^2)$, both given as options.

However, for B31.8 code, the pressure stress term $\mathrm{pd}^2/(\mathrm{D}^2 - \mathrm{d}^2)$ is not an option.

For EN 13480, the options are $[\mathrm{pD}/4\mathrm{t}]$ or $[\mathrm{pd}^2/(\mathrm{D}^2 - \mathrm{d}^2) + (\mathrm{p}/2)]$.

Peak Pressure Factor

For occasional loads (seismic, spectrum and wind), the pressure 'p' in longitudinal pressure stress is multiplied by this factor.

Bourdon Effect

Bourdon effect is the tendency of straight pipes to elongate due to internal pressure. This may be included or excluded from the analysis by checking or unchecking the box.

Internal pressure will expand the pipe cross-section radially outward (i.e., bulge out) thereby contracting the length of the pipe due to Poisson's effect. On the other hand, external pressure will contract the pipe cross-section radially inward (i.e., shrinking inwards) thereby elongating the pipe length again due to Poisson's effect. On the contrary, due to end-cap force developed by pressure, straight pipe elements will elongate for internal pressure and contract for external pressure. These physical phenomena are included in CAEPIPE calculations when the "Bourdon effect" button is turned ON. Pipe stresses and support loads generated due to such "Bourdon effect" deformations are generally considered as "Secondary Stresses and Support loads" and normally considered as part of the thermal expansion load case.

Therefore, by default, this effect is treated as an expansion load and included in the expansion and operating load cases. It is not applied to the sustained and occasional load cases.

If an environment variable ("BOURDONP") is set (to YES), however, the Bourdon effect is instead treated as a sustained load and included in the sustained and operating load cases. It is not applied to the expansion load case.

For a straight pipe, the following equation is used.

$$Pressure\ deflection = \frac{Pressure \times r_m \times L \times (1 - 2\nu)}{2 \times E \times t}$$

where
r_m = mean radius of the pipe
L = length of the pipe element
ν = Poisson's Ratio
E = Elastic Modulus (Axial modulus in the case of FRP pipes)
t = pipe thickness

Pressure Correction for Bends (Pressure stiffening effect)

Pressure correction for bends is different from and unaffected by Bourdon effect. In large diameter thin-wall bends, pressure can significantly affect their flexibility and SIF. If pressure correction for bends is used: the Flexibility of the bend is divided by

$$1 + 6 \times \left(\frac{P}{E}\right)\left(\frac{r}{t}\right)^{7/3}\left(\frac{R}{r}\right)^{1/3}$$

and the SIF for the bend is divided by

$$1 + 3.25 \times \left(\frac{P}{E}\right)\left(\frac{r}{t}\right)^{5/2}\left(\frac{R}{r}\right)^{2/3}$$

where
P = pressure
E = elastic modulus
r = mean radius of matching pipe
t = nominal wall thickness
R = bend radius

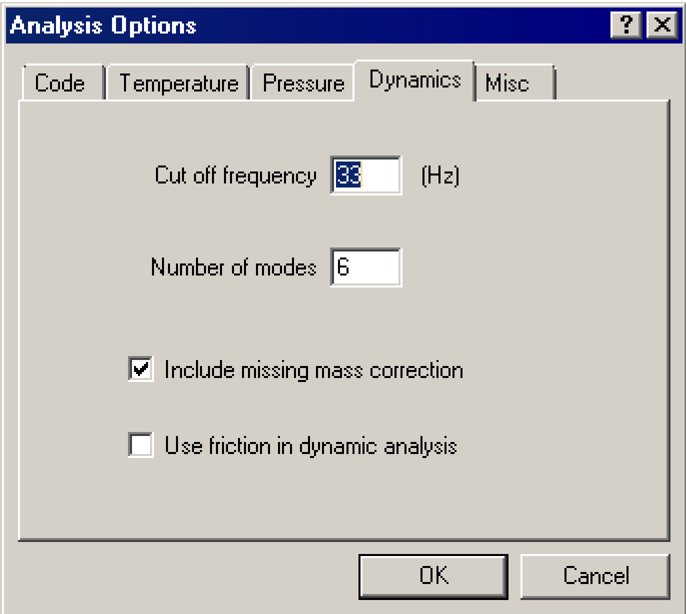
It is recommended that this option be turned 'ON' whenever the internal or external pressure is high, so that the stiffening of the bend due to internal or external pressure is considered in the analysis. In other words, when a thin-walled pipe bend is bent by bending moments at its two ends, the bend will ovalize and increase its flexibility. Internal or external pressure will reduce this bend ovalization, thereby increasing its stiffness.

Pressure correction decreases the flexibility of the piping system (by increasing the stiffness of the system because of the stiffened elbows). Hence, the system frequencies (in modal analysis) tend to increase.

This pressure correction option is provided for all piping codes available in CAEPIPE.

Dynamics

On this tab you can set options related to dynamic analysis.



Cutoff Frequency / Number of Modes

These two options jointly control how the modal analysis routine works to extract the natural frequencies. Frequency(Hertz)and the chosen number of modes will determine the minimum number of modes extracted. The modal analysis will terminate either when the number of modes requested has been extracted or after extracting an additional frequency above the specified cutoff frequency value, whichever occurs first.

For earthquake analysis, a typical value for cutoff frequency is 33 Hz. The maximum frequency you can input is 9999 Hz.

When the selected piping code is "None," and the model contains nonlinearities (limit stops, certain expansion joints), a static analysis for a given model with and without a g-load input may produce different frequencies depending upon the status of the nonlinearities due to the presence or absence of the g-load in the static load case.

Include Missing Mass Correction

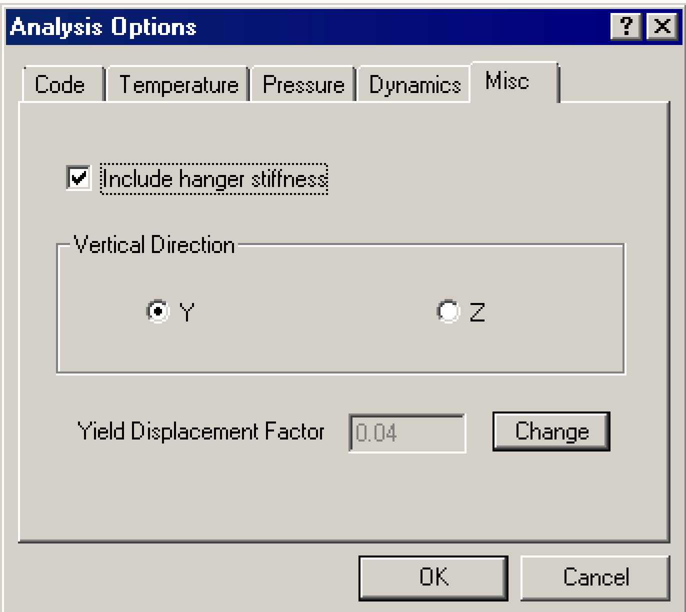
Missing mass correction to the response spectrum analysis can be included using this check box. See topic by the same name in the Technical Reference Manual.

Use Friction in Dynamic Analysis

Friction is optional in dynamic analysis. If friction is included in dynamic analysis, the global stiffness matrix arrived at upon completion of the iterative calculations for the first operating case (W+P1+T1) is used to compute frequencies and mode shapes for the piping system.

Misc

On this tab you can set miscellaneous options.



Include Hanger stiffness

When the checkbox is checked, CAEPIPE adds the hanger spring rates for variable spring and user hangers to the global stiffness matrix (recommended setting). Hanger spring rates for rod hangers (if active) are always included even when this checkbox is not checked.

If the checkbox is not checked, CAEPIPE does not include the hanger stiffnesses in the analysis (excepting for hanger spring rates for "active" rod hangers). Some users prefer this as it more closely matches "hand calculations."

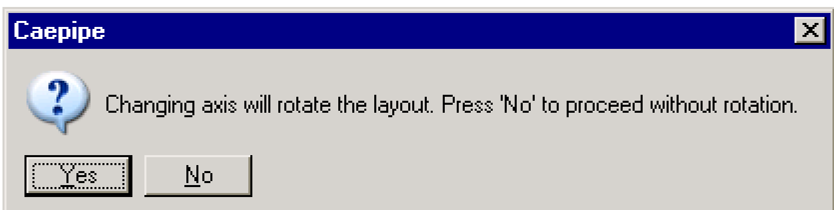
In the 1970s, performing pipe stress analysis on mainframe computers was expensive (about $1,000/run). So, pipe stress analysts would run a Thermal (T1) case alone first. Then, they would run a deadweight (DW) case with rigid supports at chosen hanger locations. Using the support loads and the hanger travel, analysts would select appropriate variable spring hangers from hanger manufacturer catalogs. To save on computing costs, they would not update the global stiffness matrix [K] with the hanger stiffnesses for the newly selected hangers because they would have to reanalyze the model which would cost more money. Many industrial plants built 30 or 40 years ago in the USA and presumably in other countries have pipe stress analyses done in this manner. These plants have archived such analysis reports that do not include hanger stiffnesses (as part of the global stiffness matrix [K]).

In later years, as and when the analysts for these plants needed to reanalyze those piping systems, they had to use modern programs like CAEPIPE. But, before they accepted the new results, they sometimes liked to verify that the new results (from CAEPIPE) matched the results from the old reports generated by the mainframe pipe stress analysis programs. This is why CAEPIPE provides the option "Do not include (or include) hanger stiffnesses." It helps the engineers compare results from CAEPIPE with the old archived reports.

Today, with cheap PC computing power, there is no reason why hanger stiffnesses should be excluded from analysis. In fact, including hanger stiffnesses provides a more accurate picture of system behavior. As such, we recommend that you "include hanger stiffnesses" in every analysis.

Vertical Direction

You may specify either Y or Z as the vertical global axis for the model. If the vertical global axis is modified while building the stress layout, CAEPIPE will display the message as shown below. Press the button "Yes" to change the vertical axis and rotate the stress layout.



Pressing the button "No" will change the axis without rotating the model.

Yield Displacement Factor

Soil (in Buried piping analysis) is modeled with an initial stiffness and an ultimate load, after reaching which, displacement continues without a further increase in load (i.e., yield stiffness becomes zero). You can change the initial stiffness by changing the Yield Displacement Factor (default=0.04, range from 0.04 to 1.0). No changes are recommended unless there is a problem with convergence during Buried Piping analysis.

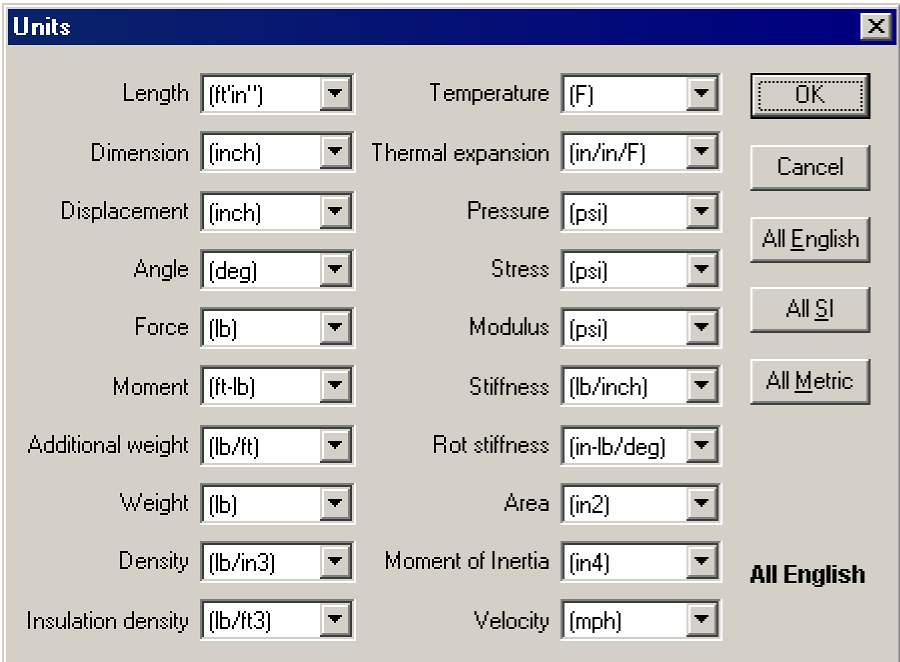
.Units (Ctrl+U).

A flexible command, Units allows you to set any combination of units: English, SI or Metric, for any item. For convenience "All English,""All SI" and "All Metric" buttons set all the units to that particular choice with a single button click.

As an example, engineers in the USA sometimes use lb/in. for stiffnesses but use SI units for the rest, for their international clients. The specific combination of units that you create is saved with the model so you do not have to reset them every time. You could do your analysis in English units and present your results to your client in SI units with the click of a button.

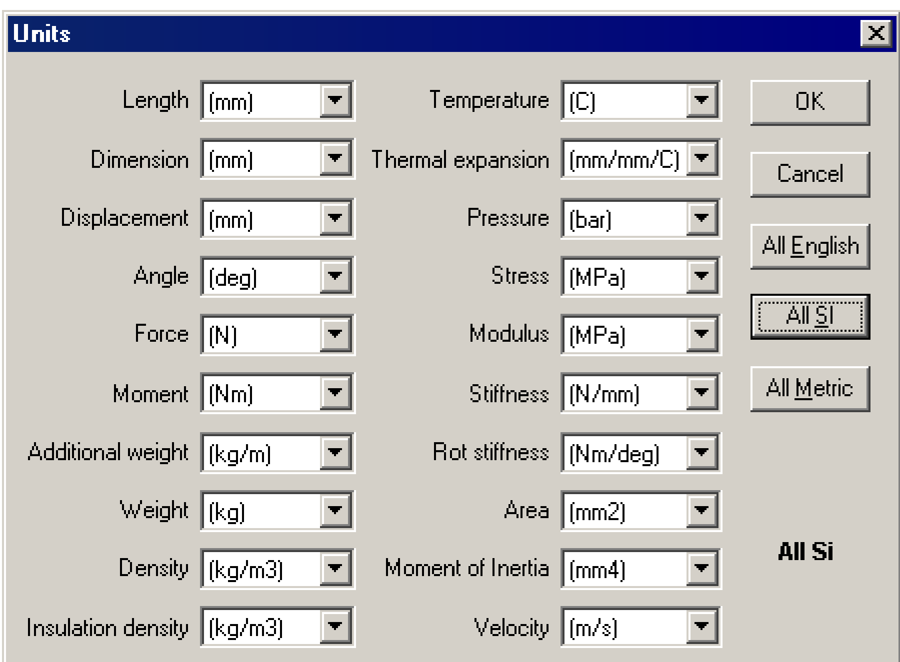
All English

All English units are shown below.



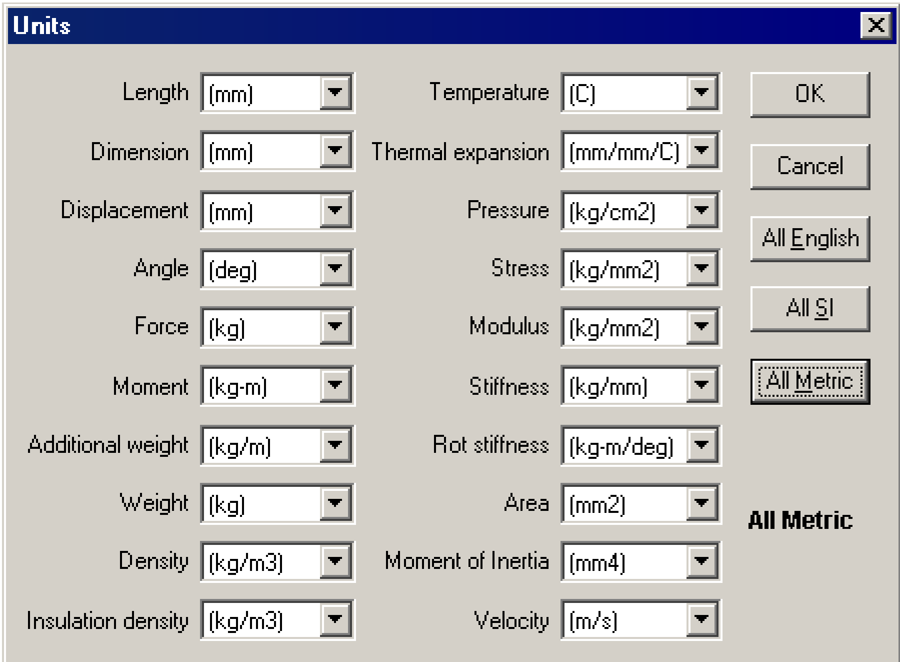
All SI

All SI units are shown below.



All Metric

All Metric units are shown below.



Note:

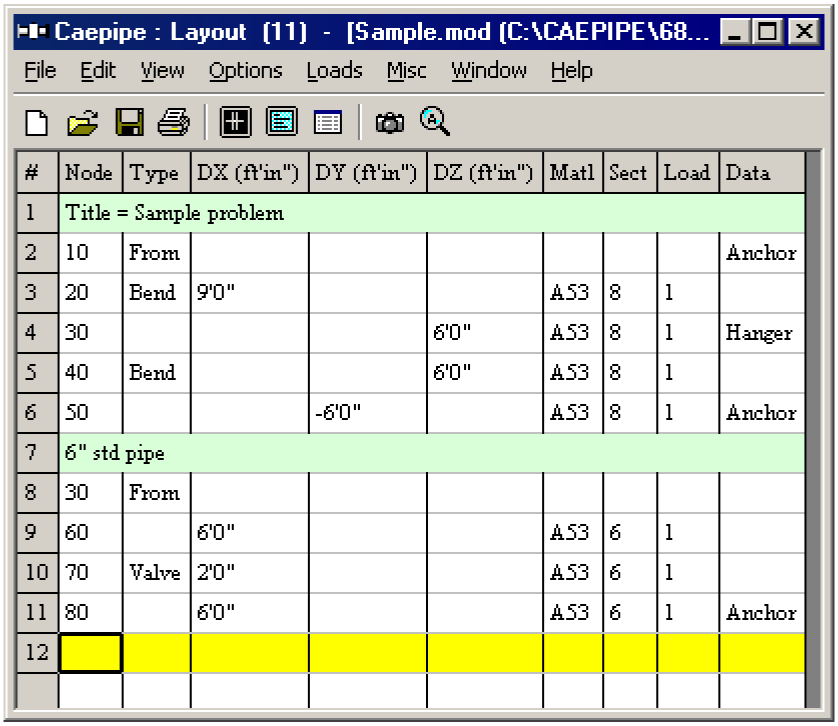
CAEPIPE input screens for Concentrated Mass, Ball Joint, Rigid Element, Valve, Bellows, Slip Joint etc. require only the "Weight" of that item to be input in lbf, kgf etc. and NOT its mass. Hence, unit for "Mass" is not listed in the Units menu. So, whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass for each item to be equal to (weight / g-value).

Similarly, for density of pipe, insulation and lining materials, CAEPIPE requires "Weight Density" to be input in lbf/in3, kgf/m3 etc. and NOT its mass density.

CAEPIPE performs all its calculations in one consistent set of units and does all unit conversions internally based on the units selected through this Units Command.

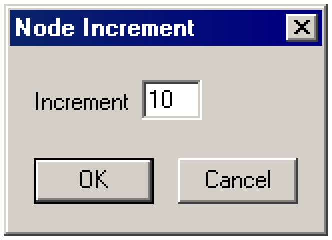
.Font.

You can display the text in all CAEPIPE text windows (Layout, List and Results) in a font face and size of your choice. Here, the font chosen is Times New Roman, 8 point:



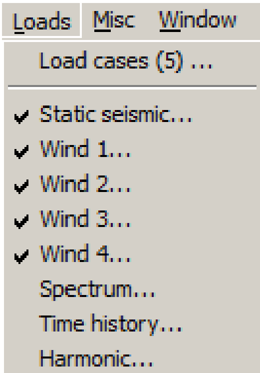
.Node Increment.

You can increase your productivity by having CAEPIPE automatically increment the node number when you model a piping system in CAEPIPE. You can turn this feature off by specifying a zero (0) increment here. The default is set to 10.



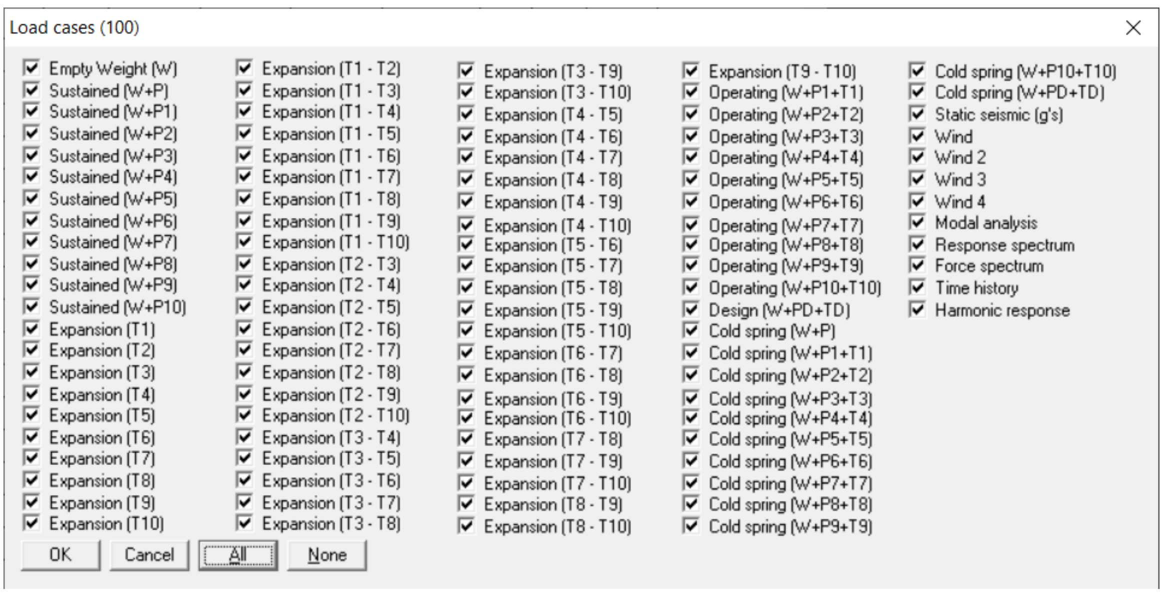
Loads Menu

From this menu, you can specify the load cases for analysis.



.Load cases.

Click on this command to select the different load cases for analysis. Most of the time, selecting a load case is as simple as checking the corresponding checkbox.



Empty Weight (W):

Empty Weight (W) analysis includes the weight of piping, components, insulation and lining but excludes the weight of content. In addition, pressure stress is excluded from the analysis.

Sustained (W+Px):

Sustained (W+Px where x = 1 to 10) analysis includes the weight of piping, components, content, insulation and lining. It also includes operating fluid pressure and its effect in the analysis.

Operating (W+Px+Tx):

Operating (W+Px+Tx where x = 1 to 10) analysis includes the weight of piping, components, content, insulation and lining. It also includes operating fluid pressure and its effect as well as the effect of temperature increase/decrease in the analysis.

Design (W+PD+TD):

Design (W+PD+TD) analysis includes the weight of piping, components, content, insulation and lining. It also includes Design and its effect as well as the effect of Design temperature increase/decrease in the analysiswhere PD = Design Pressure and TD = Design Temperature.

Cold Spring (W+PD+TD) will be shown only when a cold spring (Cut pipe element) is input into the model. When these load cases are selected for Analysis, CAEPIPE will compute and show results for Displacements, Element Forces & Moments, Support Loads and Support Load Summary. These load cases are NOT included in Stress Calculations, Rotating Equipment Qualifications and Flange Equivalent Pressure Calculations.

For a few others, load cases appear in this dialog *only after* you input their related data. For example, to perform a Response Spectrum analysis, you need to

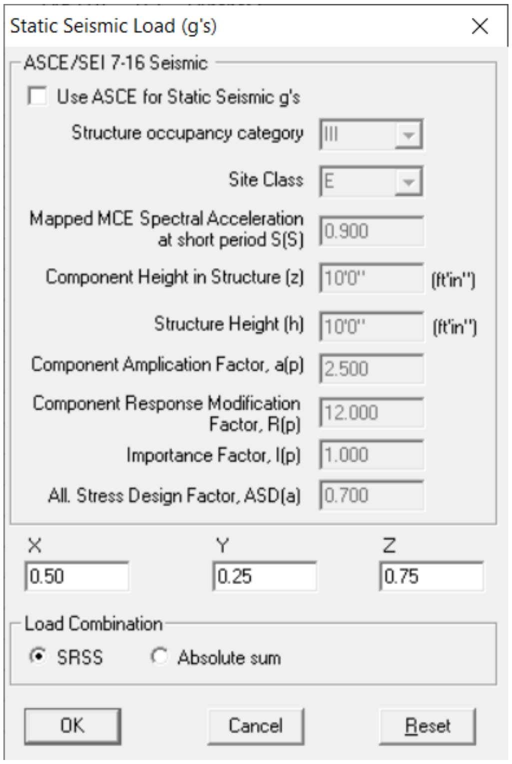
- First input Spectrums (under Misc menu)
- Then, select those spectrums under the menu Loads > Spectrum.
- Finally, you go back into menu Loads > Load Cases and check the box next to Response Spectrum for analysis.

As another example, the Cold Spring load cases do not appear in this dialog if you have not input a "Cut pipe" element in the layout window.

Further, multiple expansion and operating load cases appear here only when you set multiple thermal loads under Options > Analysis > Temperature in the Layout window.

.Static Seismic Load.

Use this load to apply static seismic loads (g-loads) computed using ASCE/SEI 7-16 or entered directly to the model. CAEPIPE applies the g-load to the piping system mass separately for X, Y and Z directions. The computed results (displacements, forces and moments) from each of the three (internal) load cases (i.e., one case corresponding to each global directional g-load) are combined using your choice of Absolute sum or Square root of sum of squares (SRSS).



CAEPIPE performs only an algebraic sum of the results for the three directional g-load cases when the Piping code = None is selected.

Static seismic is an unsigned case and you will not see a sign (+ or –) in the results for this case.

Under menu Loads > Load cases, check the box next to Static Seismic (g's) to select this case for analysis. This load is treated as an Occasional load.

The g-loads are applied only in the specified direction, i.e., a Z (g-load) of -0.189 is applied only in the -Z direction, and not in +Z direction.

A g-load input may affect the extracted natural frequencies when no piping code is selected (i.e., Piping code = None). See under "Dynamics > Cutoff Frequency" for how.

Acceleration in the vertical direction, if not provided explicitly, is generally taken to be 75% of the g-load in the horizontal directions.

The g-load values can be calculated from any of the several sources available (ASCE/SEI 7-16, ASCE A58.1, UBC, etc.). Refer to Section titled "Static Seismic Load" in CAEPIPE Technical Reference Manual for details on the implementation of ASCE/SEI 7-16 in CAEPIPE starting Version 10.30 and manual calculation as per ANSI A58.1.

Static Seismic Analysis

CAEPIPE computes the inertia force (as mass * acceleration) for each direction and applies it as an occasional load. The g-loads for the above example are 0.50g in the global X, 0.75g in the global Z, and then 0.25g in the vertical Y direction. CAEPIPE applies an X acceleration (of 0.50g) first and solves the case. This procedure is then repeated to apply accelerations in Y and Z directions as independent cases and obtain the corresponding results.

The above procedure results in three sets of solutions (displacements, element forces and moments, and support loads) to acceleration loads in X, Y and Z directions, which are typically combined in some manner. In CAEPIPE, two directional combination methods are available: SRSS (Square root of sum of squares) and ABS (Absolute).

In the SRSS method, each directional component of displacements, element forces and moments, and support loads from the three X, Y and Z acceleration analyses are squared individually and added. The square roots of these respective sums are the displacement, element force and moment, and support load at a given node. InABS method, all absolute values of each directional component of displacements, element forces and moments, and support loads are added to get the absolute values for total displacements, element forces and moments, and support loads.

The occasional stresses(S_O) are added to sustained stresses(S_L)and shown under Occasional stresses ($S_L + S_O$).

CAEPIPE does not include piping system weight in Seismic load case, a few reasons being:

1. The weight is already included in Sustained load case, and
2. Piping codes such as B31.3 specifically mention that MA and MB calculations (moments due to sustained and occasional loads respectively) be done separately before combining them to calculate Occasional stresses ($S_L + S_O$).

If you can, avoid nonlinearities such as gaps and friction when youranalysisincludesseismic loads. As it is, Static Seismic analysis is an approximation of Response Spectrum analysis, which is, in turn, an approximation of Time History analysis for a seismic event wherein seismic accelerations are applied as a function of time only at piping supports! So, the result you get from a static seismic analysis is a gross approximation.

CAEPIPE adds the support loads from a static seismic case to those from Sustained and Operating load cases to produce the combination loads at supports. Look under the Support load summary to get the different combination loads at each support.

To evaluate different directional seismic analyses:

In linear analysis (i.e., no gaps and no friction), the results from a seismic load in +X and –X directions should be the same. So, the combination ranges can be seen by studying support load summary (e.g., Sustained+Seismic, Sustained–Seismic).

In nonlinear analysis, however, the results from a seismic load in +X may NOT be the same as those from –X. In such a case, you will have to make two runs, i.e., analyze the original (say, model1.mod) with +X g-load; copy the model to model2.mod, specify a –X g-load and compute results. Now you have two sets of results, one from each model. Compare the two results carefully to identify the range (least to the most).

Or, for further post processing, export the results from each model to a CSV file via the Print > Print to Filecommand (select file type as: CSV). Import each CSV file into a spreadsheet (e.g., MS-Excel) and combine the results of interest manually.

In summary, you can set up as many runs as required for the g-loads (+ or –), depending on whether your model is linear or non-linear, and export results for those runs that you cannot combine inside CAEPIPE.

As another example, for a nonlinear model with g-loads only in the X-Y plane, you can analyze four runs (or four identical models with four different seismic loads) thus:

Run 1. Specify +X/+Y g-load
Run 2. Specify -X/-Y g-load
Run 3. Specify -X/+Y g-load
Run 4. Specify +X/-Y g-load

Examine the seismic and combination forces and moments from all runs, or in a spreadsheet (using the .CSV files CAEPIPE exports for each model) to determine the most conservative of results (and range) for further interpretation.

As another example, in a linear model with g-loads only in the X-Y plane, the two runs

Run 1. Specify +X/-Y g-load
Run 2. Specify -X/+Y g-load

are the same. So, either case (run) alone is adequate.

Or, another example, for a nonlinear model with g-loads in all three global directions, you could set up eight runs/models:

Run 1. Specify +X/+Y/+Z g-load
Run 2. Specify +X/+Y/-Z g-load

Run 3. Specify +X/-Y/+Z g-load
Run 4. Specify +X/-Y/-Z g-load

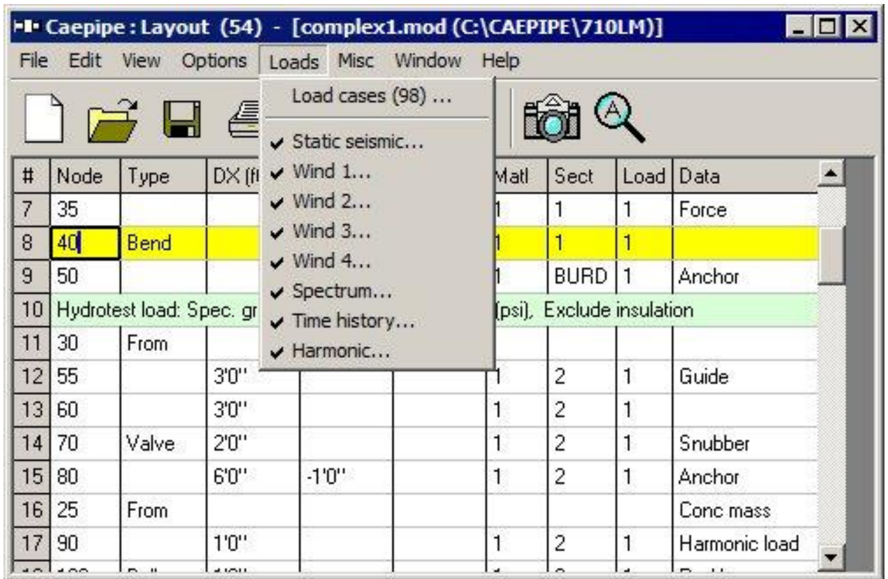
Run 5. Specify -X/+Y/+Z g-load
Run 6. Specify -X/+Y/-Z g-load

Run 7. Specify -X/-Y/+Z g-load
Run 8. Specify -X/-Y/-Z g-load

Each of these runs may yield different results. So, a careful evaluation of all eight runs is necessary to identify the most conservative solution and/or determine a range.

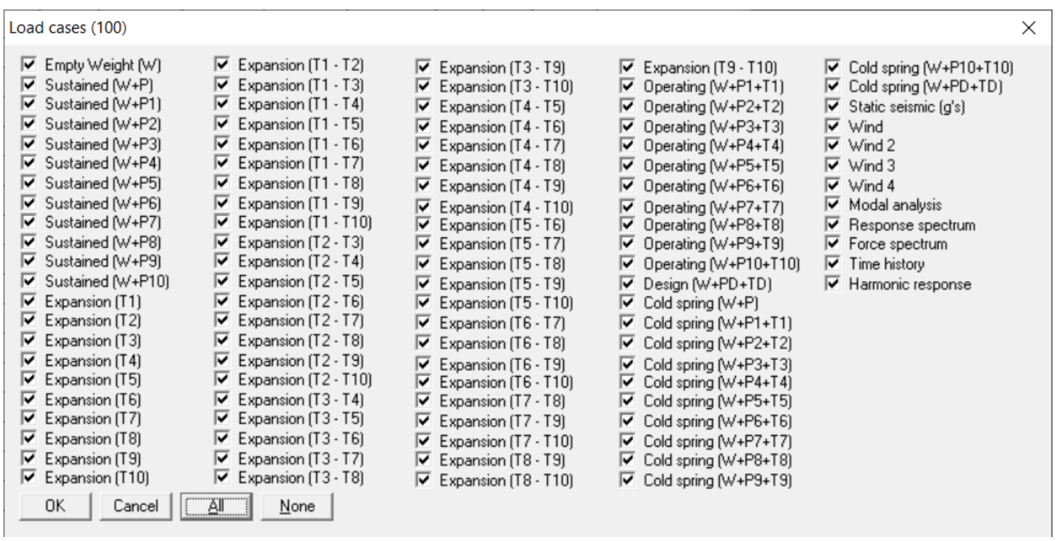
Wind Load.

You can enter up to four (4) wind load profiles namelyWind1, Wind2, Wind3 and Wind4.



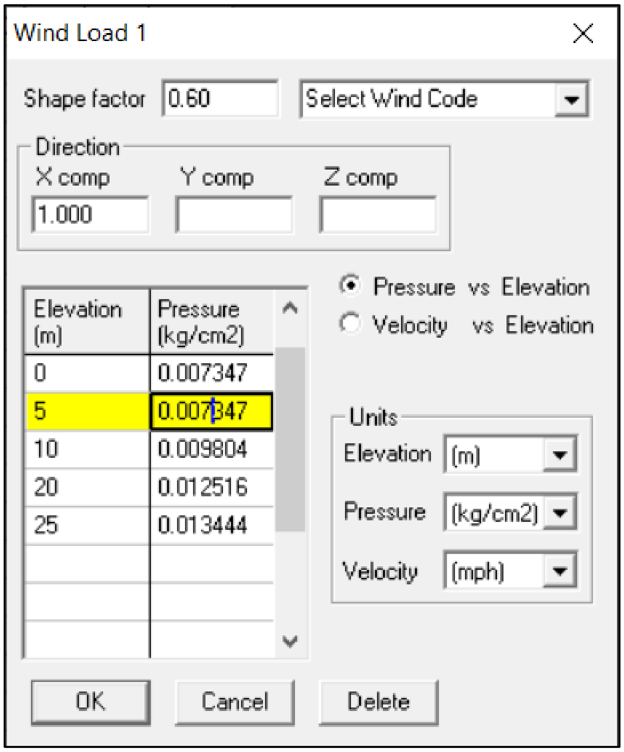
Load Combinations

With 10 thermal and four wind loads, you can analyze a total of up to 95+ load combinations (including 55 thermal load cases).



You can apply a wind load to the whole or parts of the model. CAEPIPE applies it as a lumped (concentrated force) load at the nodes (i.e., it is not a distributed load along the element). A wind profile is required for the region in which the system is installed.

The dialog for specifying wind load is given below.



When Wind Code is NOT selected then the following wind load related fields are to be specified.

Shape factor

Input a shape factor value here. CAEPIPE uses a constant value for shape factor. For example, ANSI A58.1-1982, "Minimum Design loads for Buildings and other Structures"provides a shape factor of 0.6 for circular cross-section.

When user selects Pressure versus Elevation, CAEPIPE does not use the shape factor.

Direction

Input the direction of the wind using the direction cosines (examples: for wind in Z direction, Z comp = 1; for wind in 45° X-Y plane, X comp=1, Y comp=1, Z comp=0). See section on Direction in the Technical Reference Manual.

Wind or Pressure Profile

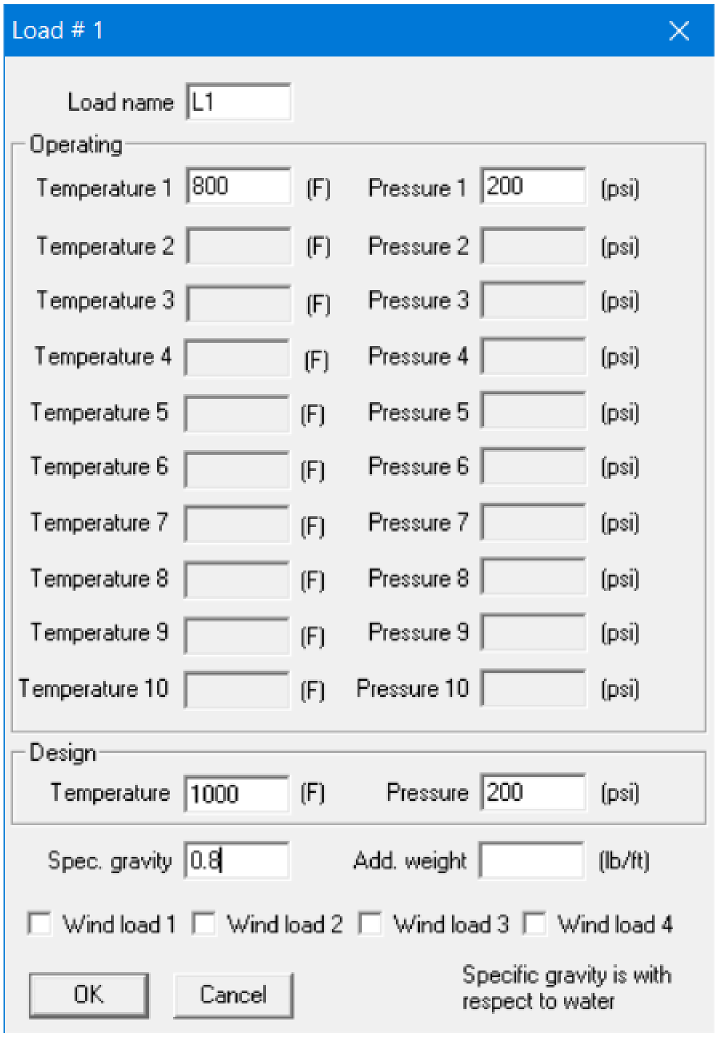
You need to input values for wind velocity or pressure at different elevations. *See Subsection titled "Wind Load" in the CAEPIPE Technical Reference Manual for details on how to computeWind Forces as per ANSI A58.1-1982.*

CAEPIPE computes the wind load on an element by interpolating the wind forces between the elevations.

Units

You can specify any combination of units for elevation, pressure and velocity.

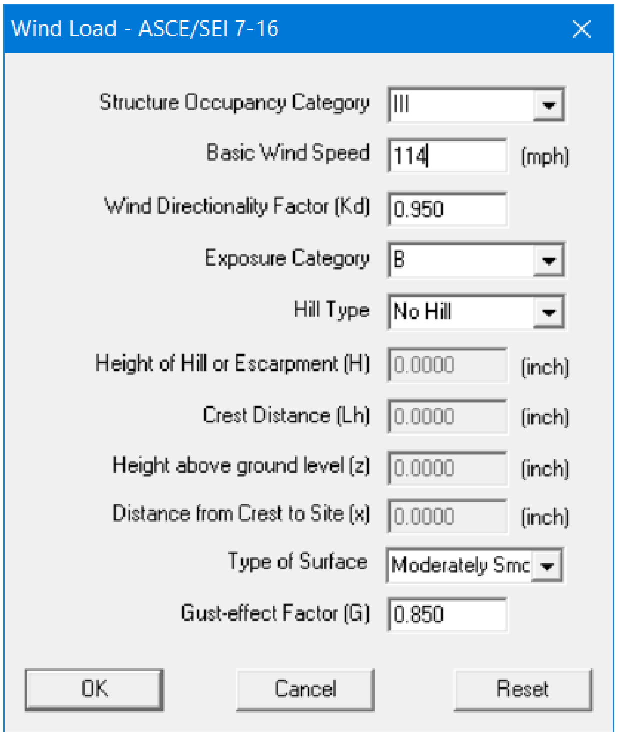
To **exclude** an element(s) from wind loading, the Load definition dialog for that element(s) should not have Wind load checked. In the dialog below, Wind load is checked.



ASCE/SEI 7-16

Starting Version 10.30, CAEPIPE has the built-in feature to compute the Design Wind Forces as per ASCE/SEI 7-16, when this code is selected. For using this feature, basic wind parameters are required to be defined for the stress system. Selecting this code without defining the basic wind parameters listed above will show a popup as shown below.

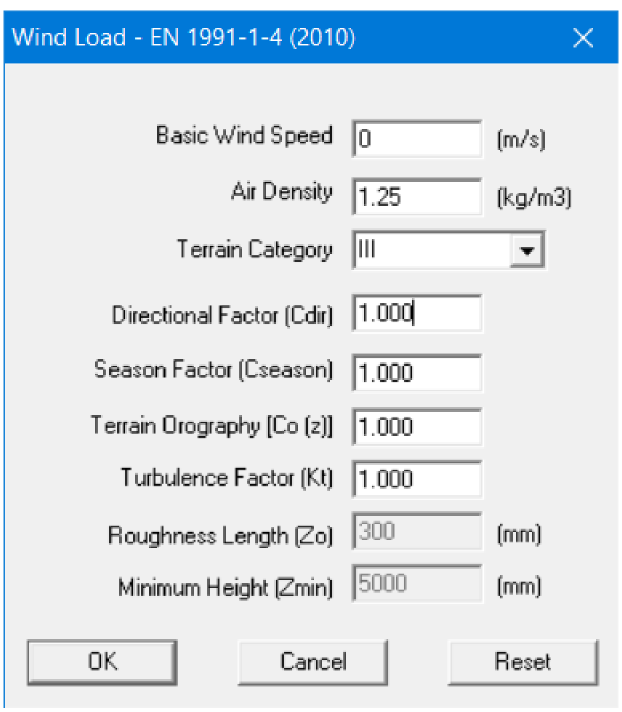
Refer to Section titled “Wind Load” in CAEPIPE Technical Reference Manual for details on implementation of ASCE/SEI 7-16 in CAEPIPE starting Version 10.30.



EN1991-1-4 (2010)

Starting Version 10.30, CAEPIPE has the built-in feature to compute the Design Wind Forces as per EN 1991-1-4 (2010), when this code is selected. For using this feature, basic wind parameters are required to be defined for the stress system. Selecting this code without defining the basic wind parameters listed above will show a popup as shown below.

Refer to Section titled "Wind Load" in CAEPIPE Technical Reference Manual for details on implementation of EN 1991-1-4 (2010) in CAEPIPE starting version 10.30



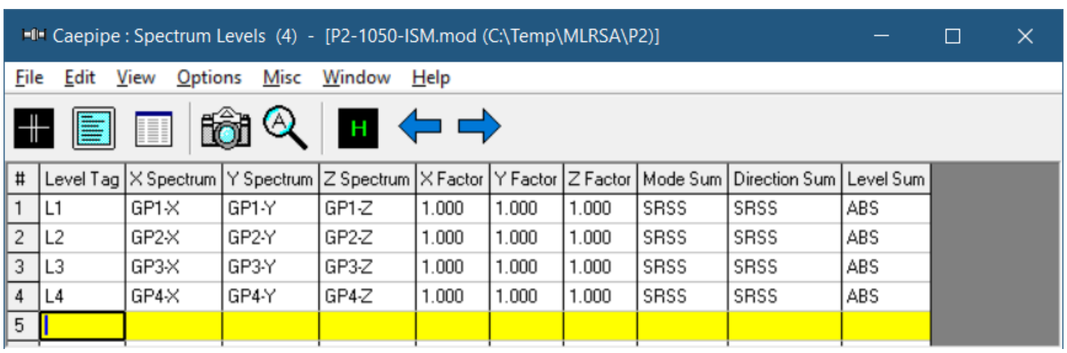
.Spectrum Load.

Shock waves (due to a seismic event), where wavelengths could be long, travel through the soil causing ground motion. Every support of a piping system connected to the ground will then experience the same excitation (hence "uniform" excitation at those supports).

On the other hand, if the piping is routed along the wall of a tall building, then the seismic motion at the ground level will cause higher excitations at higher elevations of the building. In this case, the piping supports at higher elevations will be subject to higher excitations. It is fairly common in the industry to "envelope" the excitations at different elevations to arrive at one "uniform" excitation for all supports of a piping system. Such "uniform" excitation is then specified in CAEPIPE using a response spectrum, which is a table of maximum response versus natural frequency (with damping) for single degree-of-freedom systems.

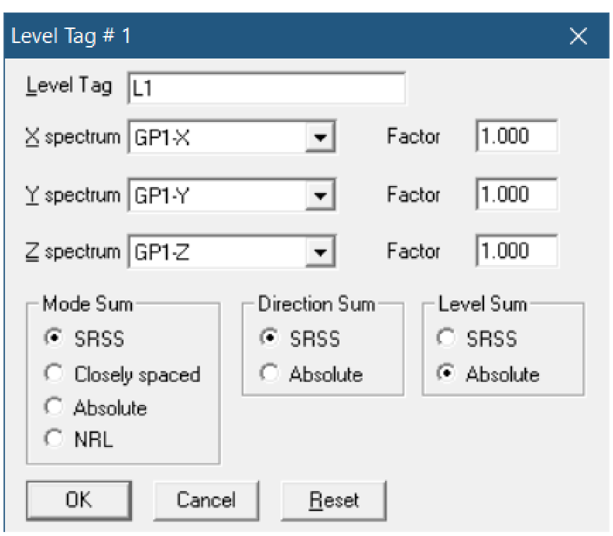
Starting Version 10.50, CAEPIPE can also perform Multi-level Response Spectrum Analysis of piping systems experiencing different spectrum loads at different supports. The supports could be at different elevations or at the same elevation, grouped under different Spectrum level tags as shown below.

The response spectrum analysis can only be initiated after at least one spectrum level is defined (under menu Load > Spectrums). If the pipe is supported on multiple levels, with different levels experiencing different spectrum loads, each of the spectrum levels should be defined here. The same level tag should be assigned to all the supports located at that particular level.



#	Level Tag	X Spectrum	Y Spectrum	Z Spectrum	X Factor	Y Factor	Z Factor	Mode Sum	Direction Sum	Level Sum
1	L1	GP1-X	GP1-Y	GP1-Z	1.000	1.000	1.000	SRSS	SRSS	ABS
2	L2	GP2-X	GP2-Y	GP2-Z	1.000	1.000	1.000	SRSS	SRSS	ABS
3	L3	GP3-X	GP3-Y	GP3-Z	1.000	1.000	1.000	SRSS	SRSS	ABS
4	L4	GP4-X	GP4-Y	GP4-Z	1.000	1.000	1.000	SRSS	SRSS	ABS
5										

Double click on the row to assign the spectrum load. Then, select the spectrum that can only be input using the drop-down menu from the spectrums defined under menu Misc> Spectrums.



Level Tag

Type an alpha-numeric name (up to 8 characters long) in this field. Example: If you have three elevation levels (ground, 100m and 300m) with different spectrum loads, you could name them as L000, L100 and L300. If a single level is defined for uniform spectrum analysis, the same level tag will be automatically assigned to all supports. For multi-level response spectrum analysis, the level tags defined here need to be assigned to each support individually or assigned to the group of supports using the Layout window > Edit > Change.

X, Y and Z spectrums

Select a spectrum from the drop-down combo box, which should have been input in the spectrum table for each global direction. The spectrum tables can be input in Layout Window Menu: Misc > Spectrums.

Factor

The multiplying (scale) factor for the spectrum is input here. The same spectrum may be multiplied by different (Scale) factors to apply spectrum loads for different dynamic events.

Mode Sum

Pick one of three choices, SRSS (square root of sum of squares), Closely spaced or Absolute. See section on Dynamic Analysis in Technical Reference Manual for more information.

Direction Sum

Pick one of two choices, SRSS (square root of the sum of squares) or Absolute. See section on Dynamic Analysis in Technical Reference Manual for more information.

Level Sum

Pick one of two choices, SRSS (square root of the sum of squares) or Absolute. Level Sum is only enabled when multiple spectrum levels are assigned. See section titled Dynamic Analysis in Technical Reference Manual for further details.

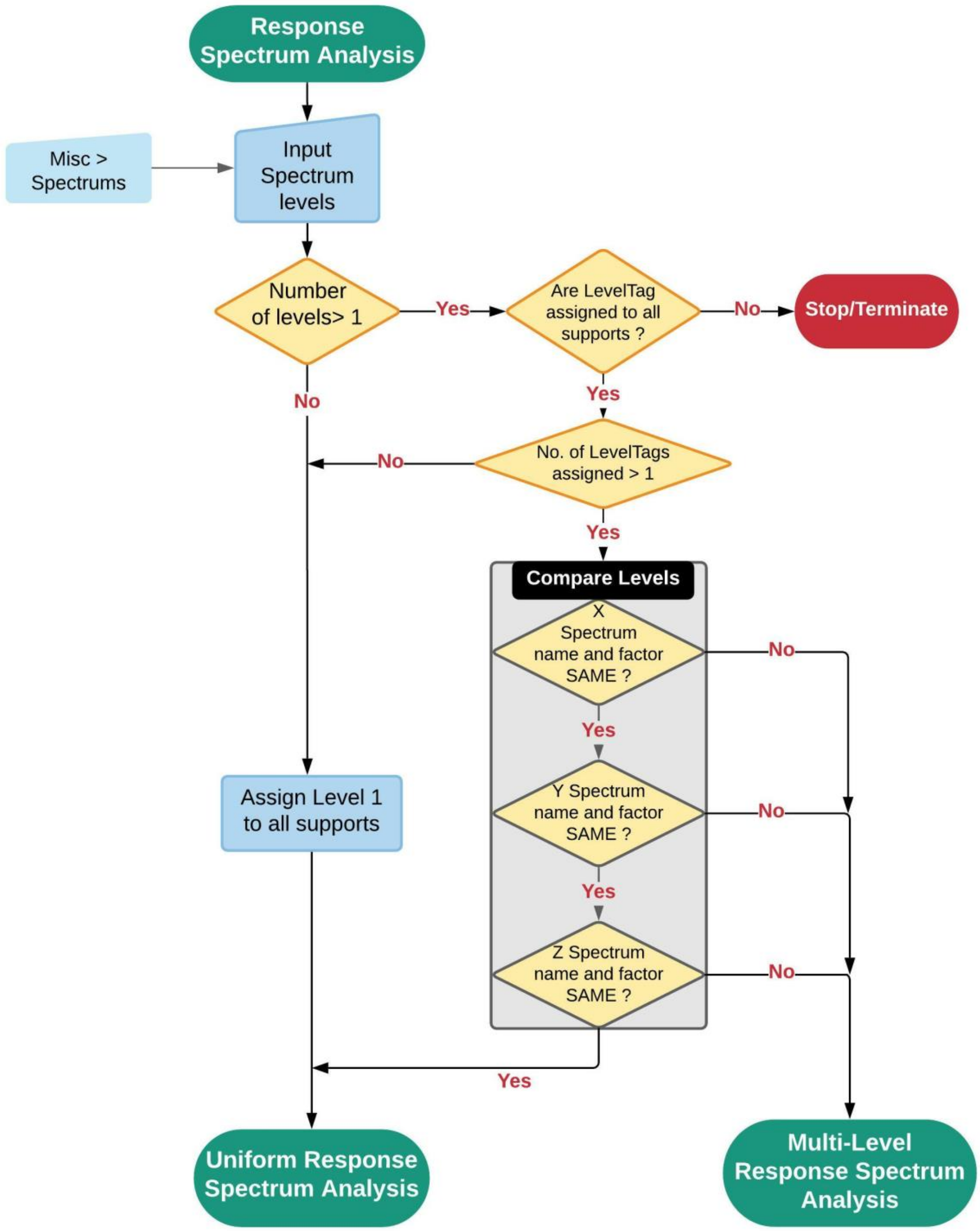
Uniform Response Spectrum and Multi-Level Response Spectrum Analysis

CAEPIPE will automatically switch between Uniform Response Spectrum and Multi-level Response Spectrum depending on the loads defined.

1) If a single spectrum level is defined, the spectrum load will be applied simultaneously at all piping supports, and CAEPIPE will perform a "Uniform response spectrum" analysis. It will compute the modal and directional responses (to this uniform excitation), which are further combined in a manner you specify.

2) If more than one level is assigned to supports, then CAEPIPE will further check if the spectrum loads are the same or different. Please refer to the flowchart below for details on the decision-making process by CAEPIPE. The Multi-level Response Spectrum Analysis will be initiated when more than one spectrum load is assigned to the supports. The combination over level contributions will be performed first, followed by interspatial and then intermodal combination.

Note: CAEPIPE will compare only the "Name" of the spectrum and "Factor" in deciding whether Uniform or Multi-level Response Spectrum Anaylsis is to be performed as shown in the flow chart below.



.Time History Load.

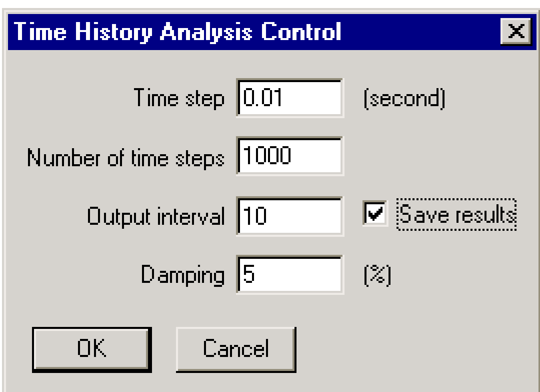
A phenomenon that causes loads to vary with time can be applied in CAEPIPE for time history analysis. You may hand-calculate these values or use a simulation program (such as a transient fluid flow program for fluid hammer analysis) to get the variation of forces or moments with time at different points in the piping system. These loads are then input into CAEPIPE as time functions, which are later applied at the corresponding nodes of the CAEPIPE model as "Time Varying Loads."

Time functions (input under menu Misc> Time Functions) are a series of non-dimensional values versus time, which describe the variation of the forcing function with time. The actual value of the time function at any time is found by linear interpolation between time points.

When you input a "Time Varying Load" at a node, you may apply a scale factor, if necessary, along with the direction of the force or moment.

CAEPIPE will then apply these loads to compute the response of the piping system by performing a time history analysis.

Various parameters for time history analysis are specified in the Time History Analysis Control dialog under menu Loads > Time History.



Time step

The time step (time interval) at which the analysis is performed should be typically no more than 10% (smaller the better) of the period of the highest frequency of interest, i.e., higher the frequency, smaller the time step, e.g., for a 33 Hz maximum frequency, the time step would be about 0.003 seconds {=10% of (1/33)}.

Number of time steps

The time history response is calculated for a total time (in seconds) of Time step × Number of time steps. This is how long you want to study system response. The total time may exceed the range of available data in the time function. The time function is only a forcing function. But, CAEPIPE can compute system response to it well after the forcing function ceases. For example, the effect of a heavy steam hammer could linger on for a minute while your forcing function data could span only 8s (seconds).

Output interval

The output interval is a multiple of time step at which you want CAEPIPE to save (and later display) results. For example, assuming a time step of 0.01s, if you wanted to see results at every 10th time step (i.e., at 0.1s, 0.2s, 0.3s, and so on), enter 10 for output interval. You need to check the "Save Results" checkbox to see these time varying results which are saved in the file *modelname.rth*. If the "Save Results" checkbox is not checked, only the enveloped results are available but not their variation in time. The .RTH files may be large for big models with many time steps (so ensure that you have adequate storage and permission settings before performing time history analysis with "Save Results" checked).

Note that CAEPIPE calculates the enveloped (maximum) results at every time step and not just at every output interval. The output interval is strictly for display purpose in the results and for saving in the .RTH file. The output interval does not affect the analysis.

Damping

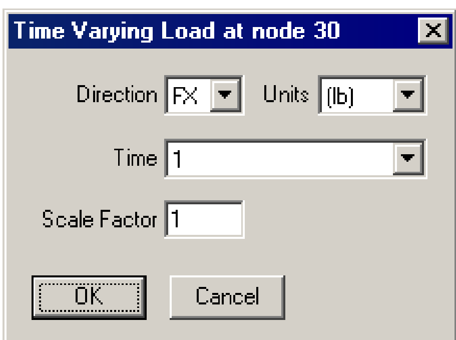
Express the damping factor as a percentage (not as a fraction). Enter 5, not 0.05, for 5% damping.

Example:

Input: Time step = 0.01s, Number of time steps: 1000, Output interval: 10, Damping: 5%, Save Results checkbox Checked.

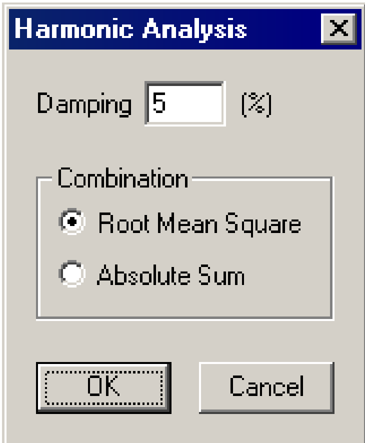
CAEPIPE calculates the solutions at every 0.01s up to 10s. The results are saved every 0.1s in the .rth file. Note however, that CAEPIPE calculates the enveloped (maximum) results at every time step interval and not just at every output interval. The output interval is strictly for display purpose in the results and does not affect analysis.

The Time Varying Load dialog is shown below:



The Time Varying Load at a node is specified by its direction, units, the associated time function and a scale factor. This scalefactor is a scalar value, which when multiplied by the non-dimensional time function will give the actual magnitude of the forcing function in the unit selected in the above dialog.

.Harmonic Load.



Harmonic loads can be loads from any sinusoidal loading, such as from rotating equipment or reciprocating pumps on a line. The magnitude of the loading should be determined before analysis. If only one compressor is on a line, then only one harmonic load is input. If more than one load is acting on the same line, then the phase (angle) or the separation in timing of application of each harmonic load becomes important (consider for example that the two loads may be equal and opposite thus canceling out any dynamic imbalance, or the two loads can be in the same direction, say +X and separated by 30° phase angle). So, the situation must be carefully analyzed before imposing these loads.

Harmonic analyses of linear structures are generally performed to determine the steady-state response to long-duration loads, which vary sinusoidally (harmonically) with time, thus enabling you to verify whether your designs will successfully overcome resonance, fatigue, and other harmful effects of harmonic vibrations. In a harmonic analysis, all long-duration loads and the structural response vary sinusoidally at the same frequency.

Various parameters for harmonic analysis are specified in this dialog shown above.

Damping

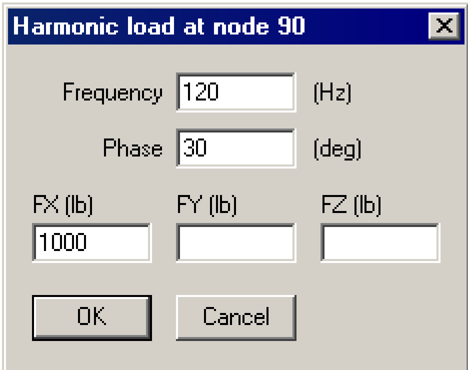
CAEPIPE applies this factor to all the modes. Enter value of critical damping as a percent, not as a fraction (Example: For 5%, enter 5, not 0.05).

Combination

CAEPIPE combines modal responses using one of two methods: Root mean square (RMS) or Absolute sum (which may be too conservative). Pick one.

After you input the data here, then apply Harmonic loads at the Data column of the corresponding nodes of interest in the piping model. Then, select Harmonic under the menu Loads > Load cases for analysis.

The Harmonic Load dialog is shown below:



The harmonic load can be imposed as a Force (FX/FY/FZ) at a specified frequency and phase angle. You may be able to get more information on the harmonic loading (mass, rpm, eccentricity, etc.) from the manufacturer of the equipment.

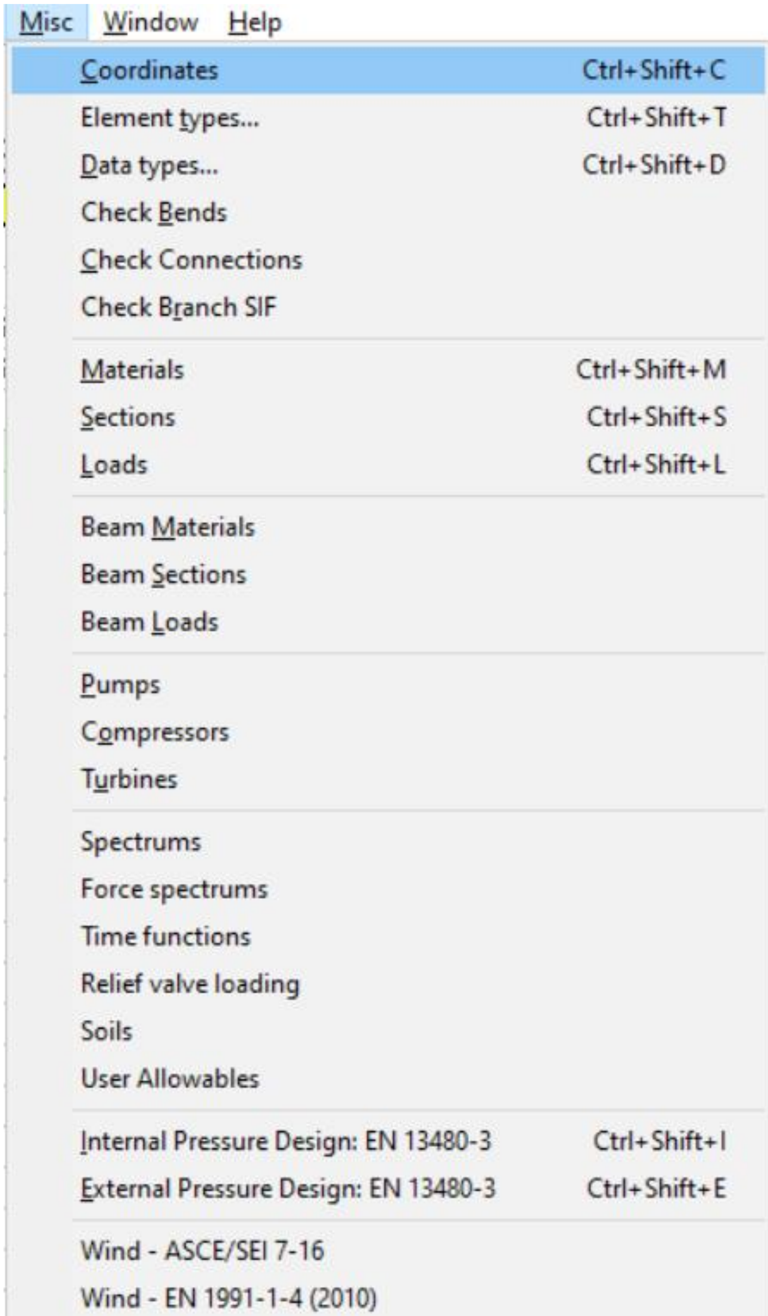
Since a harmonic load response is unsigned, results do not have signs (+ or –).

.Hydrotest Load.

Hydrostatic test load can be entered as a specialized form of a comment. Refer to the section titled "Comment" for details.

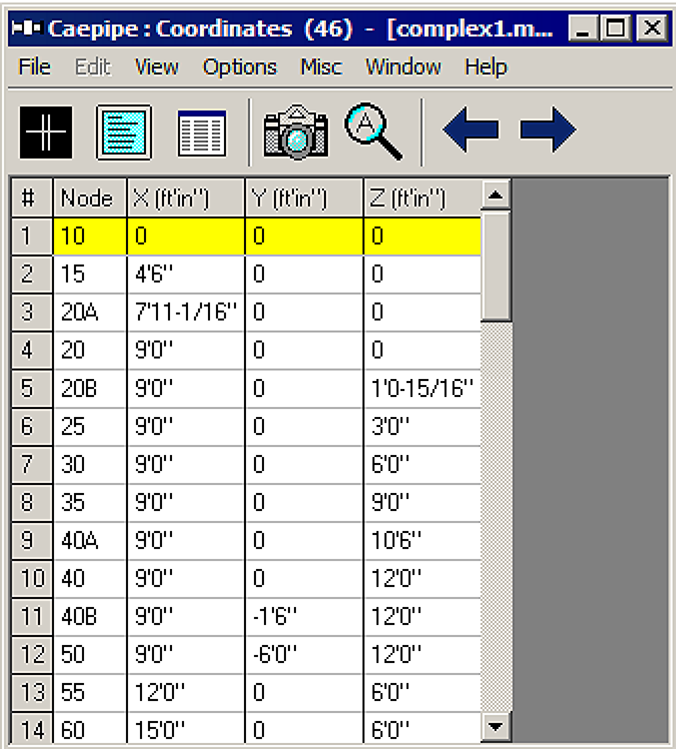
Misc Menu

A few utilities such as Check bends, Check Connections and Check Branch SIF, sundry items such as opening the list screens for materials, sections, loads, pumps, compressors and turbines, and commands to enter data for spectrums, time functions, etc., are among the commands on this menu. When Piping code is set to "None" under menu Options > Analysis, all items from Pumps to User Allowables excluding Soils are disabled.

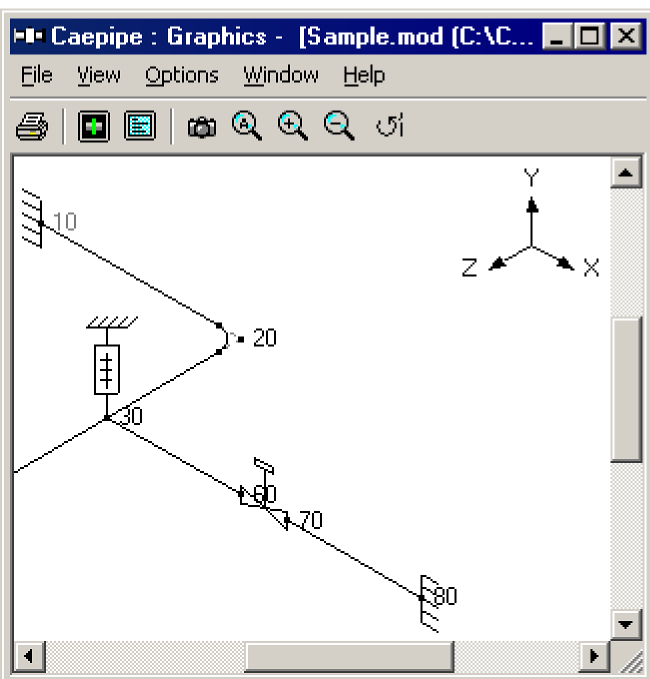


.Coordinates (Ctrl+Shft+C).

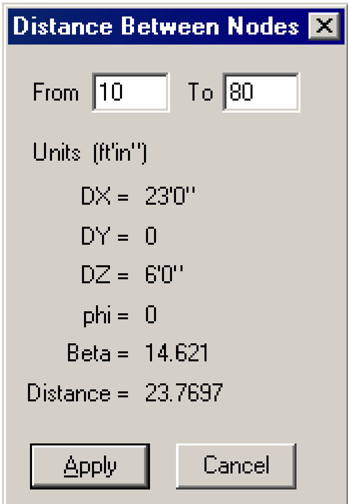
This command lists the coordinates of all the nodes (including internally generated nodes such as A, B nodes) in the model.



#	Node	X (ft'in")	Y (ft'in")	Z (ft'in")
1	10	0	0	0
2	15	4'6"	0	0
3	20A	7'11-1/16"	0	0
4	20	9'0"	0	0
5	20B	9'0"	0	1'0-15/16"
6	25	9'0"	0	3'0"
7	30	9'0"	0	6'0"
8	35	9'0"	0	9'0"
9	40A	9'0"	0	10'6"
10	40	9'0"	0	12'0"
11	40B	9'0"	-1'6"	12'0"
12	50	9'0"	-6'0"	12'0"
13	55	12'0"	0	6'0"
14	60	15'0"	0	6'0"



Straight line distance between two nodes can be found by using the menu View > Distance or by the command (Ctrl+D) in the coordinates window.



.Element types (Ctrl+Shft+T). / .Data types (Ctrl+Shft+D).

You can open these dialogs that contain the different elements or data items you can input at a node. You can also open the same dialogs by clicking on the header row in the Layout window or right clicking in the Type or Data column on an empty row.

More information about these commands is available under Layout Window in the Technical Reference Manual.

.Check Bends.

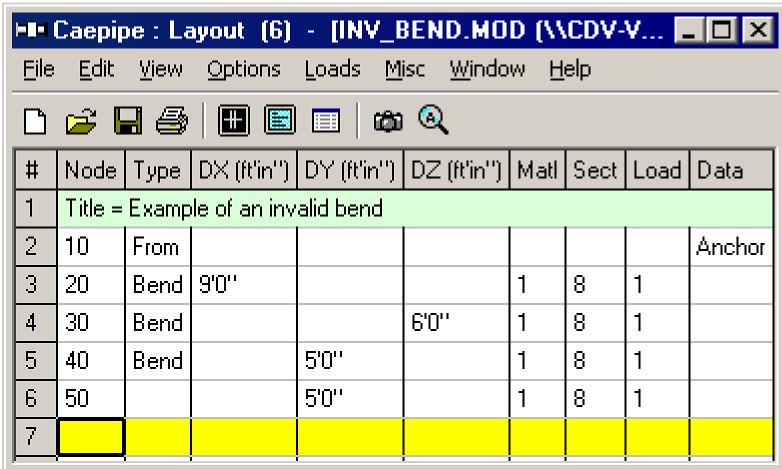
CAEPIPE can identify incorrectly and incompletely modeled bends. If you have any, you must correct them before analysis.

Case 1 (Incorrect modeling)

An "Invalid Bend" occurs if the bend geometry is input incorrectly in the layout. For example, for 90° bends, the bend radius should be less than or equal to the shorter of the two lengths, namely from the previous node to the bend node (same as "tangent intersection point") or from the bend node to the next node.

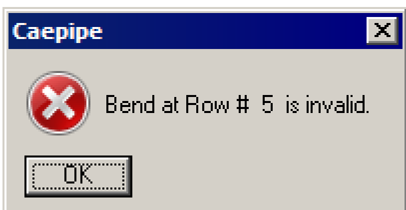
Case 2 (Incomplete modeling)

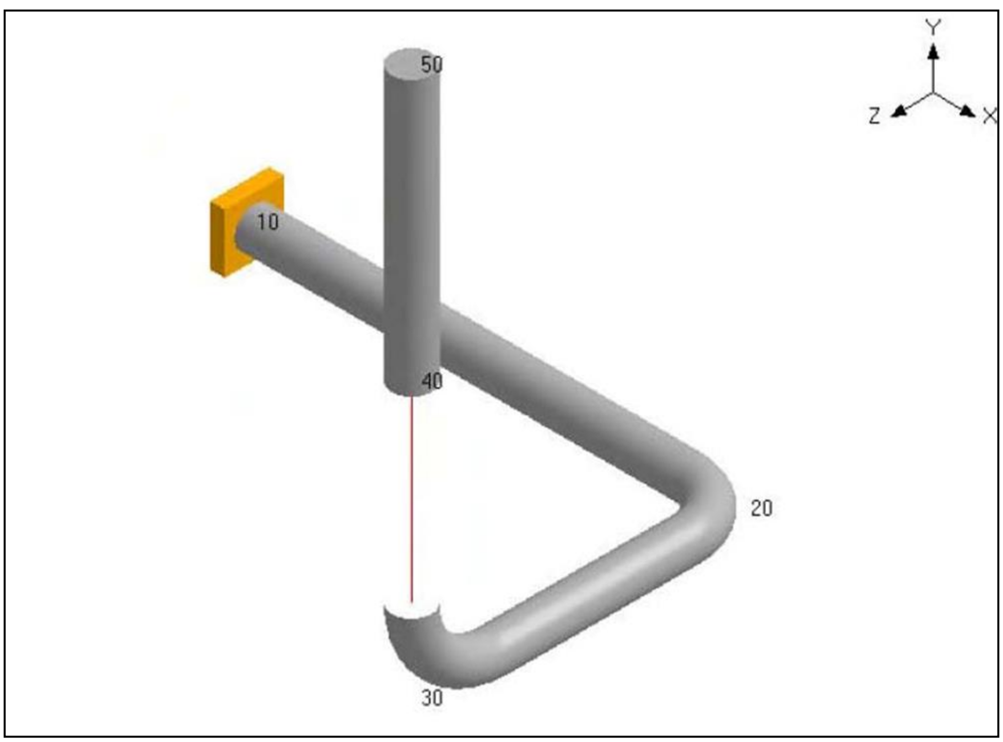
A bend needs to be given a change in direction from the previous direction on the row below it. If this change is either not specified or incorrectly specified, CAEPIPE flags this bend as an invalid bend. An example follows.



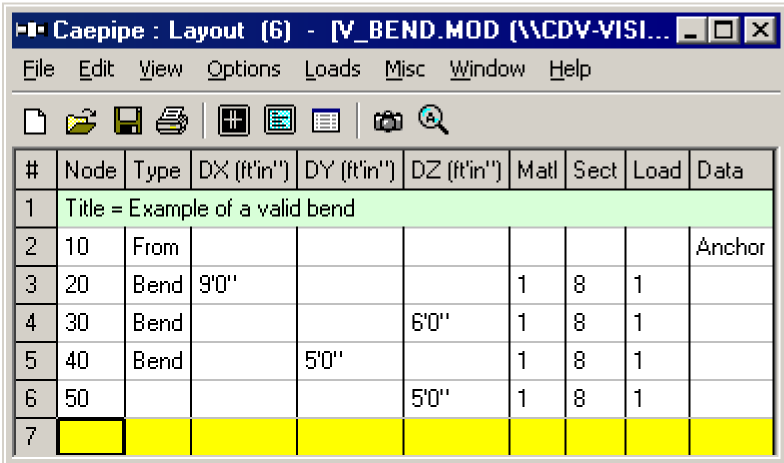
#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example of an invalid bend								
2	10	From							Anchor
3	20	Bend	9'0"			1	8	1	
4	30	Bend			6'0"	1	8	1	
5	40	Bend		5'0"		1	8	1	
6	50			5'0"		1	8	1	
7									

When you select the Check bends command under the Misc menu, you get the following message for this model.



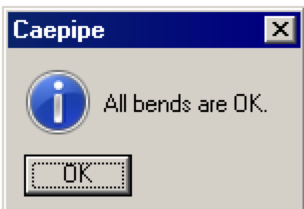


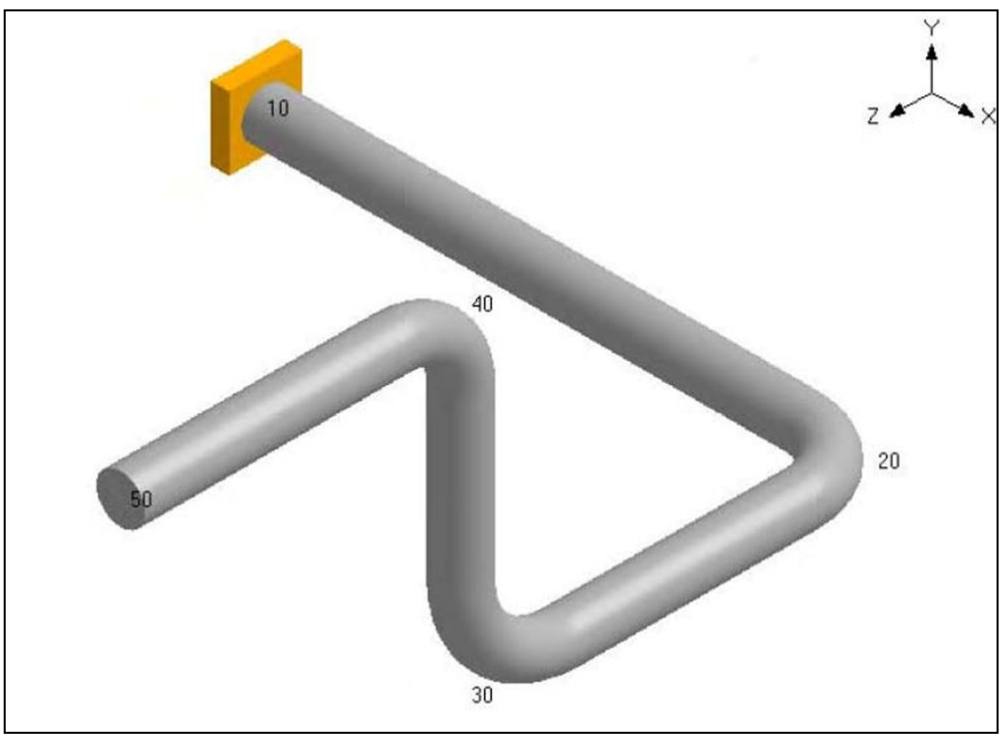
By studying the image and the Layout window above, you can see that the bend at node 20 turns from X direction into Z direction, and the bend at node 30 turns from Z direction into Y direction. But, the bend at node 40 incorrectly turns from Y direction into Y direction. To correct this, this bend needs to turn into any direction other than Y. The following screen shots show the correction. The bend at node 40 now turns into Z direction.



#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Example of a valid bend								
2	10	From							Anchor
3	20	Bend	9'0"			1	8	1	
4	30	Bend			6'0"	1	8	1	
5	40	Bend		5'0"		1	8	1	
6	50				5'0"	1	8	1	
7									

When you run the same Check bends command again, you get the following message.

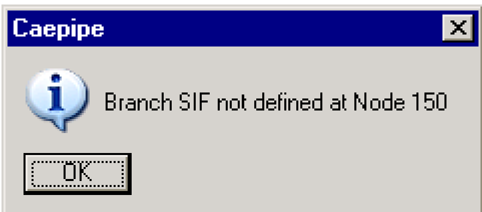




.Check Branch SIF.

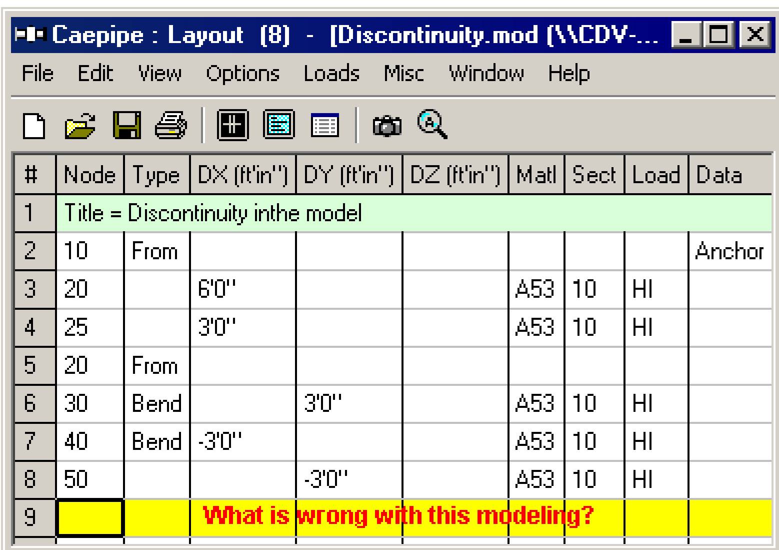
CAEPIPE will validate Branch SIFs defined in the stress model against those SIFs applicable for the selected piping code before saving the model. If any Branch SIF defined in the stress model does not match with those applicable for the piping code selected, an option is given to change that Branch SIF before saving the model.

In addition to the above check, this feature will check the layout for Branch Points without Branch SIFs defined and report the same to the user as shown below.

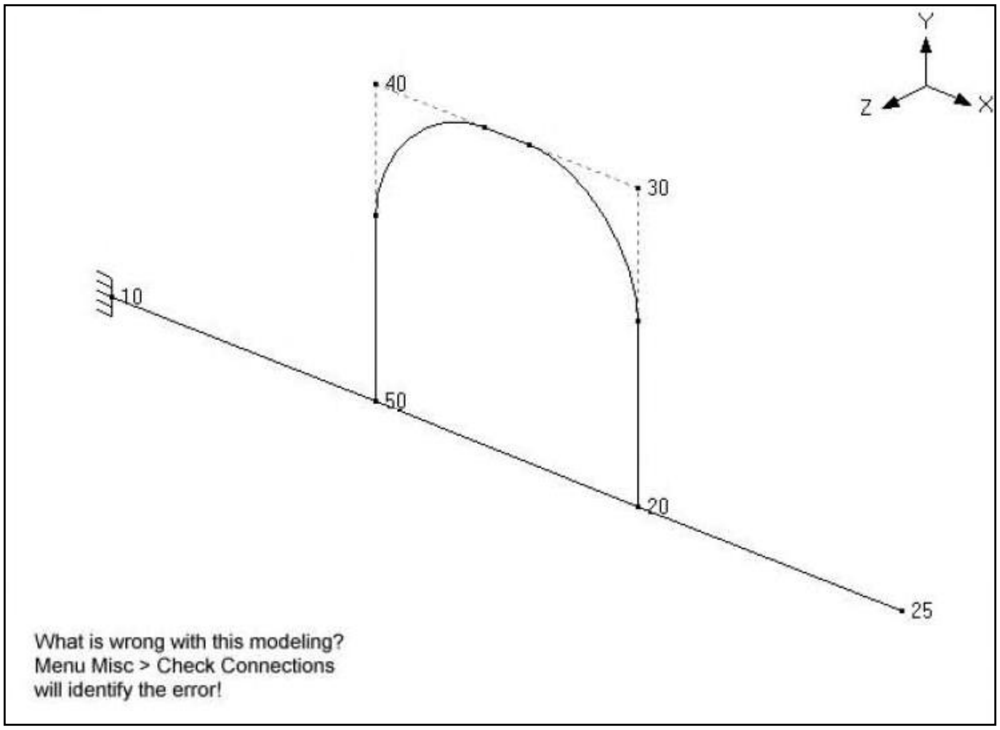


.Check Connections.

While not foolproof, this command checks for missing connections between pipes. By studying the following two figures, can you identify what is wrong with the modeling shown below? Check connections command will.

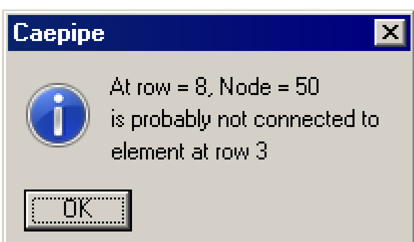


#	Node	Type	DX (ft'in")	DY (ft'in")	DZ (ft'in")	Matl	Sect	Load	Data
1	Title = Discontinuity inthe model								
2	10	From							Anchor
3	20		6'0"			A53	10	HI	
4	25		3'0"			A53	10	HI	
5	20	From							
6	30	Bend		3'0"		A53	10	HI	
7	40	Bend	-3'0"			A53	10	HI	
8	50			-3'0"		A53	10	HI	
9			What is wrong with this modeling?						



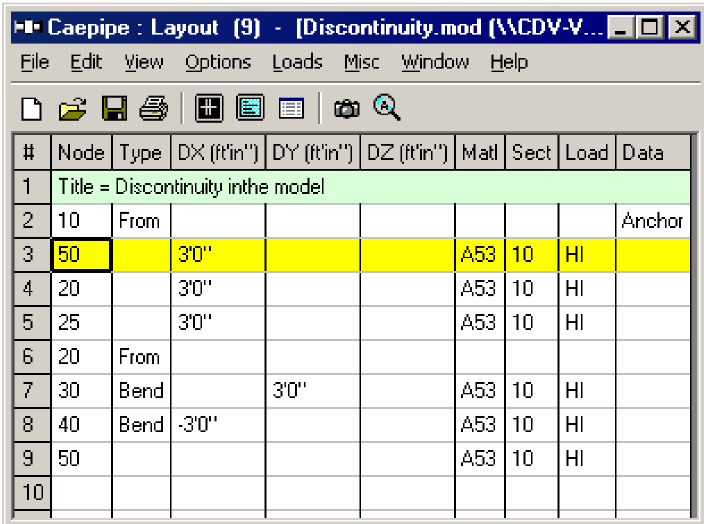
By studying the previous Layout window, you will notice that even though node 50 appears to be on the header (node 10 to node 25), it actually is not. Node 10 connects directly to node 20 which further connects to node 25. The loop begins from node 20 and goes up to node 50. But node 50 is not defined on the header. So, there is no connection between the loop and the header!

When you run the Check connections command, you get the following message.



So, you might want to run this command on complicated models with loops just to ensure that all such connections are properly modeled.

Corrected modeling is shown next. Notice that node 50 is now defined as part of the header from node 10 to node 25.



.Materials, Sections and Loads.

Please see the respective sections with the same titles in the Technical Reference Manual.

.Beam Materials, Beam Sections and Beam Loads.

These three can be found under Beam in the Technical Reference Manual.

.Pumps, Compressors and Turbines.

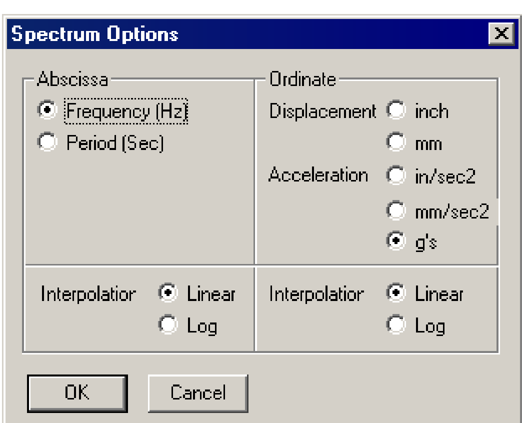
Please see the respective sections with the same title in the Technical Reference Manual.

.Spectrums.

A (uniform response) spectrum is a table of maximum response versus natural frequency for a specific excitation in single degree-of-freedom systems. You can input spectrums in three ways:

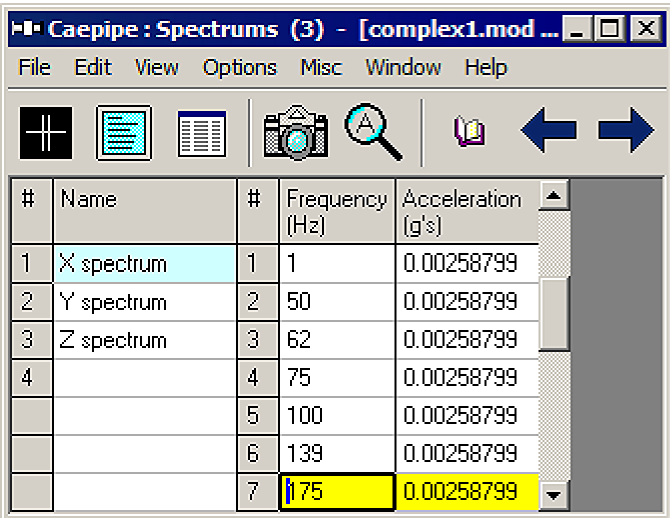
1. Input spectrums directly into the model.
2. Create a spectrum library and load spectrums from it.
3. Input spectrums from a text file.

When you use the first two methods, you may use menu Options > Spectrum command to set the different units for the X- and the Y-axes, and also choose the interpolation method.



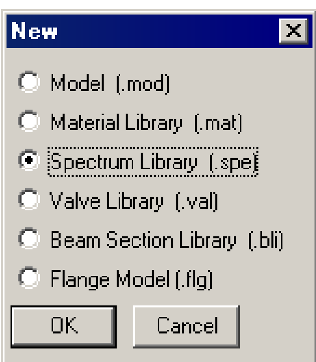
1. Input spectrums directly into the model

Select Spectrums from the Misc menu. You are shown the List window for spectrums. Start typing pairs of values into it. The frequencies or periods do not have to be in any order; CAEPIPE will sort them later. You can input as many pairs of values as required.

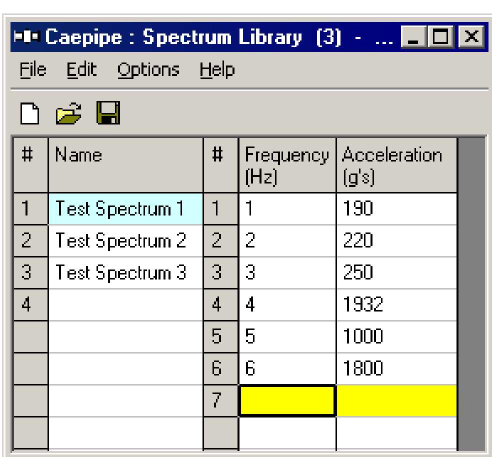


2. Create a spectrum library and load spectrums from it

From the Main, Layout or Results window, select menu File > New.



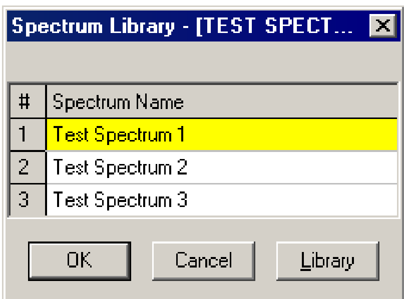
Select Spectrum library. The spectrum library List window is shown. Start typing pairs of values into it.



The frequencies or periods you type can be in any order. CAEPIPE will sort them later. You can input as many pairs of values as required. Be sure to save the file (it will be saved to a filename you specify, with a .spe extension) using the Save command in the File menu.

Note: Fourteen (14) Spectrum Libraries have been added corresponding to EL Centro (May 18, 1940), Uniform Building Code (1991 Edition) and Nuclear Regulatory Commission (NRC) Guide 1.60 [July 2014, Revision 2]. Refer to Appendix B for more details.

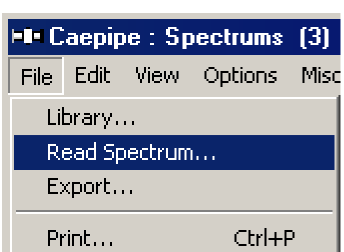
Now, open the CAEPIPE model that needs these spectrums. Open the saved library (menu File > Library) and input spectrums into the model from the shown list.



3. Input spectrums from a text file

This method involves creating a separate text file (in the format shown below) for each spectrum you want to input. The steps are given below.

a) For each spectrum, on the first line, type the Name of the spectrum (up to 31 characters). On the second line, type units for Abscissa (X-axis) and Ordinate (Y-axis) axes, separated by a space.
b) After these two lines, on each line, you can input a value pair. You can input as many of these value pair lines as required.
c) Each spectrum should be saved to a separate text file. More than one spectrum should not be input in one text file as it cannot be read into CAEPIPE.
d) Now, open the CAEPIPE model that needs these spectrums. Select Read Spectrum from the File menu, read all text files you created one after the other.



The format of a spectrum text file is shown below. The spectrum that is read appears on the row where the yellow highlight is placed (under the Name column). You can use the Edit menu commands to insert an empty row or delete an existing spectrum. Ensure that no two spectrums share the same Name. CAEPIPE issues a warning should such occur.

The spectrum text file should be in the following format:

```
Name   (up to 31 characters
Abscissa units    Ordinate units
Abscissa value1   Ordinate value1
```

```
Abscissa value2   Ordinate value2
        .                  .
        .                  .
        .                  .
Where
Abscissa units    0 = Frequency (Hz)
                  1 = Period (Sec)

Ordinate units    0 = Displacement (inch)
                  1 = Displacement (mm)
                  2 = Acceleration (in/sec2)
                  3 = Acceleration (mm/sec2)
                  4 = Acceleration (g’s)
```

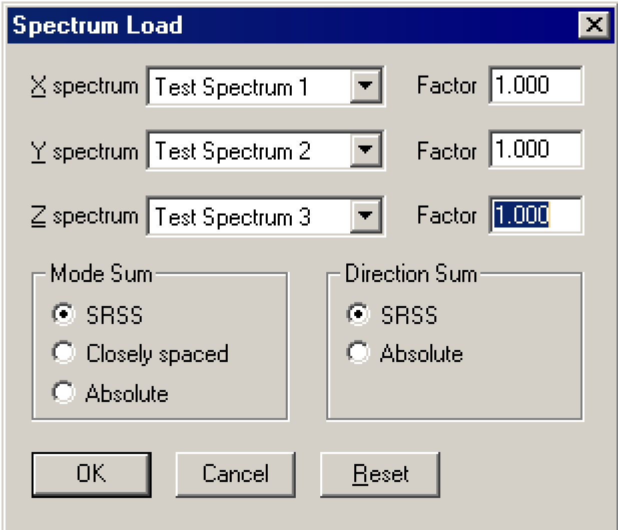
Example file

```
Test Spectrum
0   2
1   190
2   220
3   250
4   1932
5   1000
6   1800
.   .
.   .
```

Analysis note:

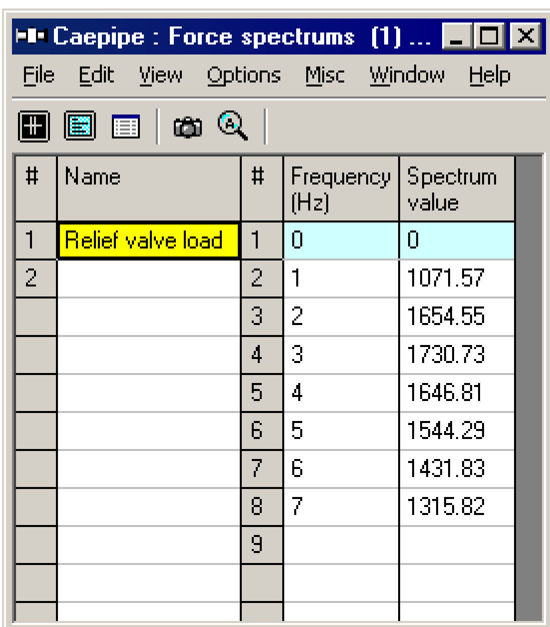
While analyzing the response spectrum case, when a mode (frequency or period) falls outside the spectrum table, CAEPIPE issues a warning and uses the value corresponding to the closest frequency or period in the spectrum table (for example, CAEPIPE uses the value corresponding to the lowest frequency if the calculated frequency is lower than the lowest in the spectrum table or uses the value corresponding to the highest frequency if the calculated frequency is higher than the highest in the spectrum table).

Once you are done inputting the different spectrums using any one of the three methods, you need to input the Spectrum load itself under the menu Loads > Spectrum in the Layout window. Details on inputting the Spectrum Load are provided earlier under the Loads menu.



.Force Spectrums.

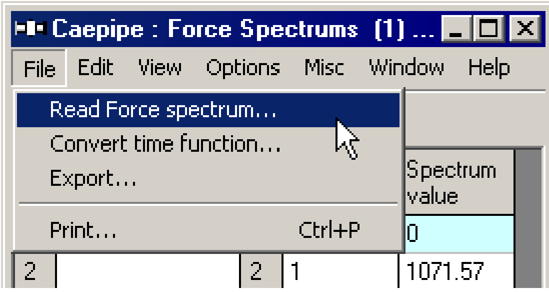
Select "Force Spectrum" through Layout window > Misc. The Force spectrum list appears.



Enter a name for the force spectrum and spectrum values versus frequencies table. In addition to inputting the force spectrum directly, it can also be read from a text file or converted from a previously defined time function.

To read a force spectrum from a text file:

use the List menu: File > Read force spectrum.



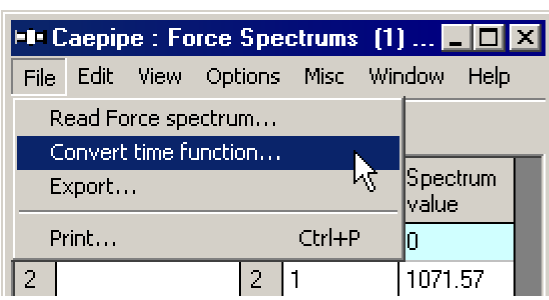
The text file should be in the following format:

Name (up to 31 characters)
Frequency (Hz) Spectrum value
Frequency (Hz) Spectrum value
Frequency (Hz) Spectrum value
. .
. .
. .

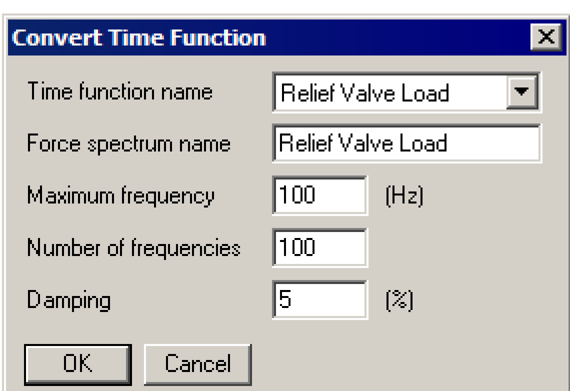
The frequencies can be in any order. They will be sorted in ascending order after reading from the file.

To convert a previously defined time function to force spectrum:

use the List menu: File > Convert time function.



The Convert Time Function dialog appears.



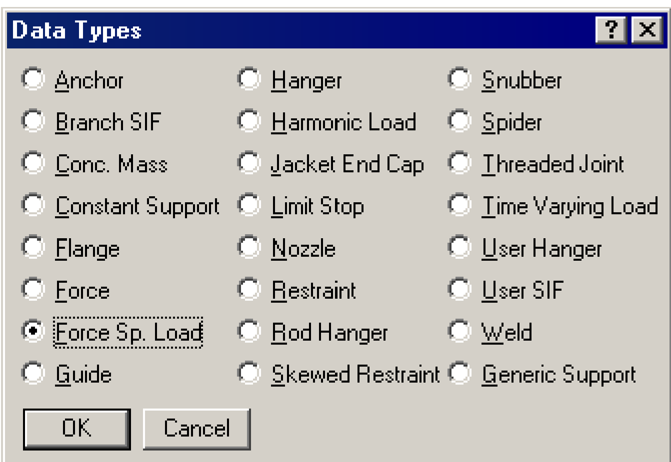
Select the time function to convert from the Time function name drop down combo box. Then input the Force spectrum name (defaults to the Time function name), Maximum frequency, Number of frequencies and the Damping. When you press Enter or click on OK, the time function will be converted to a force spectrum and entered into the force spectrum list.

The time function is converted to a force spectrum by solving the dynamic equation of motion for a damped single spring mass system with the time function as a forcing function at each frequency using Duhamel's integral and dividing the absolute maximum dynamic displacement by the static displacement.

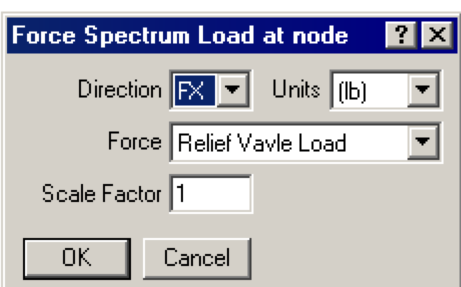
Force Spectrum Load

The force spectrum loads are applied at nodes (in Data column in Layout window). At least one force spectrum must be defined before a force spectrum load at a node can be input.

To apply the force spectrum load at a node click on the Data heading or press Ctrl+Shift+D for Data Types dialog.



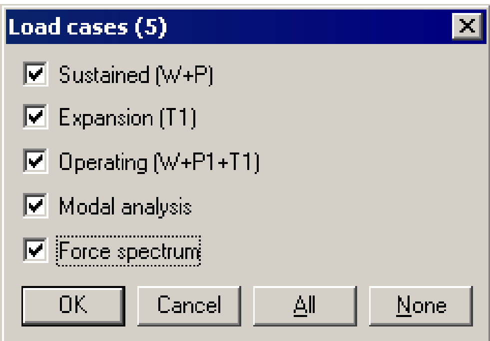
Select "Force Sp. Load" as the data type and click on OK. This opens the Force Spectrum Load dialog.



Select the direction, units and force spectrum using the drop down combo boxes and input appropriate scale factor. The scale factor can be a scalar value, which, when multiplied by the non-dimensional force spectrum, will give the actual magnitudes of the force versus frequency in the global direction and unit selected in the above dialog. Then click on OK to enter the force spectrum load at that node.

7	6" std pipe								
8	30	From							
9	60		6'0"			A53	6	1	Force sp load
10	70	Valve	2'0"			A53	6	1	

Input force spectrum loads at other nodes similarly. Then select the force spectrum load case for analysis using the Layout menu: Loads > Load cases.



Note that Modal analysis and Sustained (W+P) load cases are automatically selected when you select Force spectrum load case. The force spectrum load case is analyzed as an Occasional load.

For more details,seetopic on Force Spectrum in the Technical Reference Manual.

.Time functions.

Time functions are non-dimensionaltables (i.e., a series of time versus value pairs) which describe the variation of the forcing function with time. Usually, a transient fluid flow analysis program computes forces as a function of time at all changes in directions (bends/tees) and other points of interest. These forces/moments result from a transient event such as a fluid hammer. Separate force-time histories are then input as time functions and applied as Time Varying Loads within CAEPIPE at the Data column of thecorresponding nodes of interest in the piping model.

In CAEPIPE, the time function you define can have any interval between two time values. You can make that table as fine as you want it to be.

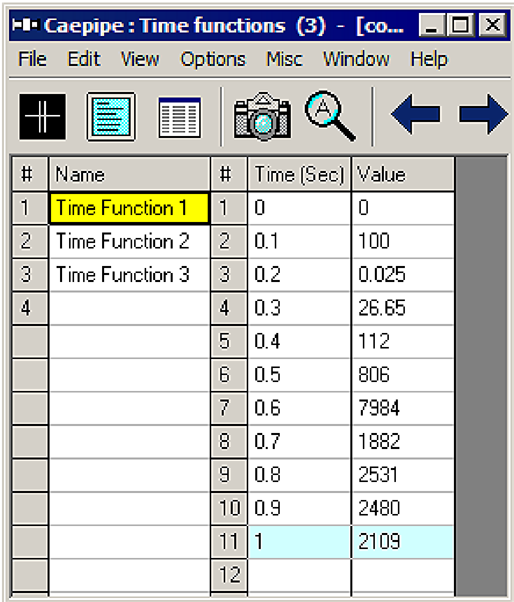
You must have a zero entry for the Value next to the first Time input. Time history analysis begins at time t=0.0. You can input as many time-value pairs as required.

There are two methods for inputting time functions.

1. Input time functions directly into the model.
2. Create a text file for each function and read it into CAEPIPE.

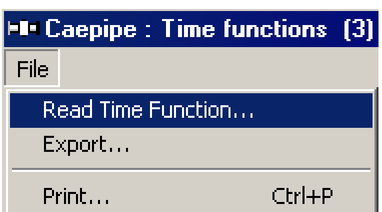
1. Input time functions directly into the model

Click on menu Misc> Time functions to type in time functions. Time is measured in seconds. Value is non-dimensional. You can assign units to these Values when you input a Time varying load at a node. For now, simply start typing the time-value pairs. Next, input parameters for the Time History Analysis Control dialog, and then input Time Varying Loads at nodes of interest.



2. Create a text file for each function and read it into CAEPIPE

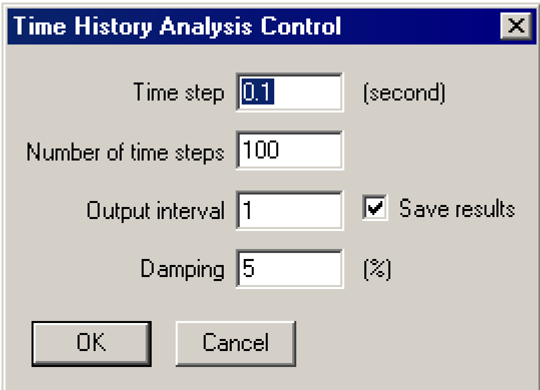
Read these text files into CAEPIPE using the menu File > Read Time Function command. For each function, you need to type a Name (up to 31 characters) on the first line, type time followed by a Value (separated by a space) on the following line. You can input as many time-value pairs (one pair on each line) as required.



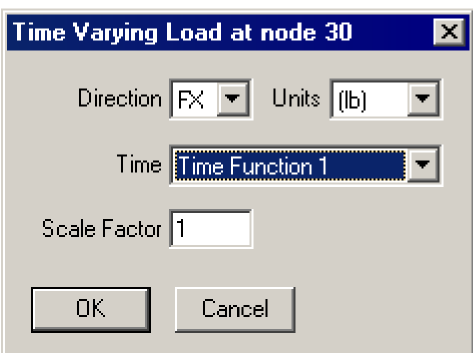
The format of the time function text file is shown below. The time function that is read from a file appears on the row where the yellow highlight is placed (under the Name column). You can use the Edit menu commands to insert an empty row or delete an existing time function. Ensure that no two time functions share the same Name. CAEPIPE issues a warning should such occur.

```
Time Function
0.0  0
0.1  0
0.2  0.025
0.3  26.65
0.4  112
0.5  806
0.6  7984
0.7  1882
0.8  2531
0.9  2480
1    2109
.    .
.    .
```

Once you are done entering time functions, specify the parameters in the Time History Analysis Control dialog under menu Loads > Time History, details of which are provided earlier under the Loads menu topic.



Then, specify Time Varying Loads at applicable nodes in the model. The following figure shows the time function as a time-varying force applied in the X direction (FX) at node 30. Under Units, you can specify one of several depending on whether you are applying the Values in the time function as a force (FX, FY, FZ) or a moment (MX, MY, MZ). For more details, please see discussion under menu Loads > Time History.



.Relief Valve Load Analysis.

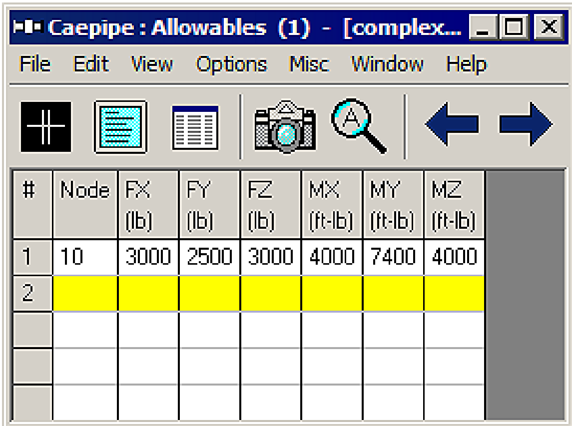
See the topic Relief Valve Load Analysis in the Technical Reference Manual for details.

.Soils.

See Buried Piping Example in the topic on Buried Piping in the Technical Reference Manual.

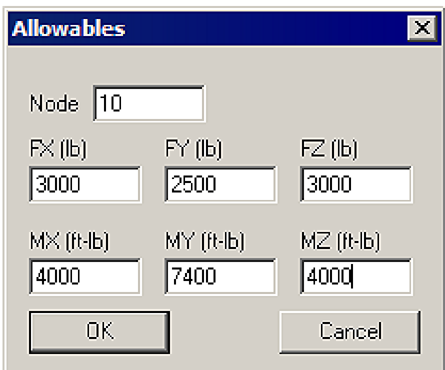
.User-Defined Allowables.

CAEPIPE allows you to define "User Allowables" for Anchors, Generic Supports, Nozzles and Restraints through Misc> User Allowables.

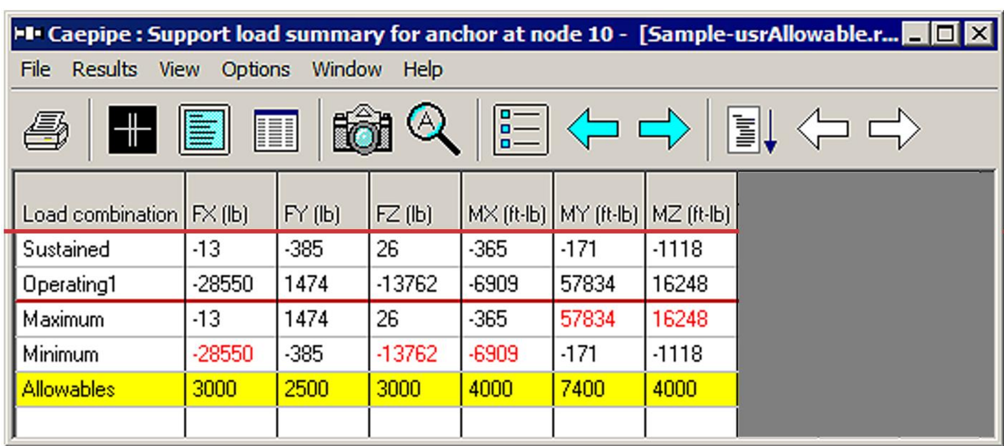


#	Node	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
1	10	3000	2500	3000	4000	7400	4000
2							

The allowables thus defined are compared against calculated loads and printed in Support Load Summary outputs.



If the calculated loads exceed the allowables, they are highlighted in red.



Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
Sustained	-13	-385	26	-365	-171	-1118
Operating1	-28550	1474	-13762	-6909	57834	16248
Maximum	-13	1474	26	-365	57834	16248
Minimum	-28550	-385	-13762	-6909	-171	-1118
Allowables	3000	2500	3000	4000	7400	4000

.Internal Pressure Design: EN 13480-3.

See the topic Pressure Design of Pipe and Pipe Fittings in the Technical Reference Manual for details.

.External Pressure Design: EN 13480-3.

See the topic Pressure Design of Pipe and Pipe Fittings in the Technical Reference Manual for details.

.Wind Load – ASCE/SEI 7-16.

Refer to Section titled "Wind Load" in CAEPIPE Technical Reference Manual for details on the implementation of ASCE/SEI 7-16 in CAEPIPE starting Version 10.30 and manual calculation as per ANSI A58.1.

.Wind Load – EN 1991-1-4 (2010).

Refer to Section titled "Wind Load" in CAEPIPE Technical Reference Manual for details on the implementation of EN 1991-1-4 (2010) in CAEPIPE starting Version 10.30.

Window Menu

For a single model file, CAEPIPE can keep up to four open windows simultaneously for your enhanced understanding of the presented information.

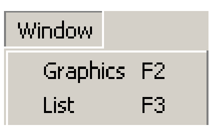
The windows are:

- Layout
- Graphics
- List, and
- Results

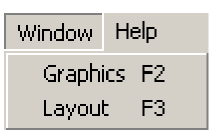
From any window, you can move focus to any other window (such as Graphics or List) using the F2 and F3 hotkeys. F2 will move focus between text (Layout, List or Results) and Graphics windows, and F3 between open text windows (between Results and Layout or Results and List).

Users who have a smaller monitor and work with maximized windows for input, list, graphics and results, will see more the advantage of these hotkeys (F2 and F3). In such a setting, one key press (F2 or F3) will quickly move the focus to another window (without having to either minimize the one where the focus is or switch to another window through the taskbar).

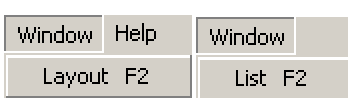
From the Layout window:



From the List window:



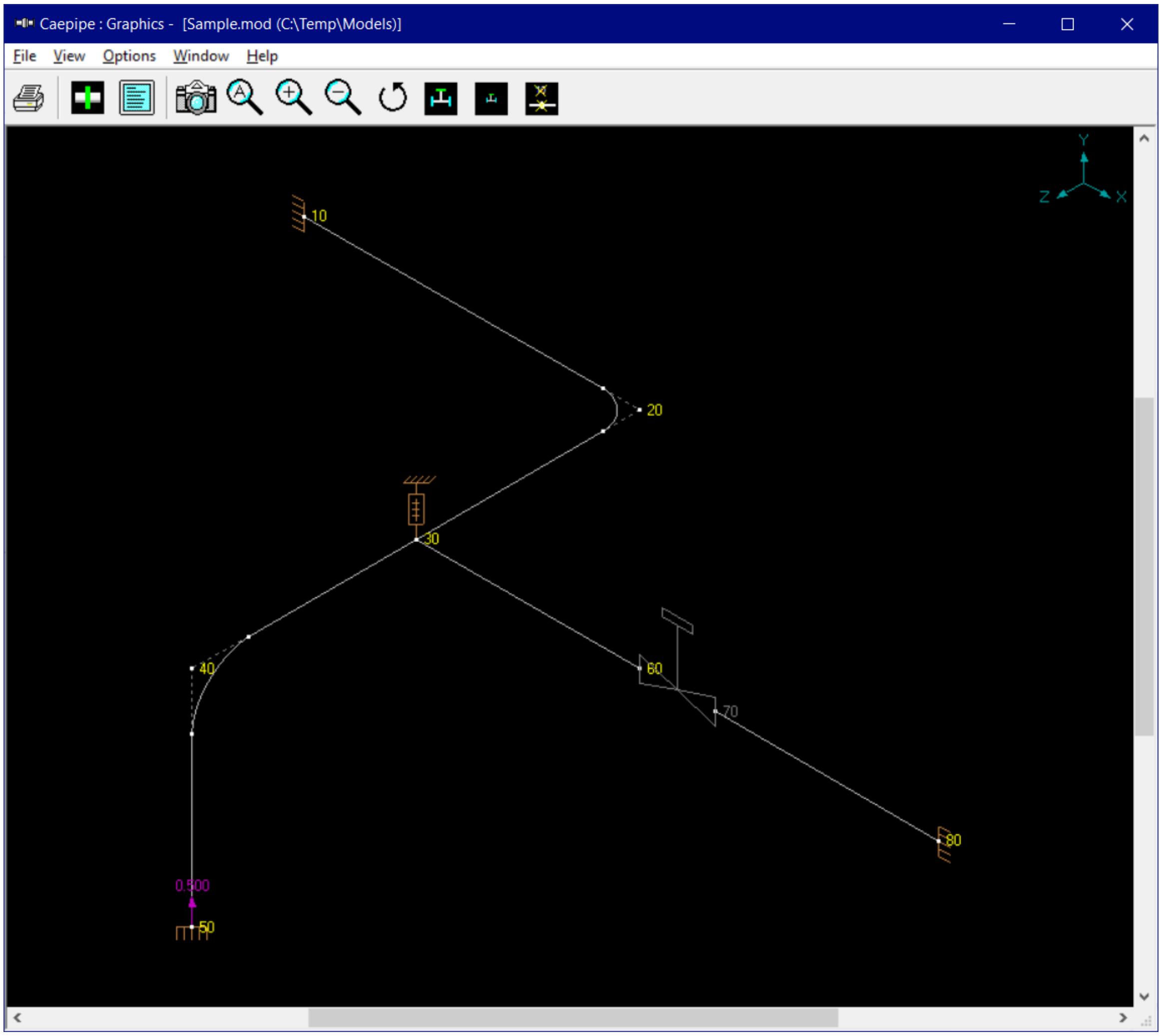
From the Graphics window:



From the Results window:

Note: In the above Window menu from the Results window, if you moved focus to the Layout window (by opening “Layout window” through “Results window > Window > Layout”), then the hotkey F3 would be assigned to Layout (not List as shown). So, the Window menu would look like:

When you open a model, CAEPIPE automatically opens the Layout and the Graphics windows. Many commands to manipulate the image such as zoom, turn (rotate), pan and print are available in this window.

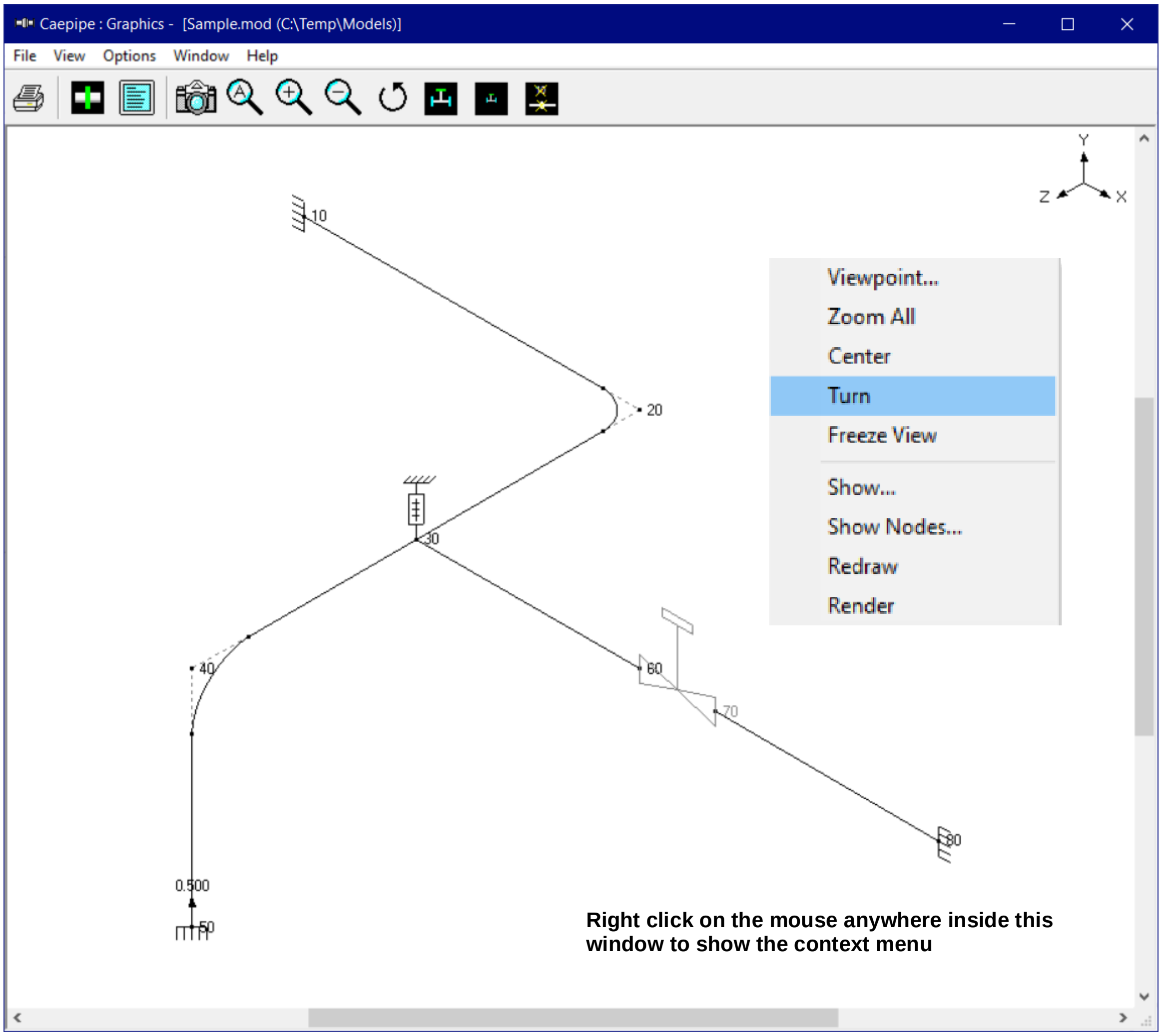


You could opt to display the image with a white background too (use Graphics window >Options > Background > White).

When a model is opened, CAEPIPE displays the last saved model view in the Graphics window.

Context Menu

Right-clicking anywhere inside this window displays the context menu.



In addition to a few commands constantly available on this menu, other commands appear on the menu depending on the results item shown in the Results window. For example, when Sorted stresses are shown, the context menu shows three commands specific to Sorted stresses (Show Stresses, Show stress ratios and [set stress/ratio] Thresholds).

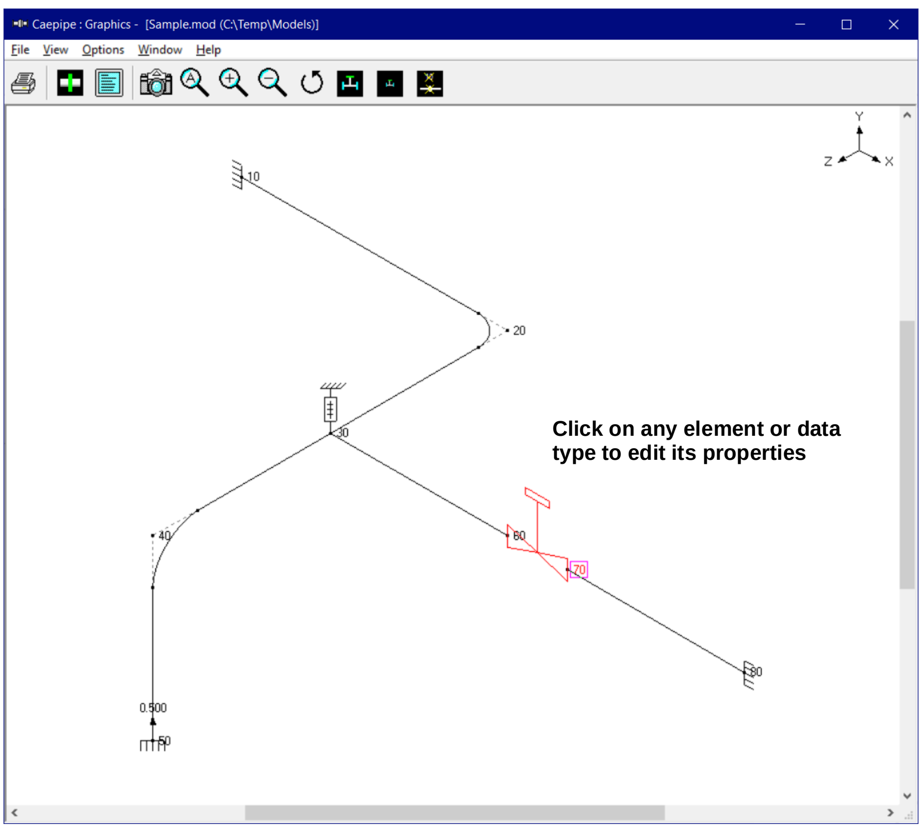
Editing in Graphics Window

Clicking on an editable element or data item brings up the related dialog (same as that you see opened in the Layout window). You can enter or modify properties inside the dialog just as if you were editing in the Layout window. This feature gives you more flexibility during editing your model. See next page for illustrative images.

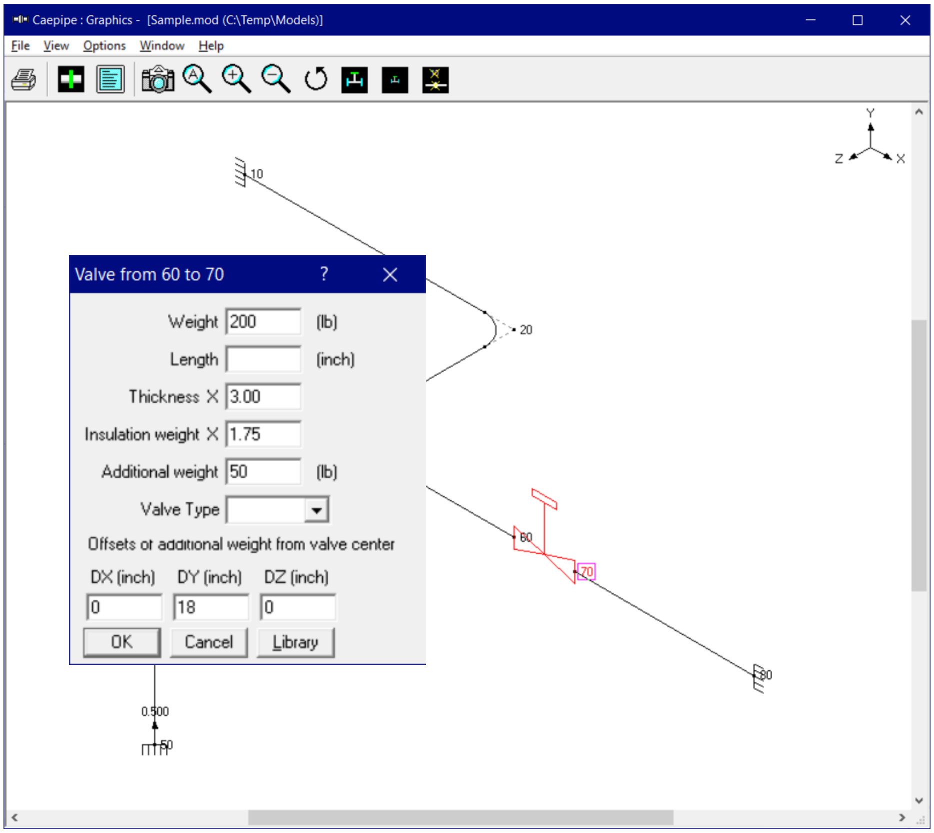
Modified or newly entered data is immediately updated in the Layout and the List windows. When you click on any graphics symbol, CAEPIPE automatically synchronizes the highlights in the Layout and the List windows to the row that contains the element/data item you clicked on, so that you see all the pertinent data about that element at the same time.

This feature works in Render mode too. Sometimes when symbols are closely grouped, you may want to zoom in on that area before clicking on a symbol. That way, CAEPIPE presents you the correct dialog box. Dialog boxes are opened only for elements/data that have editable data. For example, clicking on a pipe element will do nothing but will still synchronize the highlights anyway in the other windows to the same pipe element.

When you are viewing results, you can still click on an element/data item, except now the data you see presented are read-only (you cannot edit them).



Clicking on a valve (as shown above) will open the dialog in the window shown next. You may modify any value inside the dialog and have it updated across all open windows.



Hotkeys You Can Use

In addition to conventional MS-Windows hotkeys (such as Ctrl+P for Print, Ctrl+C for copy image, etc.), you may use the following keys. The usage of these hotkeys are explained in detail in later section of the manual.

Ctrl+A	Zoom All
Ctrl+Shift+C	Center image around a chosen point
Ctrl+T	Turn (rotate) image ON/OFF
Ctrl+S	Show
Ctrl+N	Show node numbers
Ctrl+D	Redraw image
Ctrl+R	Render image

Ctrl+Shift+X	View X
Ctrl+Shift+Y	View Y
Ctrl+Shift+Z	View Z
Ctrl+Shift+I	View Isometric

Mouse Scroll	Zoom In/Out
Ctrl+Mouse Scroll	Pan Up/Down
Shift+Mouse Scroll	Pan Left/Right
Ctrl+Mouse Left Button+Drag	Pan Graphics in Horizontal/ Vertical direction

Ctrl+T followed by	
Ctrl+Mouse Scroll	Rotate in Vertical plane
Shift+Mouse Scroll	Rotate in Horizontal plane
Ctrl+Mouse Left Button+Drag	Rotate in Horizontal/Vertical plane

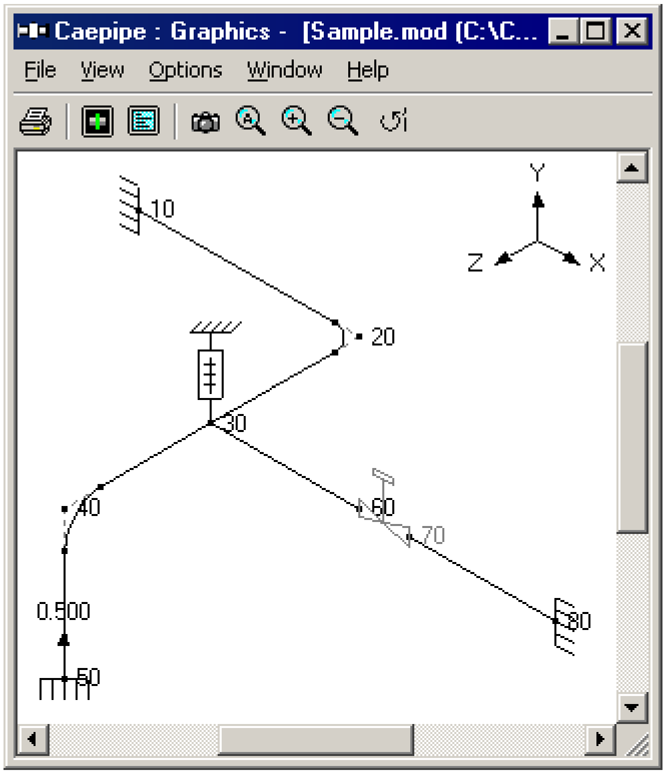
Ctrl+Numpad 8	Increase Symbol size
Ctrl+Numpad 2	Decrease Symbol size

Page Up	Zoom in
Page Down	Zoom out

F2	Move focus to Layout or List window (wherever the focus was before)
F3	Show/Hide Dots + Numbers
F4	Viewpoint
F5	Previous view
F6	Hide Current Element
F7	Make Transparent
F8	Freeze View
F9	Show All
F10	All Transparent
F11	All Opaque
F12	Show/Hide Selected Elements

Dynamic Resizing of Window Image

When you resize the Graphics window, CAEPIPE will automatically resize the image proportionately isotropically. Dynamic scaling makes it unnecessary for you to "Zoom All" every time you resize the window.



Using Scroll Thumbs

When you want to move or pan the image, simply drag the scroll thumbs in the scrollbar and the image moves accordingly. If you have activated the Turn command (menu View > Turn, or Ctrl+T), then the scroll thumbs when moved using the mouse will rotate the image.

Graphics Synchronization with Open Text Windows

The graphics image has a flashing cursor (such as on an anchor or bend) that always synchronizes with the text window that has the focus. In congested models, the flashing cursor points precisely to the location of the element you are working on by acting as a flashing beacon. Example: Move the highlight in Layout to an anchor. The Graphics and the List windows (if open) automatically show or highlight the same anchor.

Dynamic Updating of Data in All Open windows: Layout, Graphics and List

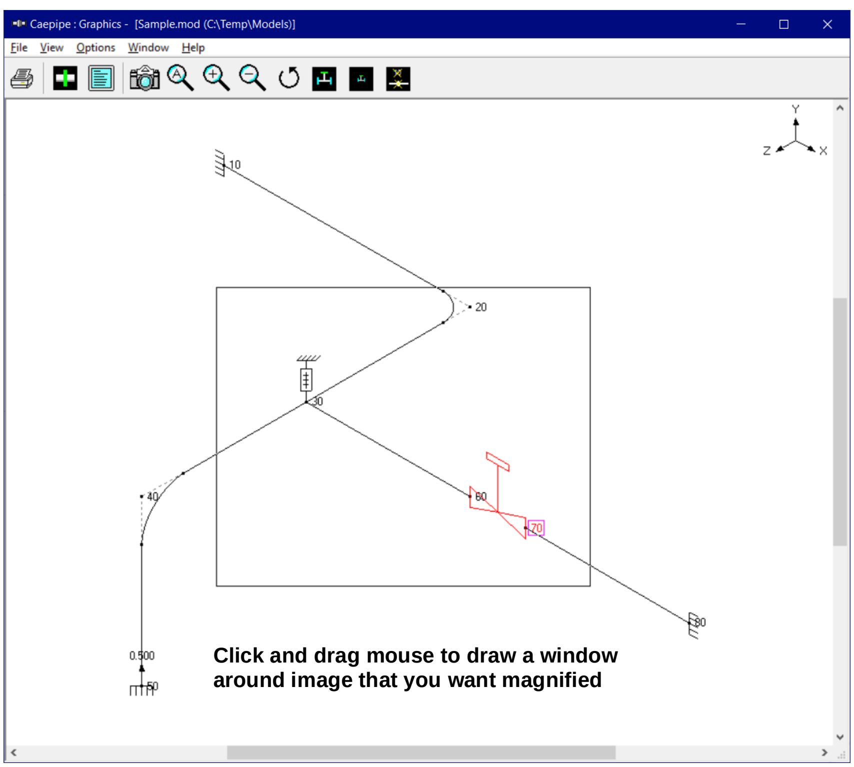
Example: Delete an Anchor or change a Material name in the List window. The Layout and the Graphics windows are automatically updated immediately, showing the changes. You are not forced to perform edits from any one given window. You can get instant feedback.

Simultaneous Visual Updates of Deflected and Mode Shapes

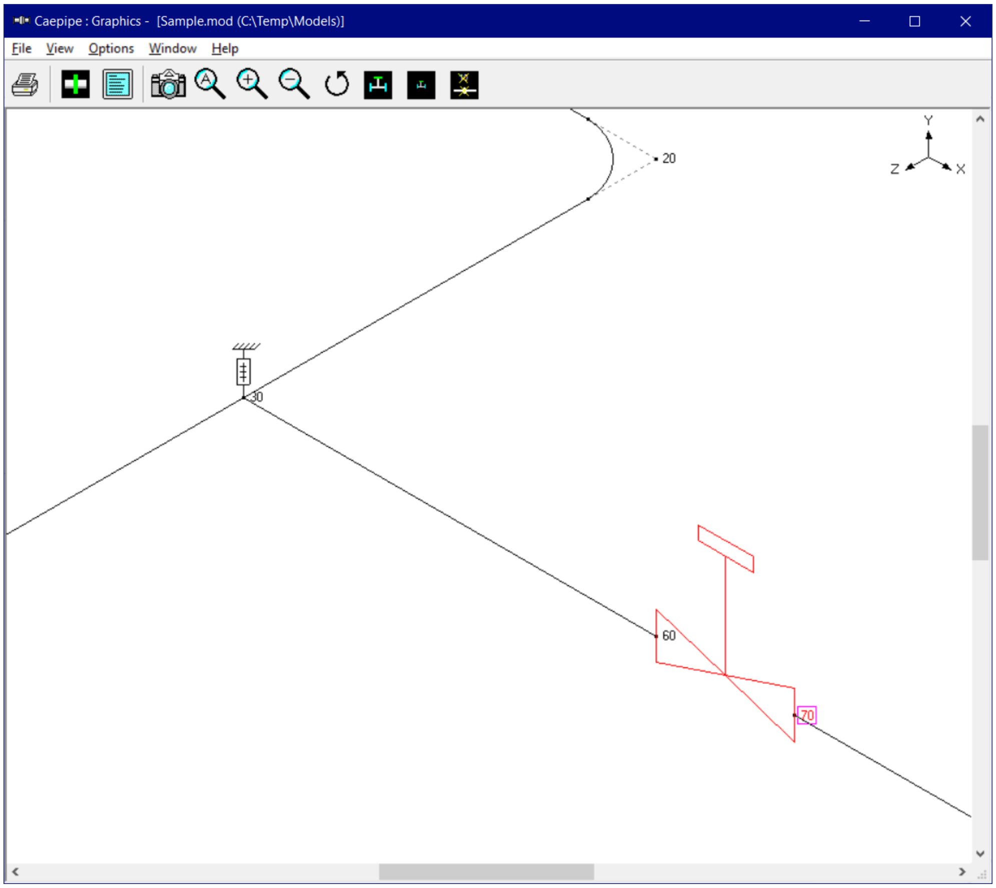
Simply switch between different load cases (or mode shapes) in the Results window to see the respective deflected/mode shape change too in the Graphics window.

Zoom Window

To enlarge an area of image, simply use the mouse to draw a window (a box) around an area you are interested in. CAEPIPE enlarges the image (inside the drawn box) when you release the mouse button.



The zoomed image is shown next.



Menus and Toolbar

The different menus in this window are explained in detail in the Menus section. A summary is given here.

File menu:

- You may print the image to a printer or a file (formats supported are Windows Enhanced Metafile (EMF), Encapsulated Postscript (EPS), HPGL plot file (PLT) and AutoCAD DXF). You can change the printer font from here.

Before printing to a printer or a file, you can change the title for the printout using the Plot Title command. Before printing a rendered image, you may want to select a white background and a quality of rendering (default is High). For DXF & PLT files use the "Drawing Size" command to change size.

View menu:

- Mainly for Graphics operations (Viewpoint, Previous view and Zoom All).

- Selective display of node numbers and graphic symbols makes CAEPIPE flexible and powerful (use Show and Show Nodes commands). In addition, other standard graphical operations such as pan, zoom, previous view, viewpoint change, center image, etc., are available through the menus or a readily available context menu.

- Selective display of elements is possible. Use the commands such as "Make Transparent", "All Opaque", etc.

- Render Mode: Use this mode for realistic visualization of your model using OpenGL®.

Even in this mode, you can selectively display symbols and/or node numbers. Use a combination of Show and Show Nodes commands available in the context menu (or from View menu). Image can be printed, too.

Options menu:

- Customize the display image by choosing a display font, repositioning the Axes symbol, and changing the size of the toolbar icons.

Window menu:

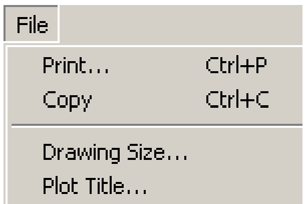
- From here, you can move the focus to the text window from which it came. F2 will move focus between text and Graphics windows. The real advantage of this hotkey (F2) is for users who have a small monitor and work with maximized windows for input, list, graphics and results. In such a setting, one keystroke (F2) will quickly move the focus to another window (without having to either minimize the one where the focus is or switch windows through the taskbar).

Help menu:

For online, Tutorial and About CAEPIPE.

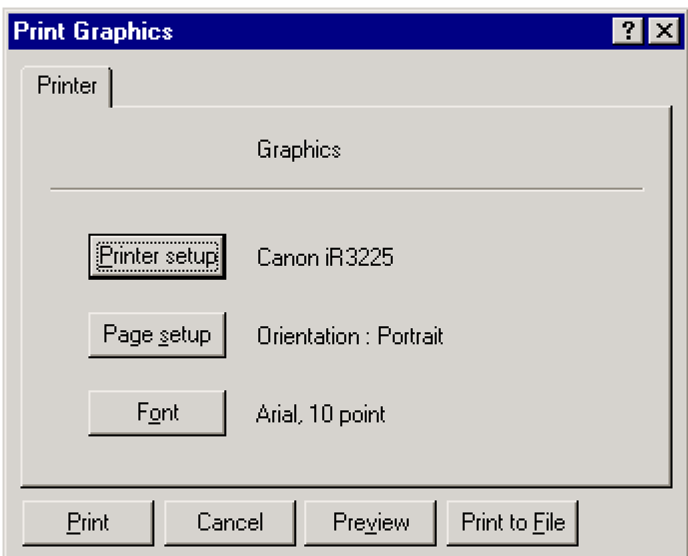
File Menu

This menu has commands for printing and copying graphics images to the clipboard. You can also set the drawing size for a printer/plotter/DXF file and type in a new title for the image (different from the Title entered on row 1 of the Layout window).



.Print (Ctrl+P).

The Print command here opens the Print Graphics dialog which allows you to select a printer and customize other print settings.



Printer

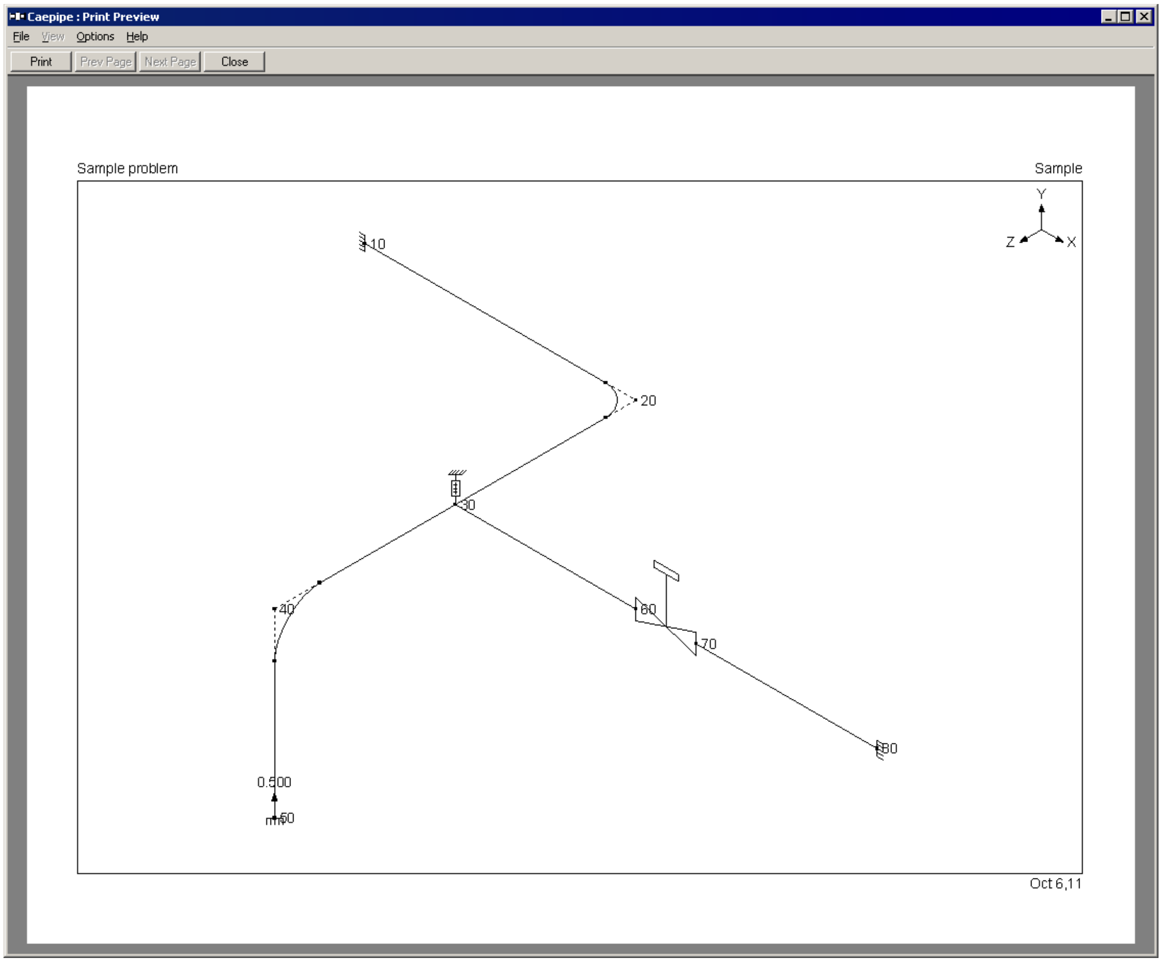
On the Printer tab, you can setup the printer and the graphics print page as well as select the font for printing graphics.

Font

The font you select here is different from the Font selected in the Graphics Font dialog under the Options menu. The Font command here sets the font for printing only, whereasthe Font commandunder the Options menu sets the font to use for Graphics display only as explained earlier in this manual.

Preview

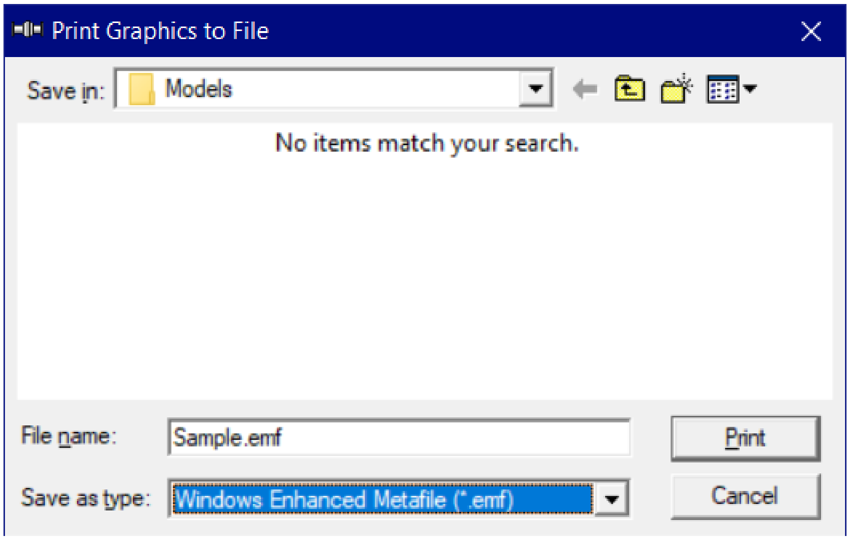
Before you print an image, you can preview it by clicking on this button.



While in the Preview mode, click on Close to return to the Graphics window or click on Print to send the job to the printer.

Print to File

This can be a useful command to convert the line drawing (vector) graphics to other vector formats such as Encapsulated Postscript (EPS), Windows Enhanced Metafile (EMF), AutoCAD's DXF or HPGL's PLT. You can specify a filename for the graphics file. CAEPIPE can also convert other vector graphics such as non-rendered stress and stress ratio plots, deflected and mode shapes to the above vector formats.

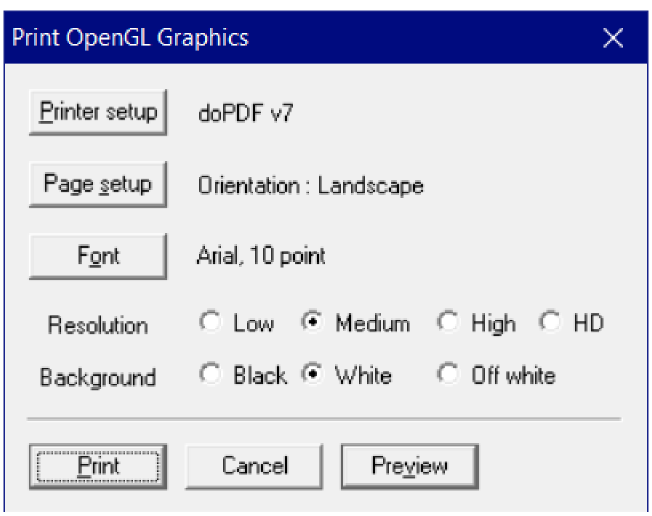


Note on DXF output

CAEPIPE sends only the model information including the line drawing into a DXF file in different layers. It does not send any results (like stresses or deflected shapes) to a DXF file.

Print Dialog for OpenGL Rendered Images

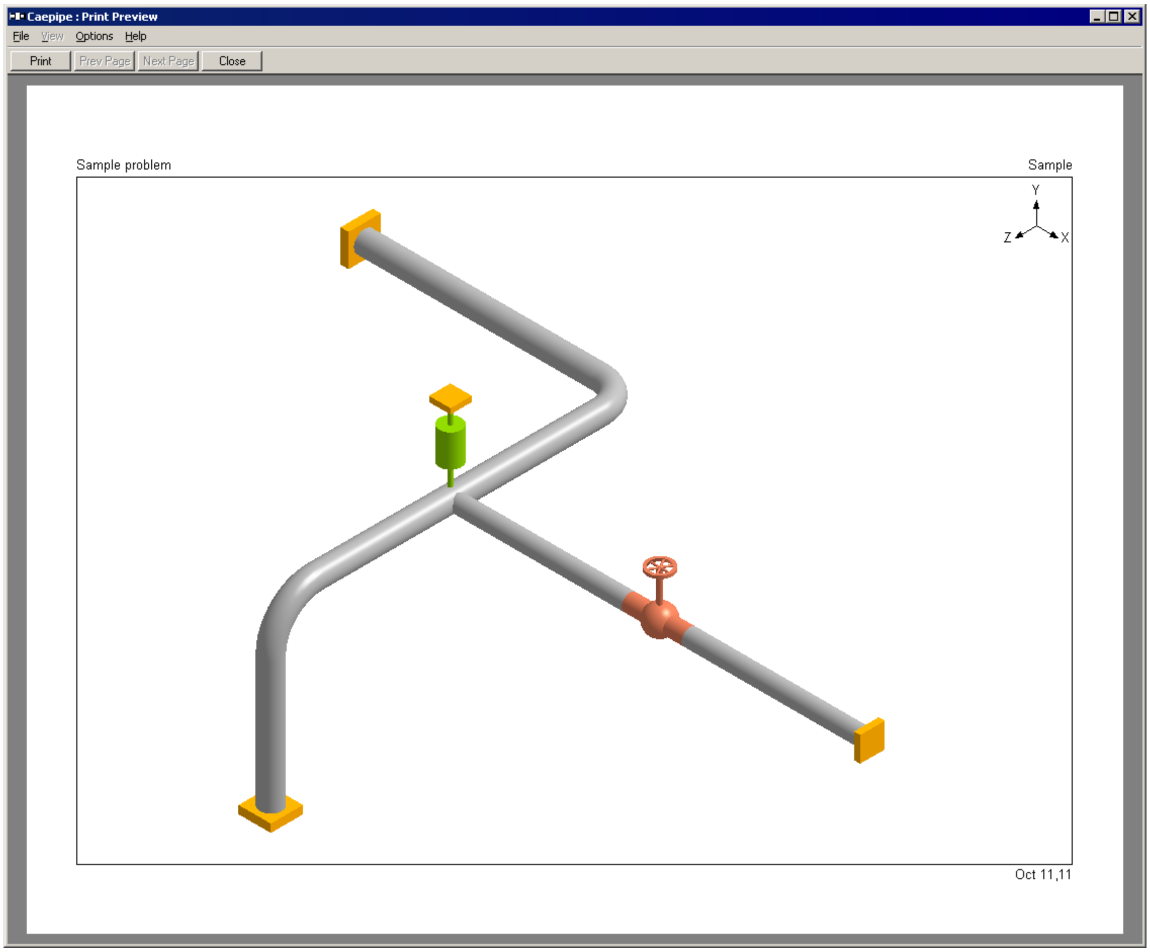
For rendered images, you will see a different Print dialog. Here, as in the earlier Print dialog, you can select a printer and customize certain print settings. The resolution and image background can be set, too. A black background will consume more of your printer ink.



Setting the resolution to High (300 DPI) or HD (1200 DPI) might take up a lot of system memory and slow your system down. Should such happen, use the Low (100 DPI) or Medium (200 DPI) setting.

Preview

Preview the OpenGL graphics rendering by clicking on this button.

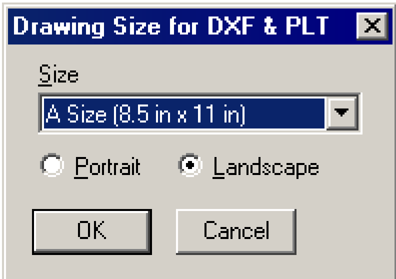


.Copy (Ctrl+C).

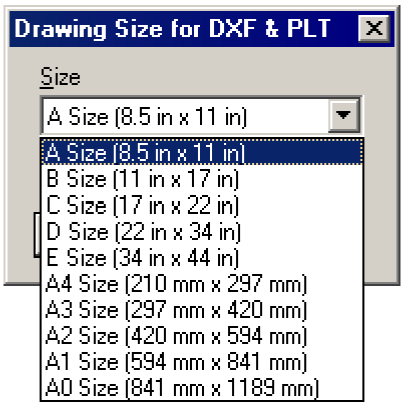
You can copy the displayed graphics image to the clipboard and then paste this image into any other program that accepts it. For example, you can copy the rendered view of a model and paste it into your paint or a graphics processing program so that you can use the finished image (possibly with your annotations) in a word processing program of your choice to generate a report.

.Drawing Size.

The drawing size command applies only to DXF and PLT files. Before you generate a DXF or a PLT from menu File > Print > Print to File, you need to set the drawing size here. The default is set to US A size (Letter – 8.5" x 11").



There are several US and ISO sizes available.

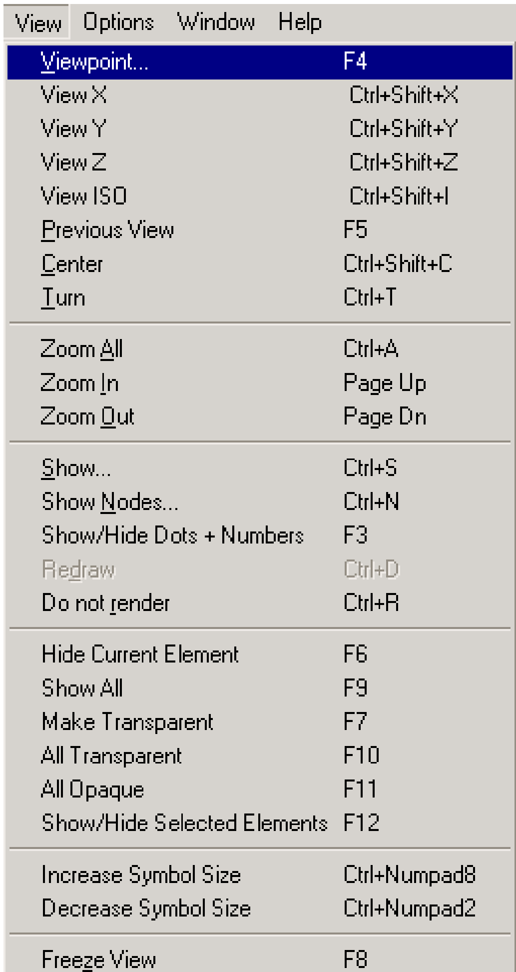


.Plot Title.

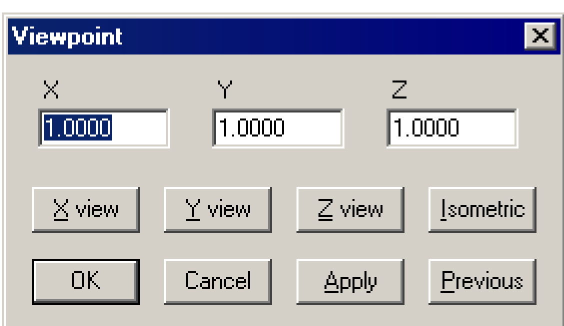
For the graphics image printout (or for the image copied through the Copy command), you may type in an image title different from the model title (in the Layout window). Clicking on Reset will restore the title to the original model Title from row 1 in the Layout window.

View Menu

This menu contains commands for graphics operations.



.Viewpoint (F4).



Use this command to set the graphics viewpoint. Several useful buttons inside the dialog allow you to change viewpoint to a preset one.

For example, if you want to see the "plan" view (Y-vertical), click on "Y view" button.

.View X (Ctrl+Shift+X).

Use this command to view the graphics from X direction.

.View Y (Ctrl+Shift+Y).

Use this command to view the graphics from Y direction.

.View Z (Ctrl+Shift+Z).

Use this command to view the graphics from Z direction.

.View ISO (Ctrl+Shift+I).

Use this command to view the graphics in Isometric mode.

.Previous View (F5).

Use this command to display the previously viewed graphics image in the Graphics window. The last used viewpoint, zoom level and area of the image are brought back into view.

.Center (Ctrl+Shift+C).

Use this command for centering the image around a particular location. When you select this command, the mouse pointer turns into a crosshair. It can be moved around by moving the mouse. Position the crosshair at the image location that you want centered and then left-click on the mouse to center the whole image around that point.

.Turn/Do not Turn (Ctrl+T).

Use this command to Turn ON / OFF image rotation mode.

To rotate the image about the horizontal or the vertical axis,move the thumb on the scroll bars or use the arrow keys on the keyboard.

Or, use

"Shift + Mouse Scroll Button Up and Down" for horizontal rotation of model

"Ctrl + Mouse Scroll Button Up and Down" for vertical rotation of model

Note:

While NOT in rotate mode (Do not turn): use:

"Shift + Mouse Scroll Button Up and Down" for horizontal translation of model

"Ctrl + Mouse Scroll Button Up and Down" for vertical translation of model

.Zoom All (Ctrl+A).

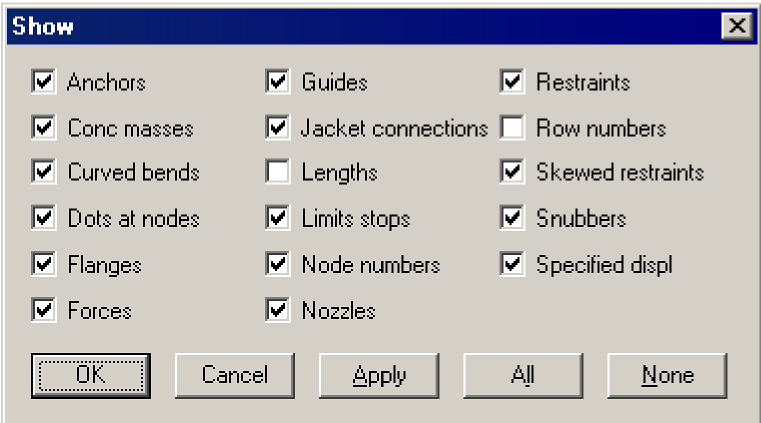
Use this command to view the whole model in the Graphics window.

.Zoom In, Zoom Out.

Use these commands to increase or decrease the magnification level for the image. Hotkeys are Page Up (for Zoom in) and Page Down (for Zoom out). You can also use the mouse wheel button to Zoom in and Zoom out.

.Show.

Use this command to either display or suppress display of various items in the Graphics window. This feature helps in reducing clutter in the displayed image. In addition, it enables you to display selectively one or more items in the Graphics window. Select or deselect items and click on OK or Apply (to see the effect immediately while keeping the dialog open). Click on All or None button to select all or deselect all the items with one click.

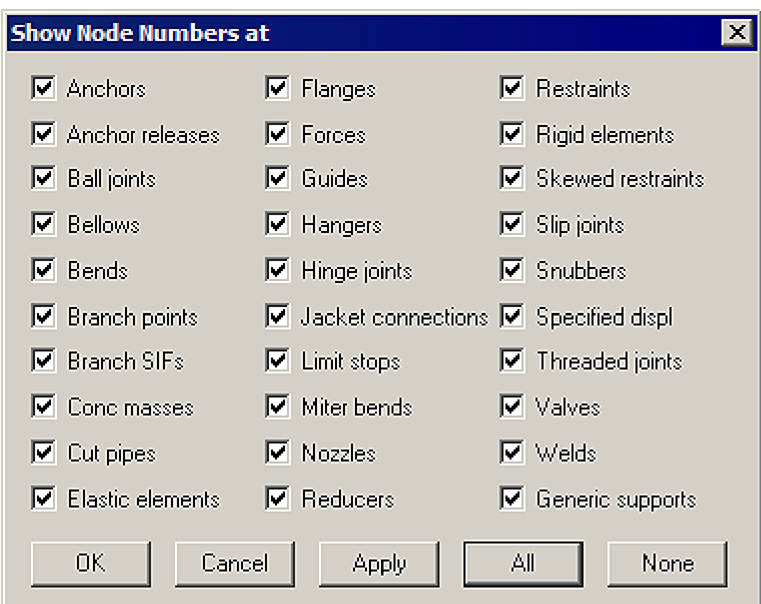


The Local Coordinate system can be shown for all the elements while viewing Results.

.Show/Hide Dots and Numbers (F3).

Use the hotkey F3 to remove Dots and Node numbers from the graphics display.

.Show Nodes.



Use this feature to show node numbers selectively at different locations (such as anchors, limit stops, valves, etc.) to check your input or to display node numbers selectively for one or more items (anchors, hangers, etc.) in the Graphics window. To show node numbers, select or deselect the required items and click on OK or Apply (to see the effect immediately while keeping the dialog open). Click on All or None button to select all or deselect all the items with one click.

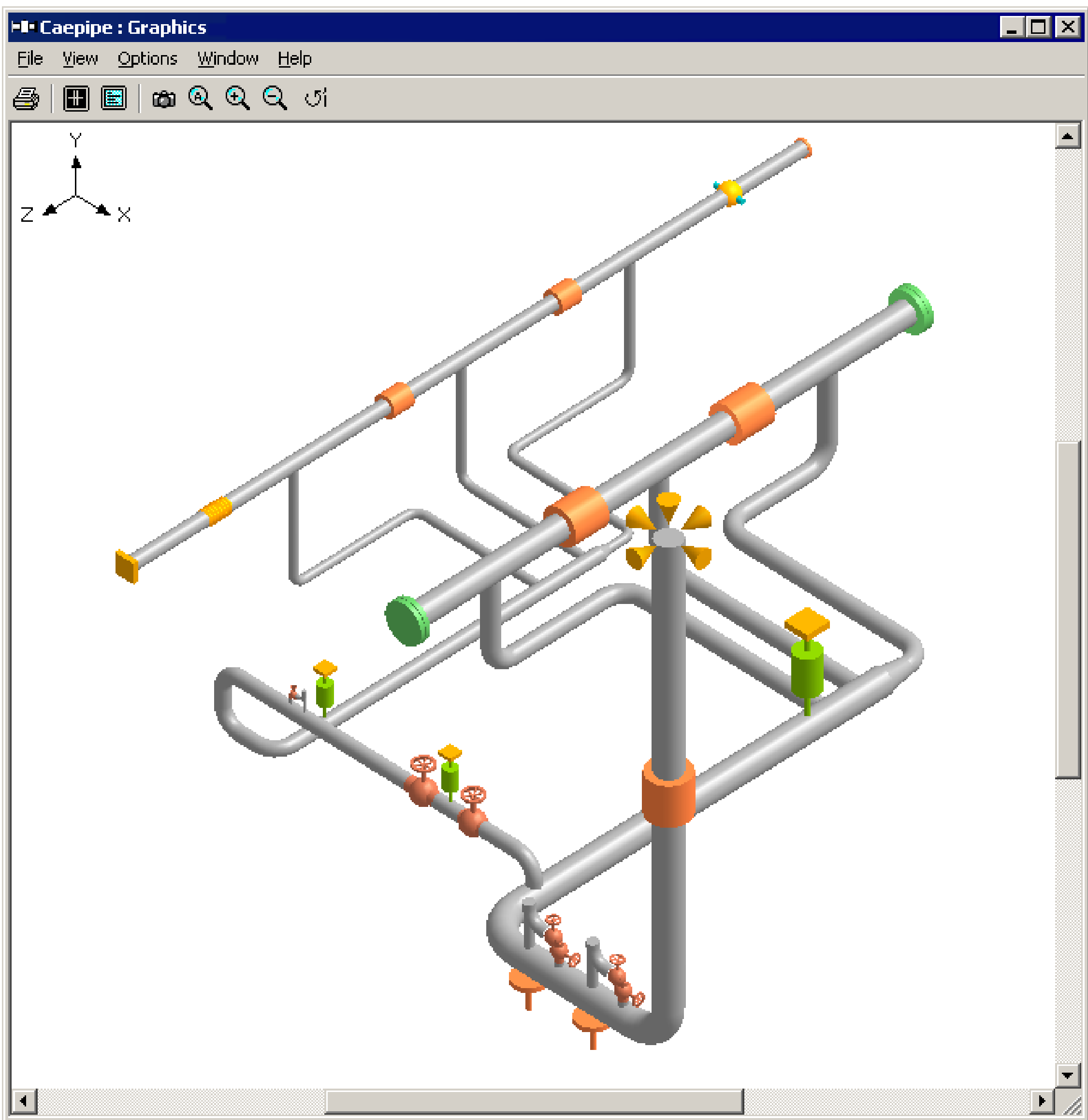
.Redraw (Ctrl+D).

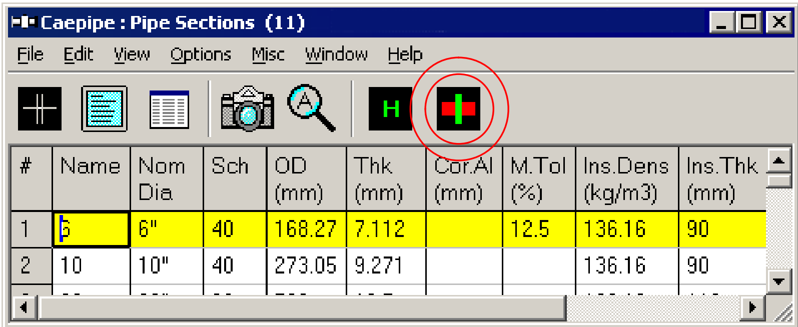
Use this command to redraw the entire graphics image. This will refresh the window with the existing image. This feature is useful when you are working with a two-monitor system.

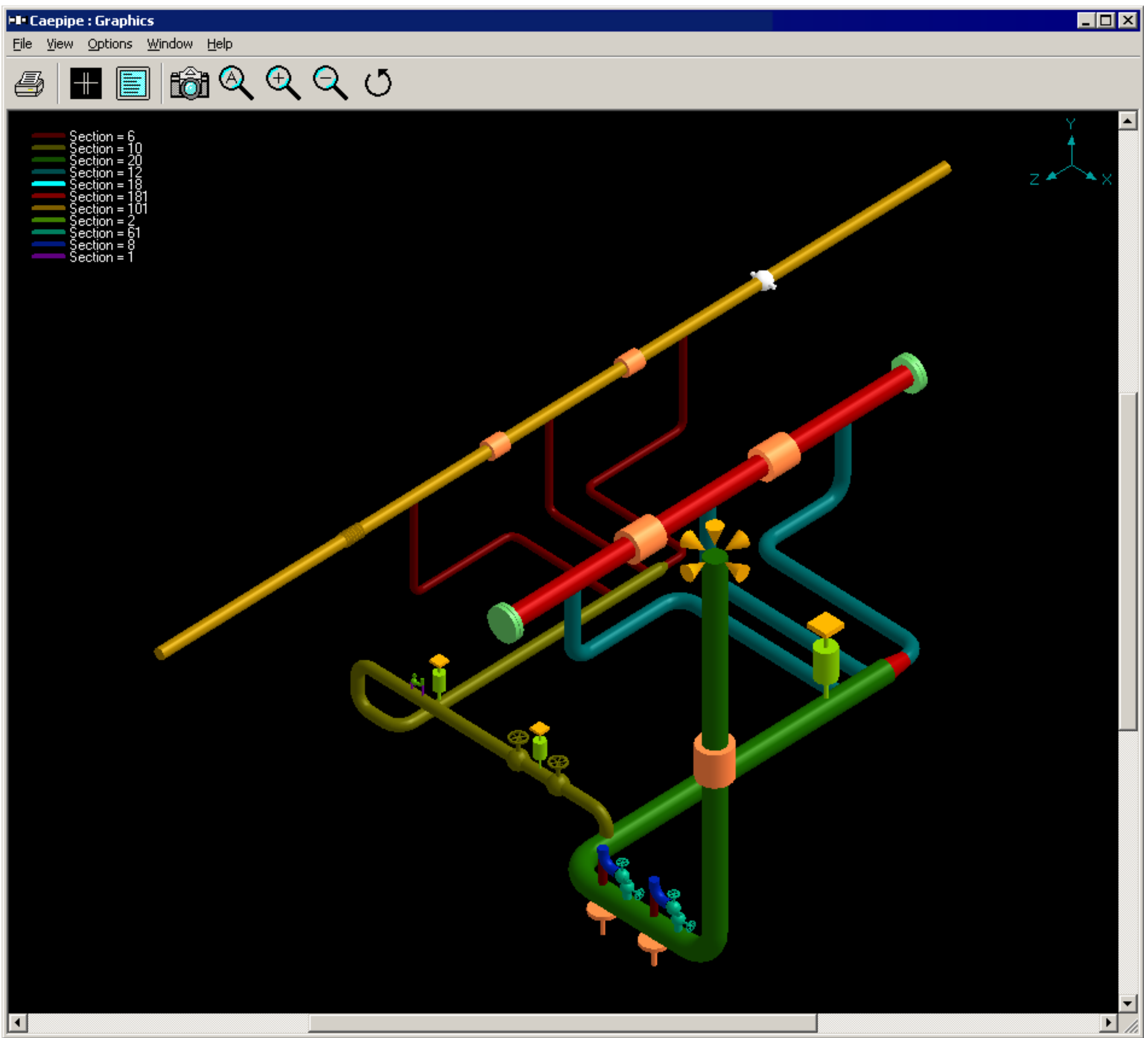
.Render (Ctrl+R).

Use this command to generate a realistic 3-D image of the model. Once you have a rendered image, you can use the Zoom All, Zoom in, Zoom out, Turn and other graphics commands on it. You can use the Show and the Show Nodes commands too to show specific items and/or node numbers. This image can be printed to a (color) printer.

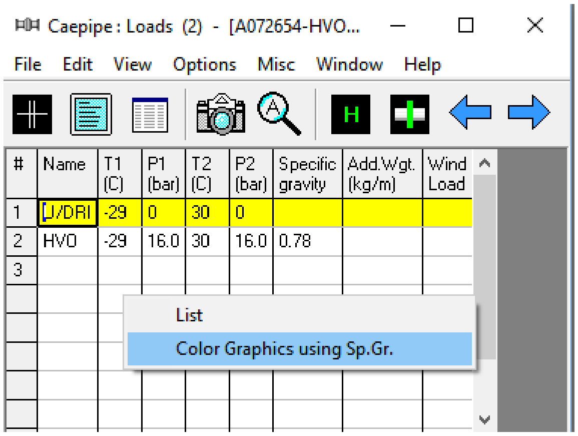
Materials, Sections and Loads are shown graphically using Color Codes in rendered graphics. Press the "Color Graphics" icon available in Materials, Sections and Loads List windows to execute the command.







Users can also plot Color Coded rendered graphics based on Specific Gravity defined. This help in identifying piping systems with different services. This can be performed by selecting the option "Color Graphicsusing Sp. Gr" by mouse right click on "List Loads" window while being in Color Graphics plot.



.Hide Current Element (F6).

With the graphics cursor flashing on the element of interest, press F6 (or select this command) to hide it from view (in rendered mode).

.Show All (F9).

Press F9 (or select this command) to display all elements in the Graphics window including those that were hidden using the "Hide Current Element" command.

.Make Transparent (F7).

Press F7 (or select this command) to make an element transparent so you can see what is behind it; useful in congested models.

.All Transparent (F10).

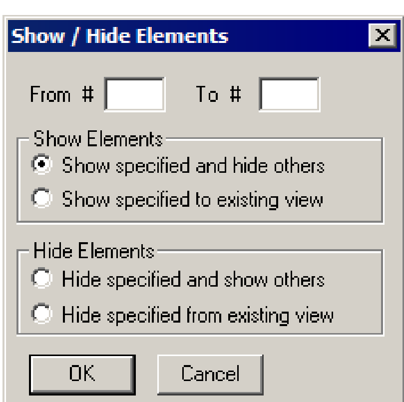
Press F10 (or select this command) to make all elements transparent. F11 will make them opaque.

.All Opaque (F11).

This is the opposite of the previous command in that it makes all transparent elements opaque. F10 will make all elements transparent.

.Show/Hide Selected Elements (F12).

This command works on showing or hiding a group of elements defined in a range. See figure for the different choices.



.Increase Symbol Size (Ctrl+Numpad 8).

This command increases the symbol size in Rendered and Line plots. Alternatively, you can use the Icon to increase the symbol size.

.Decrease Symbol Size (Ctrl+Numpad 2).

This command decreases the symbol size in Rendered and Line plots. Alternatively, you can use the Icon to decrease the symbol size.

.Freeze View (F8).

This key is a toggle. Pressing F8 freezes or unfreezes the display so you do not lose your point of view while modeling.

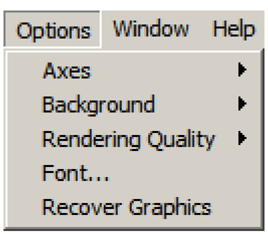
You may still perform the Zoom, Pan, Rotate, etc., with the frozen view. Also, the option "Freeze View" disables the dynamic updating of view (i.e., view remains unchanged) when you scroll in the Layout window.

.Show/Hide Dots and Numbers (F3).

Use the hotkey (F3) to remove the Dots and Node numbers from the Graphics window simultaneously. Pressing F3 (toggle) again will bring them back.

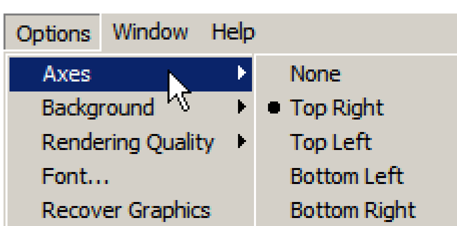
Options Menu

This menu contains commands to change the axes symbol location, color of graphics background, OpenGL rendering quality and the graphics font.



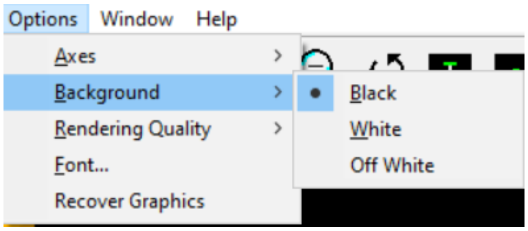
.Axes.

The axes symbol can be displayed in any one of the four corners of the Graphics window or not at all.



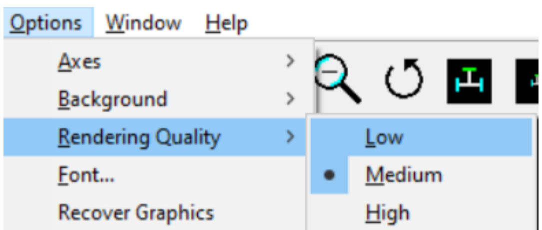
.Background.

The default background color for the Graphics windows is black. You can change background to white or Off white. Changing so will save printer ink when you print a graphics image.



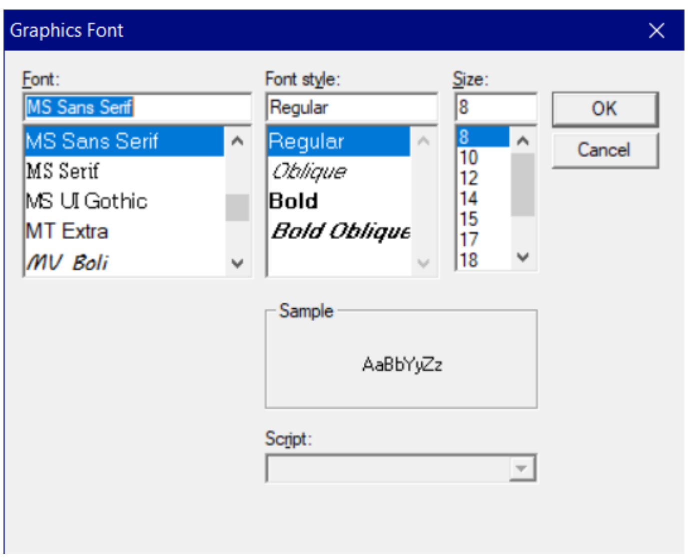
Rendering Quality.

This feature allows you to set the quality of the rendered image. It is best to set this to High unless your computer renders images noticeably slowly.

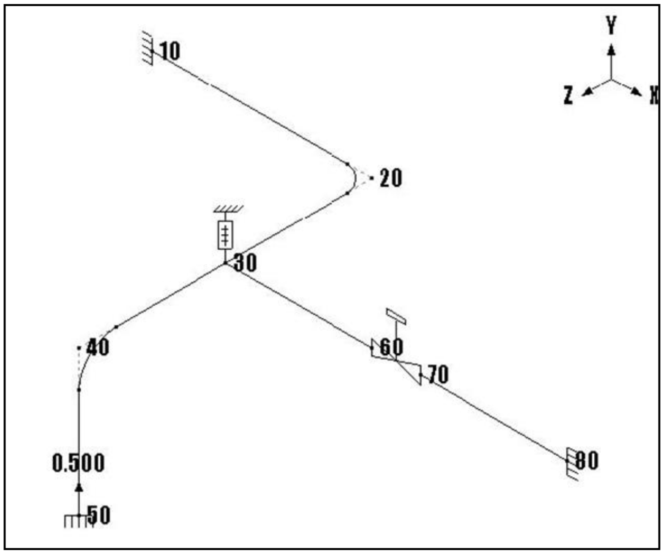


.Font.

You can select any available font for the graphics display here. Note: Font for printing graphics is selected in the Print Graphics dialog as explained earlier under the "Print" command.



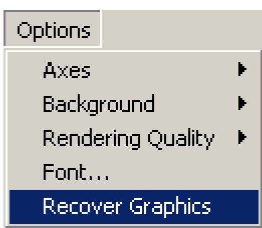
The text in the image below is shown in Impact font.



.Recover Graphics.

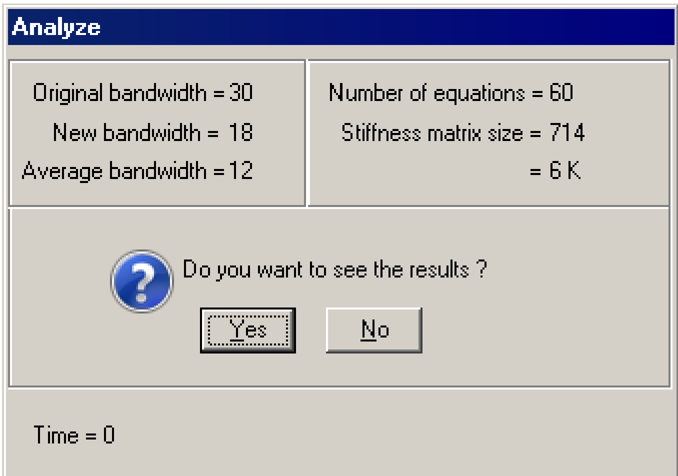
If CAEPIPE abnormally terminates, such may result in garbled graphics. You can recover the graphics image, however, using two methods:

a) Open the model, enter the data to complete the inputs and save. Re-open the model to recover the graphics, OR
b) Open the model, enter the data to complete the inputs and save. Select the command "Recover graphics" through Graphics Window > Options.



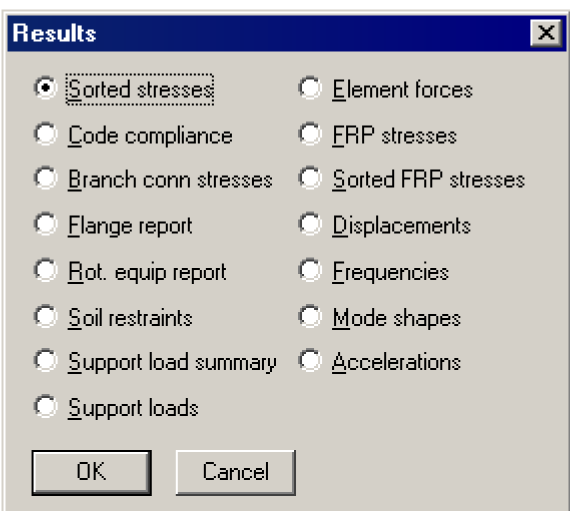
CAEPIPE opens the Results window, one among the four independent windows, either after analysis of a model or after you open a results file (.res) from the File menu in the Main, Layout or Results windows.

On successful completion of model analysis, CAEPIPE asks you whether you want to see results. Click on Yes to continue.



When you do so, the Results window opens, in which you can view every computed result for the piping model. This window displays results in an organized manner conducive to easy understanding of the results. The other windows (Layout and List) may be opened from here in read-only mode for better comprehension, i.e., any detail in these windows may be viewed but cannot be modified. To modify the input data, you must open the Layout window in input mode (by selecting menu File > Input).

A dialog listing the different results for a model is shown below. Select an item to see it.



While viewing results, it is helpful to be aware of the following:

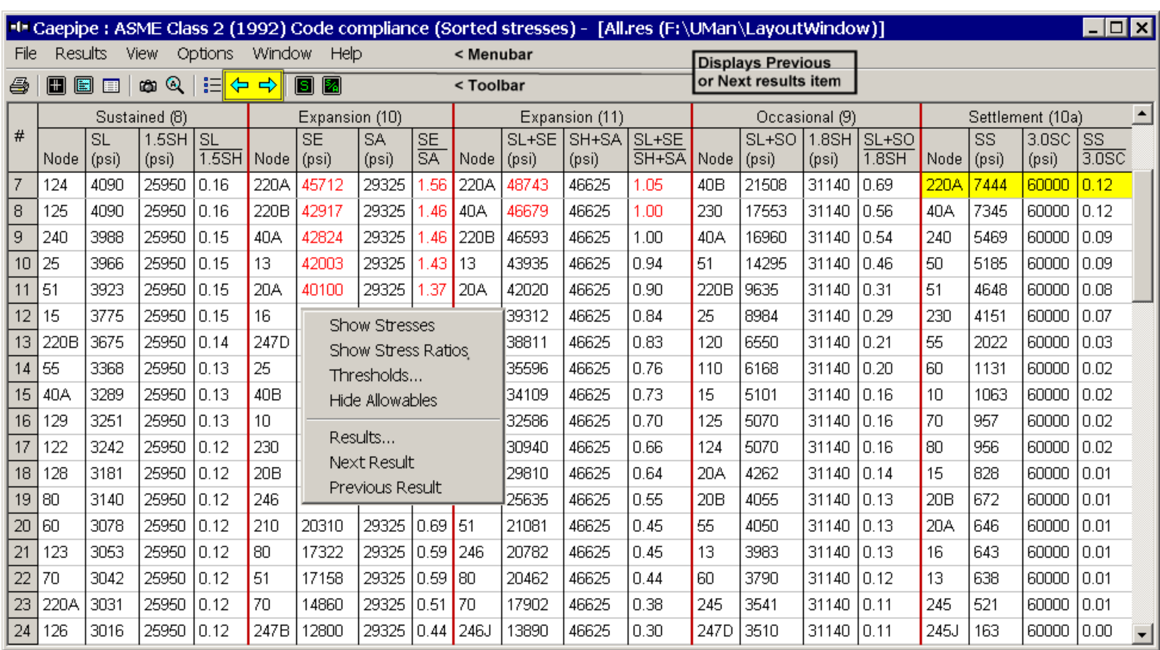
- The name of the results item and the load case (if applicable) are always shown in the title bar of the Results window. See image below.

- Synchronized highlight: When you move the highlight to a node number (or an element) in the Results window, the highlight in the Graphics window is automatically synchronized to the corresponding node number (or element).

- One key navigation: By merely pressing the Tab key, you can move forward through the different results one at a time (Sorted stresses, Code Compliance, Support Loads, Element forces, etc.). Move backward by using Shift+Tab, or by clicking on the cyan colored arrows (left and right). In this window, the display order of results is circular, i.e., if you press Tab from Sorted Stresses, you will see Code compliance next, or if you press Shift+Tab, you will see Time History results (or whichever is the last item on the list of computed results).

- Changing Toolbar and Menus: As you move forward or backward through the results, the toolbar changes with the displayed result. Also, the menu commands in Results and View menus change. They will contain commands relevant to the the displayed result.

- Simultaneous graphics: You can view graphics simultaneously with results (and input).



#	Sustained (8)				Expansion (10)				Expansion (11)				Occasional (9)				Settlement (10a)			
	Node	SL (psi)	1.5SH (psi)	SL / 1.5SH	Node	SE (psi)	SA (psi)	SE / SA	Node	SL+SE (psi)	SH+SA (psi)	SL+SE / SH+SA	Node	SL+SO (psi)	1.8SH (psi)	SL+SO / 1.8SH	Node	SS (psi)	3.0SC (psi)	SS / 3.0SC
7	124	4090	25950	0.16	220A	45712	29325	1.56	220A	48743	46625	1.05	40B	21508	31140	0.69	220A	7444	60000	0.12
8	125	4090	25950	0.16	220B	42917	29325	1.46	40A	46679	46625	1.00	230	17553	31140	0.56	40A	7345	60000	0.12
9	240	3988	25950	0.15	40A	42824	29325	1.46	220B	46593	46625	1.00	40A	16960	31140	0.54	240	5469	60000	0.09
10	25	3966	25950	0.15	13	42003	29325	1.43	13	43935	46625	0.94	51	14295	31140	0.46	50	5185	60000	0.09
11	51	3923	25950	0.15	20A	40100	29325	1.37	20A	42020	46625	0.90	220B	9635	31140	0.31	51	4648	60000	0.08
12	15	3775	25950	0.15	16					39312	46625	0.84	25	8984	31140	0.29	230	4151	60000	0.07
13	220B	3675	25950	0.14	247D					38811	46625	0.83	120	6550	31140	0.21	55	2022	60000	0.03
14	55	3368	25950	0.13	25					35596	46625	0.76	110	6168	31140	0.20	60	1131	60000	0.02
15	40A	3289	25950	0.13	40B					34109	46625	0.73	15	5101	31140	0.16	10	1063	60000	0.02
16	129	3251	25950	0.13	10					32586	46625	0.70	125	5070	31140	0.16	70	957	60000	0.02
17	122	3242	25950	0.12	230					30940	46625	0.66	124	5070	31140	0.16	80	956	60000	0.02
18	128	3181	25950	0.12	20B					29810	46625	0.64	20A	4262	31140	0.14	15	828	60000	0.01
19	80	3140	25950	0.12	246					25635	46625	0.55	20B	4055	31140	0.13	20B	672	60000	0.01
20	60	3078	25950	0.12	210	20310	29325	0.69	51	21081	46625	0.45	55	4050	31140	0.13	20A	646	60000	0.01
21	123	3053	25950	0.12	80	17322	29325	0.59	246	20782	46625	0.45	13	3983	31140	0.13	16	643	60000	0.01
22	70	3042	25950	0.12	51	17158	29325	0.59	80	20462	46625	0.44	60	3790	31140	0.12	13	638	60000	0.01
23	220A	3031	25950	0.12	70	14860	29325	0.51	70	17902	46625	0.38	245	3541	31140	0.11	245	521	60000	0.01
24	126	3016	25950	0.12	247B	12800	29325	0.44	246J	13890	46625	0.30	247D	3510	31140	0.11	245J	163	60000	0.00

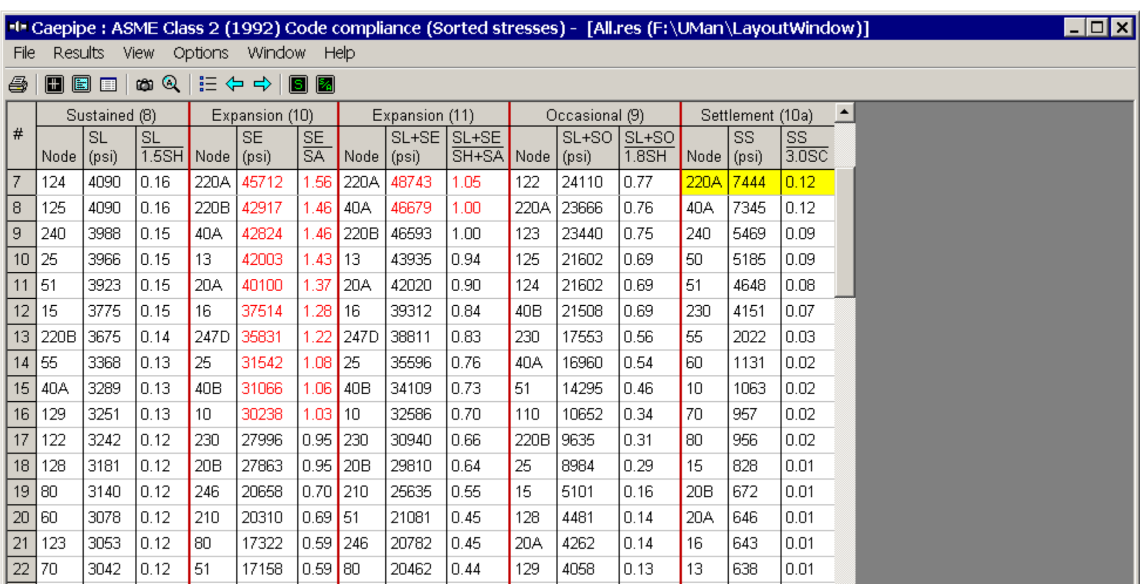
- For a few items, you will notice left and right black arrows. By clicking on these arrow buttons (or pressing the corresponding arrow keys on the keyboard), you can change the load case. Similarly, in some other results, you will notice left and right white arrows. By clicking on these arrow buttons (or pressing the corresponding arrow keys on the keyboard with the CTRL key), you can display results for other related items (for example, while viewing Support loads, you can change over to other supports [like limit stops, hangers, restraints, etc.]).

Depending on the model configuration, you might see more choices within a topic. For example, you might see the Other Forces dialog from Pipe Forces results screen. You might also see the Other Supports dialog from Support Loads results.

Each results item has a context menu as shown in the sorted stresses window (see previous image). Right-click in the results window to see what the context menu offers.

Sorted Stresses

For sorted stresses, you are shown Show stresses, Show stress ratios, Thresholds and Hide Allowables (see image on previous page). The resulting actions of the first three commands apply to the Graphics window and the last one applies to the Results window. For example, in the previous image CAEPIPE will remove five columns (1.5SH, SA, SH+SA, 1.8SH and 3.0SC) when you select "Hide Allowables" thereby allowing you to reduce the information presented. See next figure (in which "Allowables" are hidden).



#	Sustained (8)			Expansion (10)			Expansion (11)			Occasional (9)			Settlement (10a)		
	Node	SL (psi)	SL / 1.5SH	Node	SE (psi)	SE / SA	Node	SL+SE (psi)	SL+SE / SH+SA	Node	SL+SO (psi)	SL+SO / 1.8SH	Node	SS (psi)	SS / 3.0SC
7	124	4090	0.16	220A	45712	1.56	220A	48743	1.05	122	24110	0.77	220A	7444	0.12
8	125	4090	0.16	220B	42917	1.46	40A	46679	1.00	220A	23666	0.76	40A	7345	0.12
9	240	3988	0.15	40A	42824	1.46	220B	46593	1.00	123	23440	0.75	240	5469	0.09
10	25	3966	0.15	13	42003	1.43	13	43935	0.94	125	21602	0.69	50	5185	0.09
11	51	3923	0.15	20A	40100	1.37	20A	42020	0.90	124	21602	0.69	51	4648	0.08
12	15	3775	0.15	16	37514	1.28	16	39312	0.84	40B	21508	0.69	230	4151	0.07
13	220B	3675	0.14	247D	35831	1.22	247D	38811	0.83	230	17553	0.56	55	2022	0.03
14	55	3368	0.13	25	31542	1.08	25	35596	0.76	40A	16960	0.54	60	1131	0.02
15	40A	3289	0.13	40B	31066	1.06	40B	34109	0.73	51	14295	0.46	10	1063	0.02
16	129	3251	0.13	10	30238	1.03	10	32586	0.70	110	10652	0.34	70	957	0.02
17	122	3242	0.12	230	27996	0.95	230	30940	0.66	220B	9635	0.31	80	956	0.02
18	128	3181	0.12	20B	27863	0.95	20B	29810	0.64	25	8984	0.29	15	828	0.01
19	80	3140	0.12	246	20658	0.70	210	25635	0.55	15	5101	0.16	20B	672	0.01
20	60	3078	0.12	210	20310	0.69	51	21081	0.45	128	4481	0.14	20A	646	0.01
21	123	3053	0.12	80	17322	0.59	246	20782	0.45	20A	4262	0.14	16	643	0.01
22	70	3042	0.12	51	17158	0.59	80	20462	0.44	129	4058	0.13	13	638	0.01

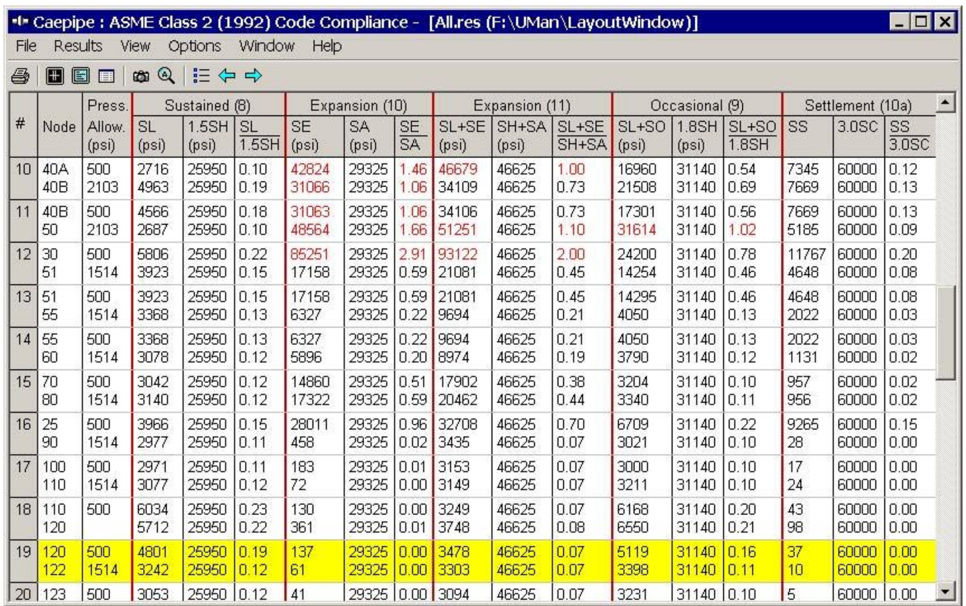
A few screenshots of Results window with the different results items are shown next.

Code Compliance

Here, CAEPIPE displays the stresses on an element-by-element basis. Sorted stresses screen (i.e., the previous results item) shows stresses at nodes sorted in the descending order of stress ratios. In this Code Compliance results item, CAEPIPE displays stresses for each element (highlight is on element 19, nodes 120 and 122).

From the sample "Code Compliance" results of CAEPIPE shown below, you will observe that the 2nd Column titled "Press. Allow" output the following for each element.

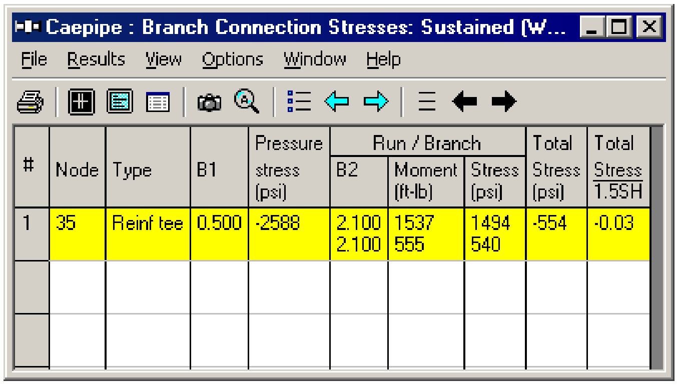
1. First row outputs the "Design Pressure" input for that element.
2. Second row outputs the "Calculated Allowable Pressure" for that element as per the equation provided in the corresponding piping code selected for analysis. Please note, when the "Design Pressure" input for an element exceedsthe "Allowable Pressure"computed for that element, then CAEPIPE will change the display color of Design Pressure to RED.



#	Node	Press. Allow. (psi)	Sustained (8) SL (psi)	1.5SH (psi)	SL / 1.5SH	Expansion (10) SE (psi)	SA (psi)	SE / SA	Expansion (11) SL+SE (psi)	SH+SA (psi)	SL+SE / SH+SA	Occasional (9) SL+SO (psi)	1.8SH (psi)	SL+SO / 1.8SH	Settlement (10a) SS	3.0SC	SS / 3.0SC
10	40A 40B	500 2103	2716 4963	25950 25950	0.10 0.19	42824 31066	29325 29325	1.46 1.06	46679 34109	46625 46625	1.00 0.73	16960 21508	31140 31140	0.54 0.69	7345 7669	60000 60000	0.12 0.13
11	40B 50	500 2103	4566 2687	25950 25950	0.18 0.10	31063 48564	29325 29325	1.06 1.66	34106 51251	46625 46625	0.73 1.10	17301 31614	31140 31140	0.56 1.02	7669 5185	60000 60000	0.13 0.09
12	30 51	500 1514	5806 3923	25950 25950	0.22 0.15	85251 17158	29325 29325	2.91 0.59	93122 21081	46625 46625	2.00 0.45	24200 14254	31140 31140	0.78 0.46	11767 4648	60000 60000	0.20 0.08
13	51 55	500 1514	3923 3368	25950 25950	0.15 0.13	17158 6327	29325 29325	0.59 0.22	21081 9694	46625 46625	0.45 0.21	14295 4050	31140 31140	0.46 0.13	4648 2022	60000 60000	0.08 0.03
14	55 60	500 1514	3368 3078	25950 25950	0.13 0.12	6327 5896	29325 29325	0.22 0.20	9694 8974	46625 46625	0.21 0.19	4050 3790	31140 31140	0.13 0.12	2022 1131	60000 60000	0.03 0.02
15	70 80	500 1514	3042 3140	25950 25950	0.12 0.12	14860 17322	29325 29325	0.51 0.59	17902 20462	46625 46625	0.38 0.44	3204 3340	31140 31140	0.10 0.11	957 956	60000 60000	0.02 0.02
16	25 90	500 1514	3966 2977	25950 25950	0.15 0.11	28011 458	29325 29325	0.96 0.02	32708 3435	46625 46625	0.70 0.07	6709 3021	31140 31140	0.22 0.10	9265 28	60000 60000	0.15 0.00
17	100 110	500 1514	2971 3077	25950 25950	0.11 0.12	183 72	29325 29325	0.01 0.00	3153 3149	46625 46625	0.07 0.07	3000 3211	31140 31140	0.10 0.10	17 24	60000 60000	0.00 0.00
18	110 120	500	6034 5712	25950 25950	0.23 0.22	130 361	29325 29325	0.00 0.01	3249 3748	46625 46625	0.07 0.08	6168 6550	31140 31140	0.20 0.21	43 98	60000 60000	0.00 0.00
19	120 122	500 1514	4801 3242	25950 25950	0.19 0.12	137 61	29325 29325	0.00 0.00	3478 3303	46625 46625	0.07 0.07	5119 3398	31140 31140	0.16 0.11	37 10	60000 60000	0.00 0.00
20	123	500	3053	25950	0.12	41	29325	0.00	3094	46625	0.07	3231	31140	0.10	5	60000	0.00

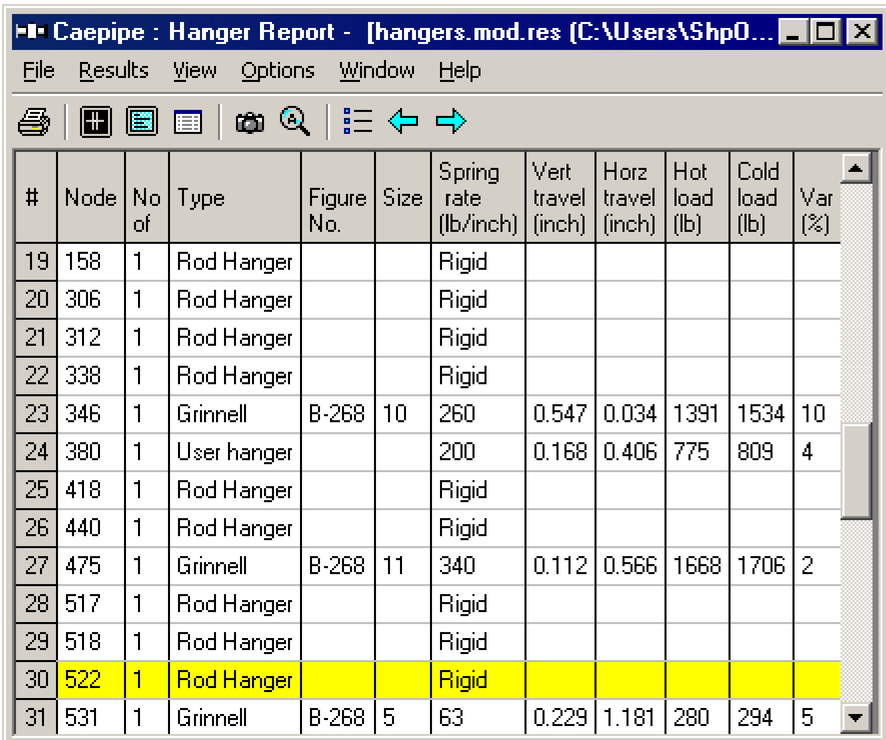
Branch Connection Stresses

Branch connection stresses are available only for ASME Section III codes.You can change the load case by pressing the left or right black arrow key.



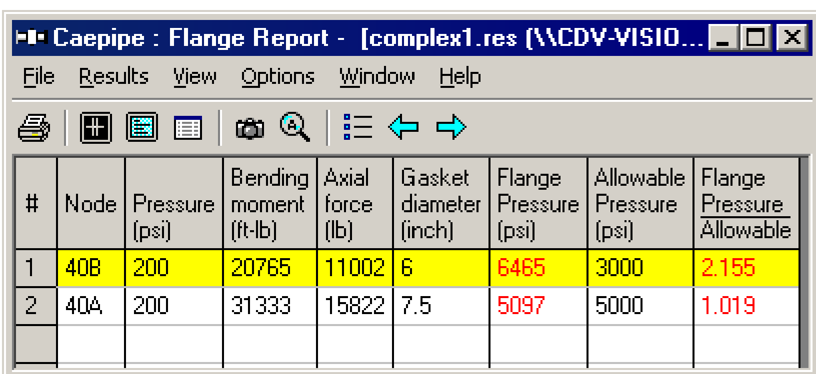
#	Node	Type	B1	Pressure stress (psi)	Run / Branch B2	Moment (ft-lb)	Stress (psi)	Total Stress (psi)	Total Stress / 1.5SH
1	35	Reinf tee	0.500	-2588	2.100 2.100	1537 555	1494 540	-554	-0.03

Hanger Report



#	Node	No of	Type	Figure No.	Size	Spring rate (lb/inch)	Vert travel (inch)	Horz travel (inch)	Hot load (lb)	Cold load (lb)	Var (%)
19	158	1	Rod Hanger			Rigid					
20	306	1	Rod Hanger			Rigid					
21	312	1	Rod Hanger			Rigid					
22	338	1	Rod Hanger			Rigid					
23	346	1	Grinnell	B-268	10	260	0.547	0.034	1391	1534	10
24	380	1	User hanger			200	0.168	0.406	775	809	4
25	418	1	Rod Hanger			Rigid					
26	440	1	Rod Hanger			Rigid					
27	475	1	Grinnell	B-268	11	340	0.112	0.566	1668	1706	2
28	517	1	Rod Hanger			Rigid					
29	518	1	Rod Hanger			Rigid					
30	522	1	Rod Hanger			Rigid					
31	531	1	Grinnell	B-268	5	63	0.229	1.181	280	294	5

Flange Report



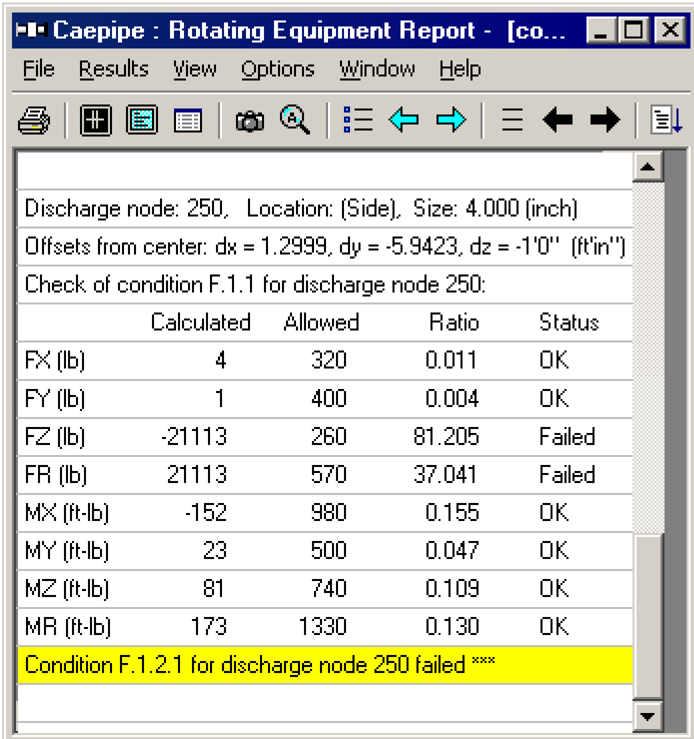
#	Node	Pressure (psi)	Bending moment (ft-lb)	Axial force (lb)	Gasket diameter (inch)	Flange Pressure (psi)	Allowable Pressure (psi)	Flange Pressure / Allowable
1	40B	200	20765	11002	6	6465	3000	2.155
2	40A	200	31333	15822	7.5	5097	5000	1.019

See the end of the "Flange" section in the Technical Reference Manualfor suggestions on how to reduce the ratio in the last column.

Rotating Equipment Reports

The rotating equipment reports are produced for all Operating load cases. The example model shown below has several cold spring operating load cases and hence you see the left and right black arrow keys.

The left and right white arrow keys advance (or go back) to the next (or previous) rotating equipment report.



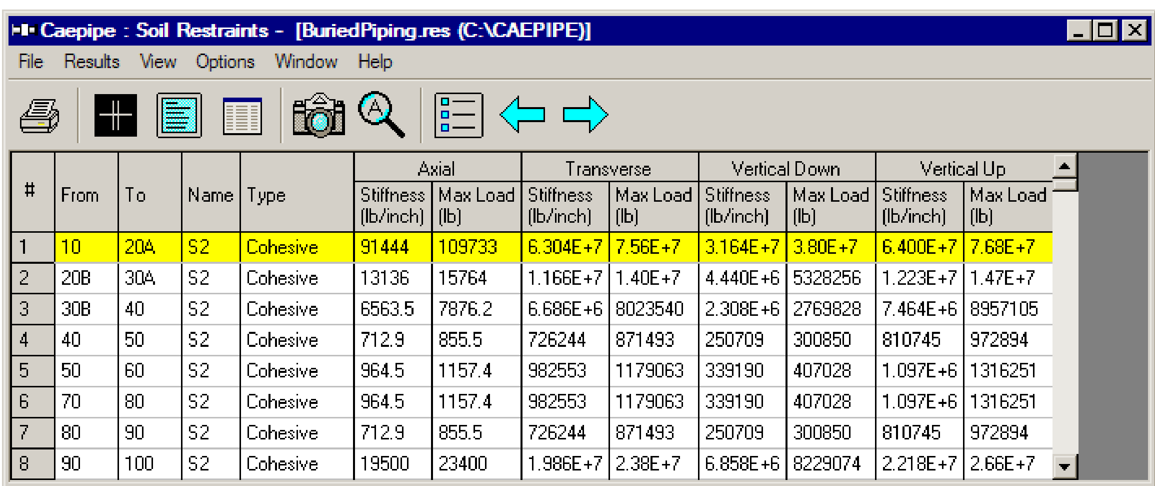
Discharge node: 250, Location: (Side), Size: 4.000 (inch)
Offsets from center: dx = 1.2999, dy = -5.9423, dz = -1'0" (ft'in")
Check of condition F.1.1 for discharge node 250:

	Calculated	Allowed	Ratio	Status
FX (lb)	4	320	0.011	OK
FY (lb)	1	400	0.004	OK
FZ (lb)	-21113	260	81.205	Failed
FR (lb)	21113	570	37.041	Failed
MX (ft-lb)	-152	980	0.155	OK
MY (ft-lb)	23	500	0.047	OK
MZ (ft-lb)	81	740	0.109	OK
MR (ft-lb)	173	1330	0.130	OK

Condition F.1.2.1 for discharge node 250 failed ***

Soil Restraints

Soil restraints are shown when you have buried piping in your model.



#	From	To	Name	Type	Axial Stiffness (lb/inch)	Axial Max Load (lb)	Transverse Stiffness (lb/inch)	Transverse Max Load (lb)	Vertical Down Stiffness (lb/inch)	Vertical Down Max Load (lb)	Vertical Up Stiffness (lb/inch)	Vertical Up Max Load (lb)
1	10	20A	S2	Cohesive	91444	109733	6.304E+7	7.56E+7	3.164E+7	3.80E+7	6.400E+7	7.68E+7
2	20B	30A	S2	Cohesive	13136	15764	1.166E+7	1.40E+7	4.440E+6	5328256	1.223E+7	1.47E+7
3	30B	40	S2	Cohesive	6563.5	7876.2	6.686E+6	8023540	2.308E+6	2769828	7.464E+6	8957105
4	40	50	S2	Cohesive	712.9	855.5	726244	871493	250709	300850	810745	972894
5	50	60	S2	Cohesive	964.5	1157.4	982553	1179063	339190	407028	1.097E+6	1316251
6	70	80	S2	Cohesive	964.5	1157.4	982553	1179063	339190	407028	1.097E+6	1316251
7	80	90	S2	Cohesive	712.9	855.5	726244	871493	250709	300850	810745	972894
8	90	100	S2	Cohesive	19500	23400	1.986E+7	2.38E+7	6.858E+6	8229074	2.218E+7	2.66E+7

Support Load Summary for 150+ Load Combinations

This gives you a summary of many combinations of different load cases by showing the algebraically minimum and maximum loads. This table can be of use to a support designer.

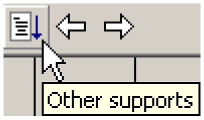
CAEPIPE will report support load summaries at every support in the model for over 150 load combinations (not all are shown below).

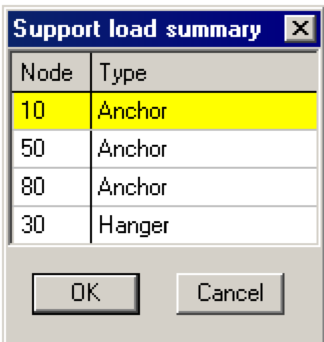
Caepipe : Support load summary for anchor at node 10 - [complex1.res (\\CDV-VISIO...

File Results View Options Window Help

Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
Cold spring1+Settlement	-19755	4678	-2644	-1082	-8746	6836
Cold spring2+Settlement	-21765	2995	-3742	2427	-4196	4535
Cold spring3+Settlement	16248	1918	17258	4100	-99573	3063
Hydrotest	-328	2813	148	885	-1360	4286
Cold spring0+Wind	7104	2872	11109	1498	-68550	4366
Cold spring1+Wind	-19906	5633	-2950	-3273	-6597	8141
Cold spring2+Wind	-21916	3949	-4048	237	-2047	5840
Cold spring3+Wind	16097	2873	16952	1909	-97424	4368
Cold spring0+Seismic	8869	3162	12452	2539	-61037	4763
Cold spring0-Seismic	5342	2597	9641	481	-75763	3991
Cold spring1+Seismic	-18141	5923	-1606	-2231	916	8538
Cold spring1-Seismic	-21668	5358	-4418	-4289	-13811	7766
Cold spring2+Seismic	-20151	4240	-2704	1278	5466	6237
Cold spring2-Seismic	-23678	3675	-5516	-780	-9261	5465
Cold spring3+Seismic	17862	3163	18296	2951	-89911	4765
Cold spring3-Seismic	14335	2598	15484	893	-104638	3992
Cold spring0+Response	7934	3077	11783	2351	-63331	4646
Cold spring0-Response	6277	2682	10310	669	-73469	4107
Cold spring1+Response	-19076	5838	-2275	-2419	-1378	8421
Cold spring1-Response	-20733	5443	-3748	-4101	-11516	7882
Cold spring2+Response	-21085	4154	-3374	1090	3171	6120
Cold spring2-Response	-22743	3760	-4847	-592	-6967	5581
Cold spring3+Response	16927	3078	17626	2763	-92206	4648
Cold spring3-Response	15270	2683	16153	1081	-102344	4109
Cold spring0+Time history	7105	2879	11046	1511	-68395	4376
Cold spring1+Time history	-19905	5640	-3013	-3260	-6442	8151
Cold spring2+Time history	-21915	3957	-4111	250	-1893	5850
Cold spring3+Time history	16098	2880	16889	1922	-97270	4378
Cold spring0+Harmonic	7106	2879	11046	1510	-68400	4377
Cold spring0-Harmonic	7105	2879	11046	1510	-68401	4377
Cold spring1+Harmonic	-19904	5640	-3012	-3260	-6447	8152
Cold spring1-Harmonic	-19905	5640	-3012	-3260	-6448	8152
Cold spring2+Harmonic	-21914	3957	-4110	249	-1897	5851
Cold spring2-Harmonic	-21914	3957	-4110	249	-1898	5851
Cold spring3+Harmonic	16099	2881	16890	1922	-97274	4379
Cold spring3-Harmonic	16098	2881	16890	1922	-97275	4378
Maximum	17862	5923	18296	4100	5466	8538
Minimum	-23678	1917	-5516	-4289	-104638	3061

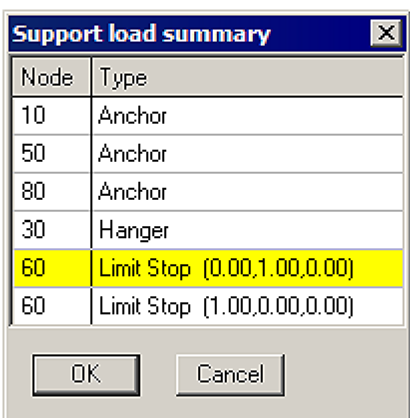
The supports at which a load summary is available can be shown by clicking on the Other Supports button (immediately to the left of the left white arrow button). Select the support from the list of available supports.





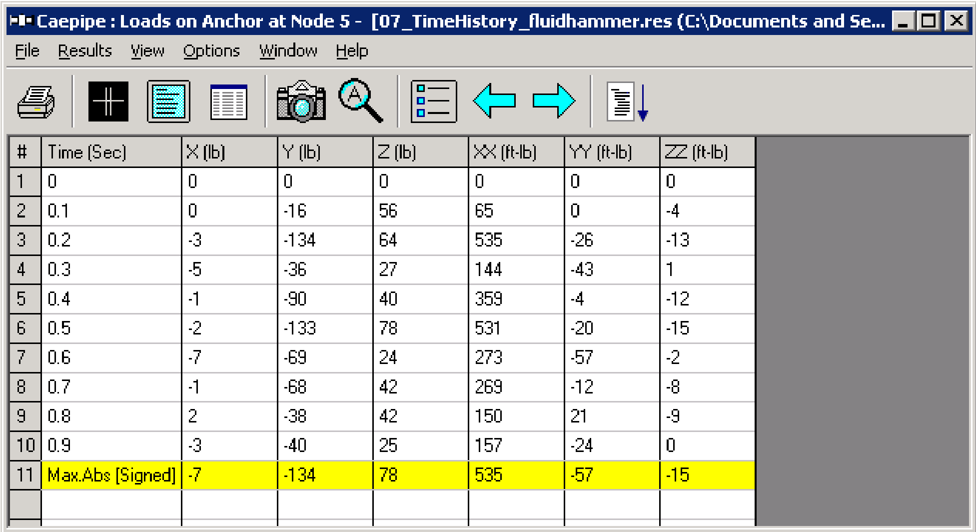
Here, support load summaries are available for the four supports shown. Summaries may be included in the "Print to file" option (in addition to inclusion in formatted reports).

To clearly show the direction of each limit stop among multiple limit stops at the same node, the direction is shown alongside the limit stop (see node 60 below).

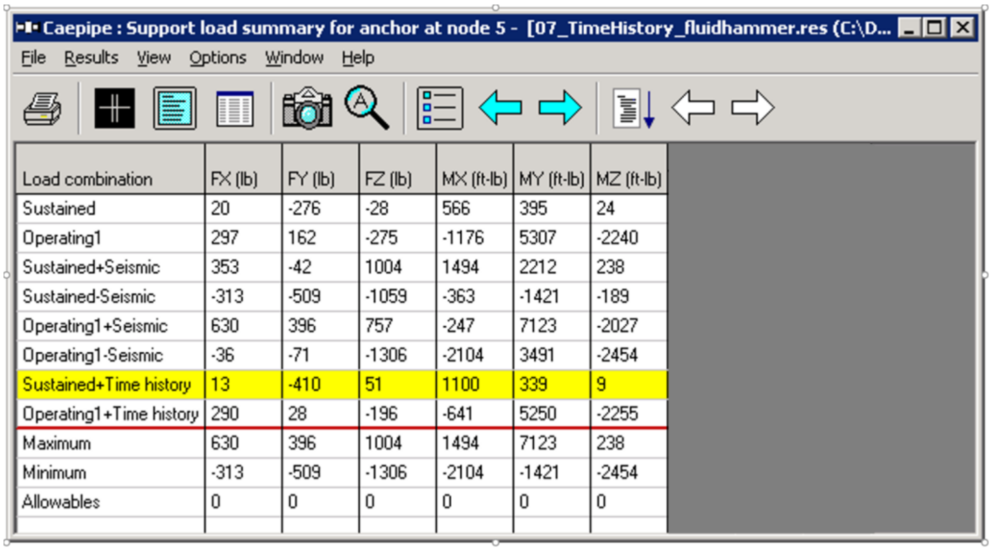


Note:

For Time History load case, CAEPIPE will find Signed Forces and/or Moments with Maximum Absolute Magnitude under Support Loads at a particular node as shown below and add the same along with its sign to other load cases such as Sustained, Operating, etc. in the Support Load Summary results.



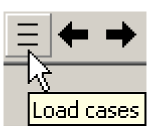
#	Time (Sec)	X (lb)	Y (lb)	Z (lb)	XX (ft-lb)	YY (ft-lb)	ZZ (ft-lb)
1	0	0	0	0	0	0	0
2	0.1	0	-16	56	65	0	-4
3	0.2	-3	-134	64	535	-26	-13
4	0.3	-5	-36	27	144	-43	1
5	0.4	-1	-90	40	359	-4	-12
6	0.5	-2	-133	78	531	-20	-15
7	0.6	-7	-69	24	273	-57	-2
8	0.7	-1	-68	42	269	-12	-8
9	0.8	2	-38	42	150	21	-9
10	0.9	-3	-40	25	157	-24	0
11	Max.Abs (Signed)	-7	-134	78	535	-57	-15

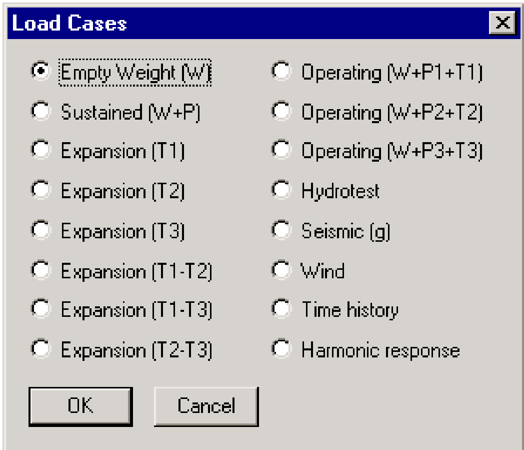


Load combination	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
Sustained	20	-276	-28	566	395	24
Operating1	297	162	-275	-1176	5307	-2240
Sustained+Seismic	353	-42	1004	1494	2212	238
Sustained-Seismic	-313	-509	-1059	-363	-1421	-189
Operating1+Seismic	630	396	757	-247	7123	-2027
Operating1-Seismic	-36	-71	-1306	-2104	3491	-2454
Sustained+Time history	13	-410	51	1100	339	9
Operating1+Time history	290	28	-196	-641	5250	-2255
Maximum	630	396	1004	1494	7123	238
Minimum	-313	-509	-1306	-2104	-1421	-2454
Allowables	0	0	0	0	0	0

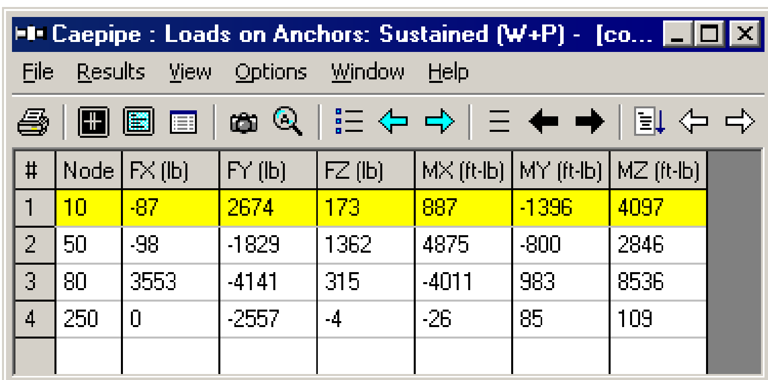
Support Loads

Support Loads are shown for all support types. For a support type, the support loads can be shown for all load cases by clicking on the black right/left arrow or simply press arrow keys on the keyboard. A list of load cases can be shown by clicking on the Load cases button.



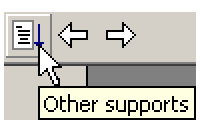


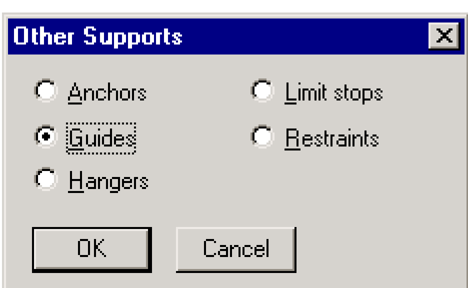
Support loads on all the supports in the model are shown in the next window. The window first shows anchor loads, if present. Then, when you click on the left or right white arrow, it shows you loads at different supports.



#	Node	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
1	10	-87	2674	173	887	-1396	4097
2	50	-98	-1829	1362	4875	-800	2846
3	80	3553	-4141	315	-4011	983	8536
4	250	0	-2557	-4	-26	85	109

You can find out which other supports are in the model by clicking on the Other Supports button (immediately to the left of the left white arrow button). When you click on this button, the list of available support types is shown. Select the type.

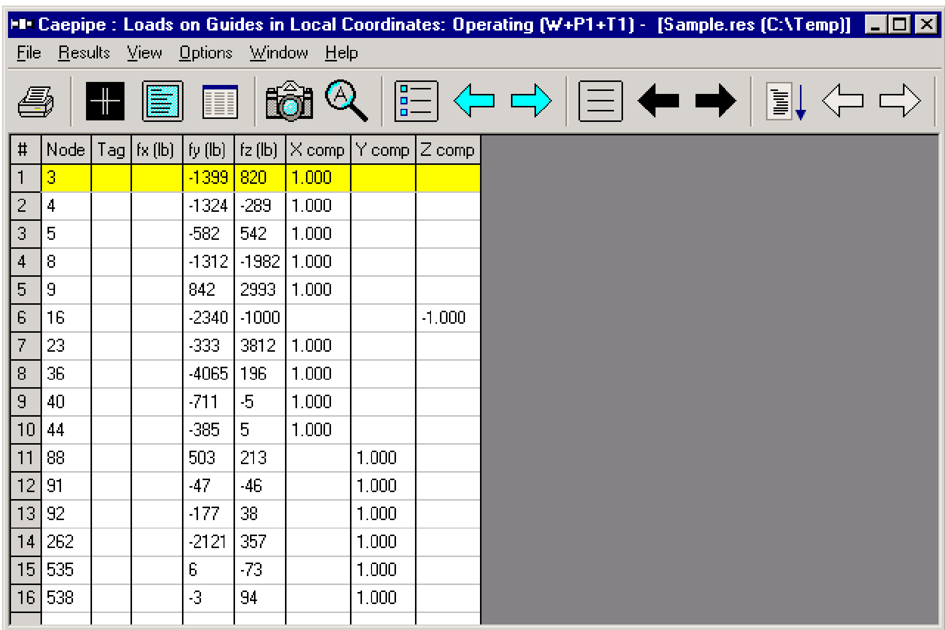




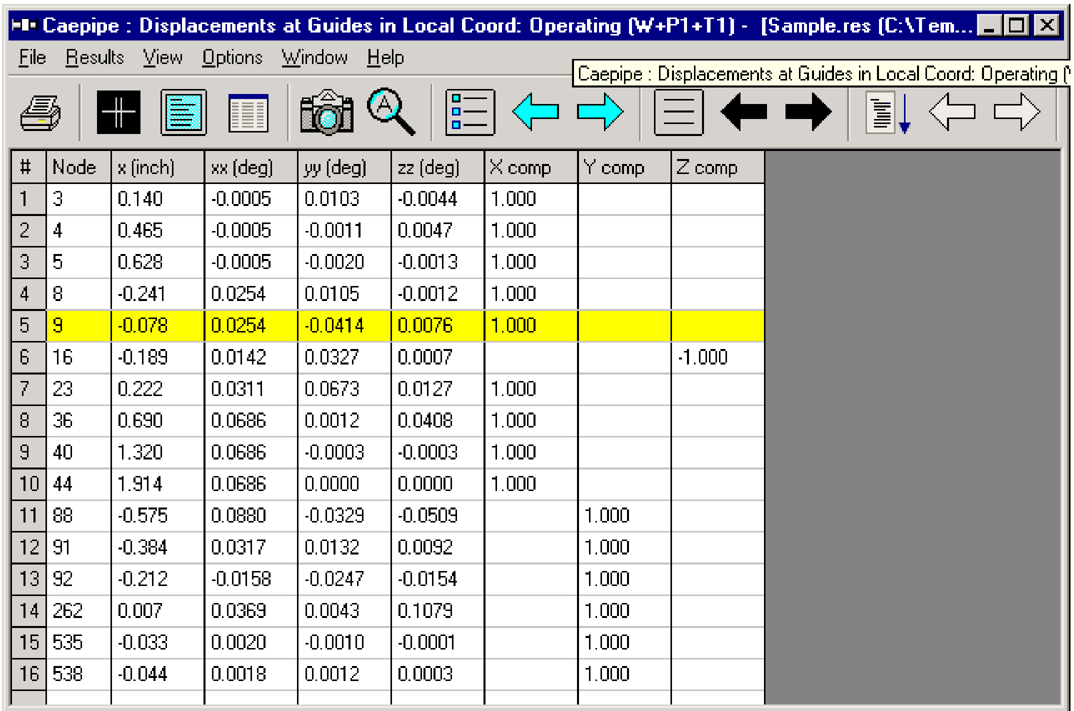
In any window of CAEPIPE, whenever it displays the results in LCS, it will label the heading texts with the "Lower Case" letters. For example, the "Forces & Moments" in LCS are designated using "fx/fy/fz" and "mx/my/mz". Similarly, the displacements in LCS using "x/y/z/xx/yy/zz".

On the other hand, CAEPIPE displays the results in GCS by designating the heading texts with the “Upper Case” letters”. For example, the “Forces & Moments” in GCS are designated using “FX/FY/FZ” and “MX/MY/MZ” and the displacements in GCS as “X/Y/Z/XX/YY/ZZ”.

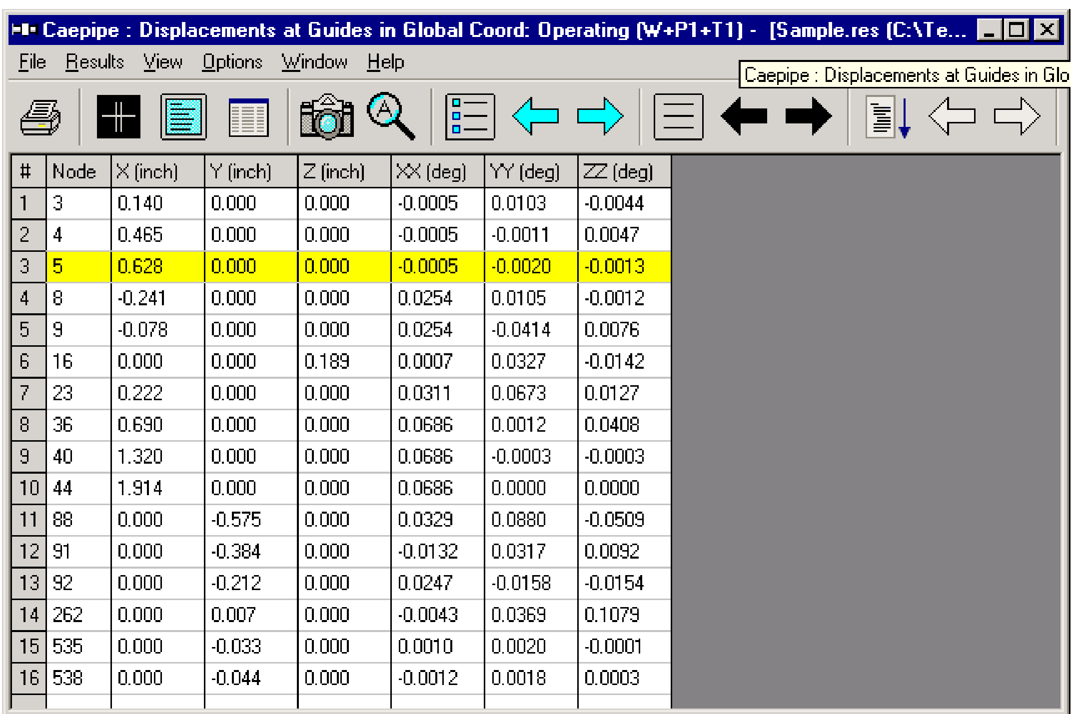
For example, when a Guide is located in a vertical section of pipe, then the local x-axis for the Guide is vertical, whereas the local y-axis and local z-axis are in the horizontal plane. Local Coordinate System (LCS) for the Guide can be displayed through View > List > Guide and right-click mouse and select the option “Show LCS”. Since the pipe can freely move axially at the Guide (assuming zero friction), only lateral forces fy and fz are generated at the Guide in the local y-axis and local z-axis directions. On the other hand, pipe displacements at the Guide can be seen both in Global and Local Coordinate Systems. These can be seen in the snapshots below.



#	Node	Tag	fx (lb)	fy (lb)	fz (lb)	X comp	Y comp	Z comp
1	3			-1399	820	1.000		
2	4			-1324	-289	1.000		
3	5			-582	542	1.000		
4	8			-1312	-1982	1.000		
5	9			842	2993	1.000		
6	16			-2340	-1000			-1.000
7	23			-333	3812	1.000		
8	36			-4065	196	1.000		
9	40			-711	-5	1.000		
10	44			-385	5	1.000		
11	88			503	213		1.000	
12	91			-47	-46		1.000	
13	92			-177	38		1.000	
14	262			-2121	357		1.000	
15	535			6	-73		1.000	
16	538			-3	94		1.000	



#	Node	x (inch)	xx (deg)	yy (deg)	zz (deg)	X comp	Y comp	Z comp
1	3	0.140	-0.0005	0.0103	-0.0044	1.000		
2	4	0.465	-0.0005	-0.0011	0.0047	1.000		
3	5	0.628	-0.0005	-0.0020	-0.0013	1.000		
4	8	-0.241	0.0254	0.0105	-0.0012	1.000		
5	9	-0.078	0.0254	-0.0414	0.0076	1.000		
6	16	-0.189	0.0142	0.0327	0.0007			-1.000
7	23	0.222	0.0311	0.0673	0.0127	1.000		
8	36	0.690	0.0686	0.0012	0.0408	1.000		
9	40	1.320	0.0686	-0.0003	-0.0003	1.000		
10	44	1.914	0.0686	0.0000	0.0000	1.000		
11	88	-0.575	0.0880	-0.0329	-0.0509		1.000	
12	91	-0.384	0.0317	0.0132	0.0092		1.000	
13	92	-0.212	-0.0158	-0.0247	-0.0154		1.000	
14	262	0.007	0.0369	0.0043	0.1079		1.000	
15	535	-0.033	0.0020	-0.0010	-0.0001		1.000	
16	538	-0.044	0.0018	0.0012	0.0003		1.000	



#	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	3	0.140	0.000	0.000	-0.0005	0.0103	-0.0044
2	4	0.465	0.000	0.000	-0.0005	-0.0011	0.0047
3	5	0.628	0.000	0.000	-0.0005	-0.0020	-0.0013
4	8	-0.241	0.000	0.000	0.0254	0.0105	-0.0012
5	9	-0.078	0.000	0.000	0.0254	-0.0414	0.0076
6	16	0.000	0.000	0.189	0.0007	0.0327	-0.0142
7	23	0.222	0.000	0.000	0.0311	0.0673	0.0127
8	36	0.690	0.000	0.000	0.0686	0.0012	0.0408
9	40	1.320	0.000	0.000	0.0686	-0.0003	-0.0003
10	44	1.914	0.000	0.000	0.0686	0.0000	0.0000
11	88	0.000	-0.575	0.000	0.0329	0.0880	-0.0509
12	91	0.000	-0.384	0.000	-0.0132	0.0317	0.0092
13	92	0.000	-0.212	0.000	0.0247	-0.0158	-0.0154
14	262	0.000	0.007	0.000	-0.0043	0.0369	0.1079
15	535	0.000	-0.033	0.000	0.0010	0.0020	-0.0001
16	538	0.000	-0.044	0.000	-0.0012	0.0018	0.0003

In summary, CAEPIPE displays / output the "Support Loads" for different types of supports in different coordinate systems as stated below.

a. Anchor – Always in Global Coordinate System (GCS).

b. Rigid Restraints – Always in GCS.

c. Guide – By default in Local Coordinate System (LCS). User can view the "Support Loads" in GCS by clicking the icon "G" available in the tool bar.

d. Limit Stops (Resting / Roller Supports) – By default in LCS. User can view the "Support Loads" in GCS by clicking the icon "G" available in the tool bar.

e. Skewed Restraints – Always in the direction of the support defined.

f. Spring Hangers / Rod Hangers / User Hanger / Constant Support Hangers – Always in Global Vertical direction.

Global Forces & Moments can be transformed into Local Forces & Moments using a Rotation Matrix formed by the direction vector of the preceding element (or succeeding element, if there is no preceding element). For your convenience, an Excel sheet that converts Global Forces & Moments to the Local Forces & Moments once the required values are input is available at www.sstusa.com/downloads/GCS_LCS.xlsx.

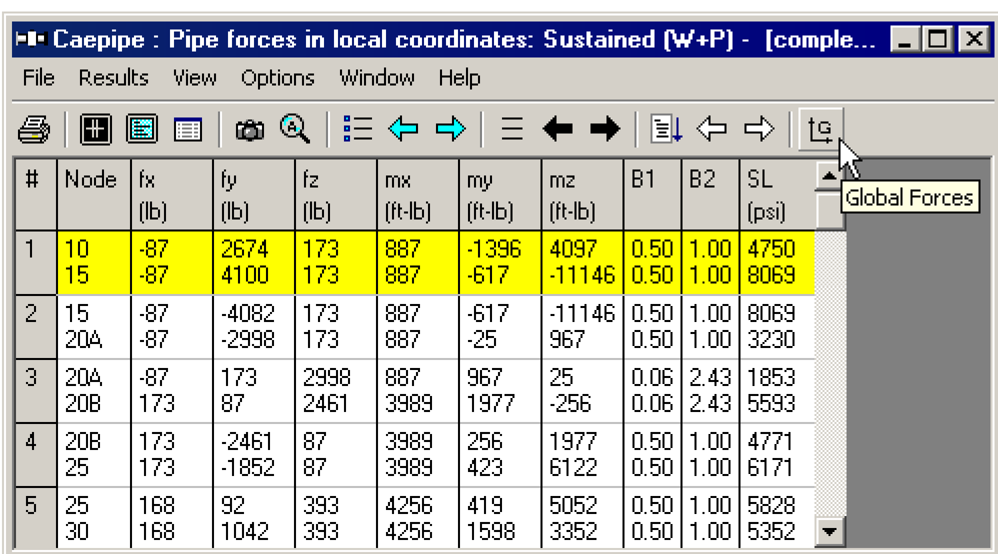
To validate this Excel sheet procedure, two (2) CAEPIPE models were analyzed. First one has an Anchor attached to a skewed pipe wherein the Anchor loads are output in GCS (www.sstusa.com/downloads/BMS.mod). In the 2nd model, the Anchor has been replaced with 6 skewed restraints (3 translational and 3 rotational) in the adjacent element's LCS system. The GCS results from the first model were entered in the Excel sheet in order to obtain forces and moments in LCS. These forces and moments in LCS are identical to the results from the 2nd model with skewed restraints (www.sstusa.com/downloads/BMS_SR.mod). We recommend that an independent validation of the excel procedure is carried out at your end to convert Global Forces & Moments to the Local Forces & Moments.

Results from MultipleThermal and Wind Loads

Displacements, Support Loads, Element Forces, etc., are calculated for the below shown thermal and wind load cases, among others. The sorted stresses screen will always display for a specific node the highest expansion stress among all the selected thermal load cases. Similarly, if any of the four wind load cases happens to be the worst among all occasional loads for a specific node, then that occasional stress value will be shown under occasional stresses.

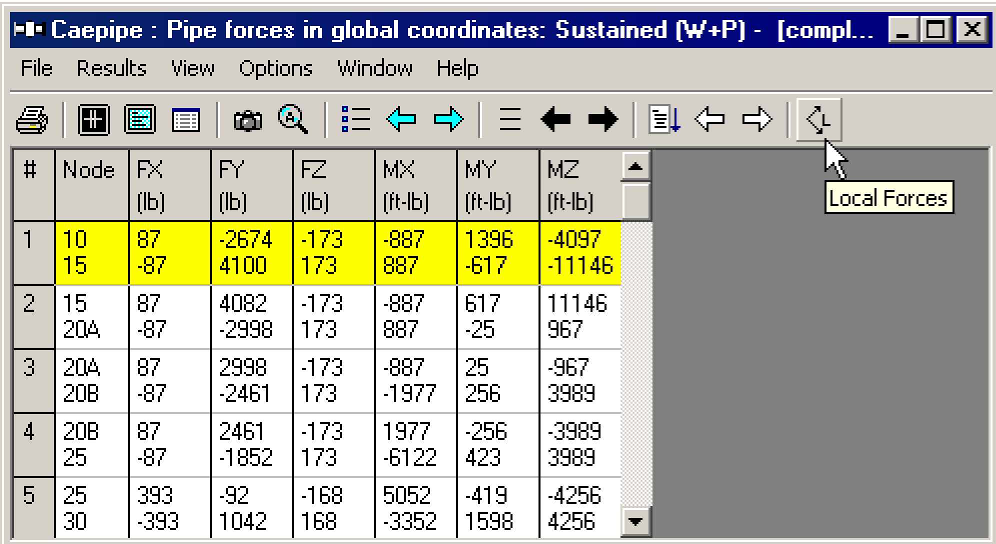
Element Forces

Forces and moments on all pipe elements are shown next. If you had other elements such as a valve, you can display forces and moments for them too by clicking on the Other Forces button on the toolbar.



#	Node	fx (lb)	fy (lb)	fz (lb)	mx (ft-lb)	my (ft-lb)	mz (ft-lb)	B1	B2	SL (psi)
1	10 15	-87 -87	2674 4100	173 173	887 887	-1396 -617	4097 -11146	0.50 0.50	1.00 1.00	4750 8069
2	15 20A	-87 -87	-4082 -2998	173 173	887 887	-617 -25	-11146 967	0.50 0.50	1.00 1.00	8069 3230
3	20A 20B	-87 173	173 87	2998 2461	887 3989	967 1977	25 -256	0.06 0.06	2.43 2.43	1853 5593
4	20B 25	173 173	-2461 -1852	87 87	3989 3989	256 423	1977 6122	0.50 0.50	1.00 1.00	4771 6171
5	25 30	168 168	92 1042	393 393	4256 4256	419 1598	5052 3352	0.50 0.50	1.00 1.00	5828 5352

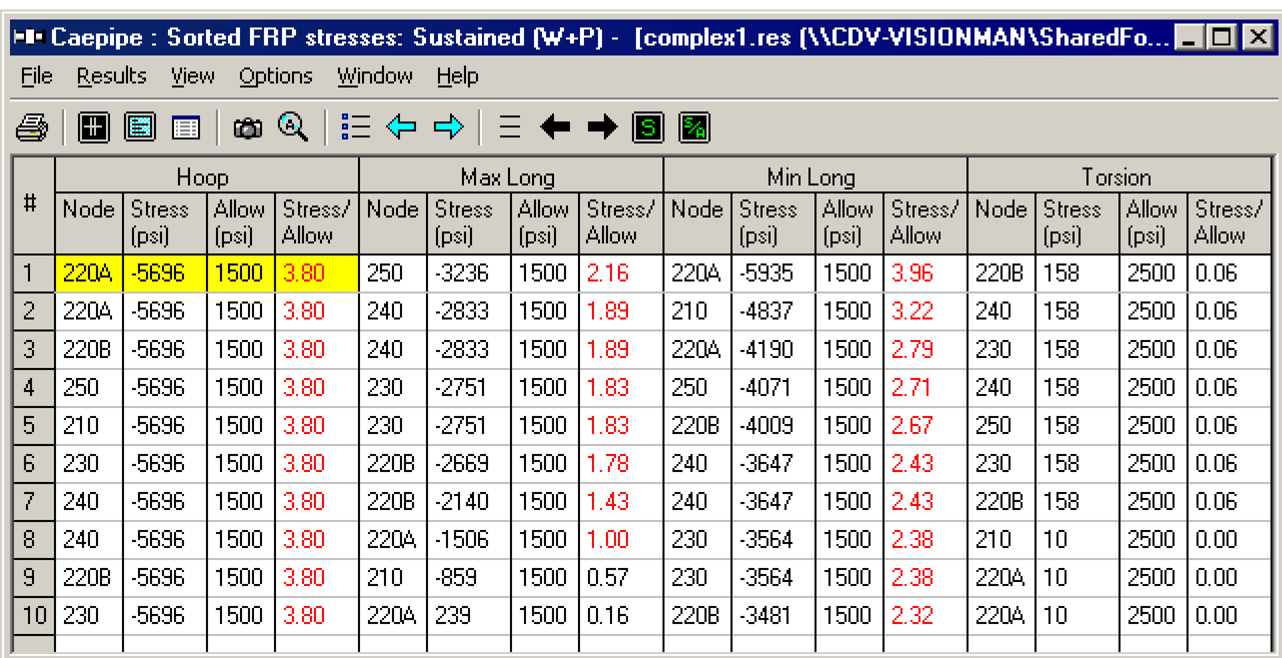
The default view for forces and moments is in the local coordinate system. They can be shown in the global coordinate system too by clicking on the Global forces button (or selecting Global Forces command from the Results menu, hotkey: F7). When you select Global forces, the forces and moments are shown in global coordinates and the button changes in the same location (on the toolbar) to an "L" for Local forces. So, should you want to return to pipe forces in local coordinates, simply click again at the same location on the Local forces button.



#	Node	FX (lb)	FY (lb)	FZ (lb)	MX (ft-lb)	MY (ft-lb)	MZ (ft-lb)
1	10	87	-2674	-173	-887	1396	-4097
	15	-87	4100	173	887	-617	-11146
2	15	87	4082	-173	-887	617	11146
	20A	-87	-2998	173	887	-25	967
3	20A	87	2998	-173	-887	25	-967
	20B	-87	-2461	173	-1977	256	3989
4	20B	87	2461	-173	1977	-256	-3989
	25	-87	-1852	173	-6122	423	3989
5	25	393	-92	-168	5052	-419	-4256
	30	-393	1042	168	-3352	1598	4256

The convention in CAEPIPE is to display the headings for global forces and moments in uppercase (FX, FY, FZ, etc.) and display the headings for local forces and moments in lower case (fx, fy, fz, etc.).

Sorted FRP Stresses

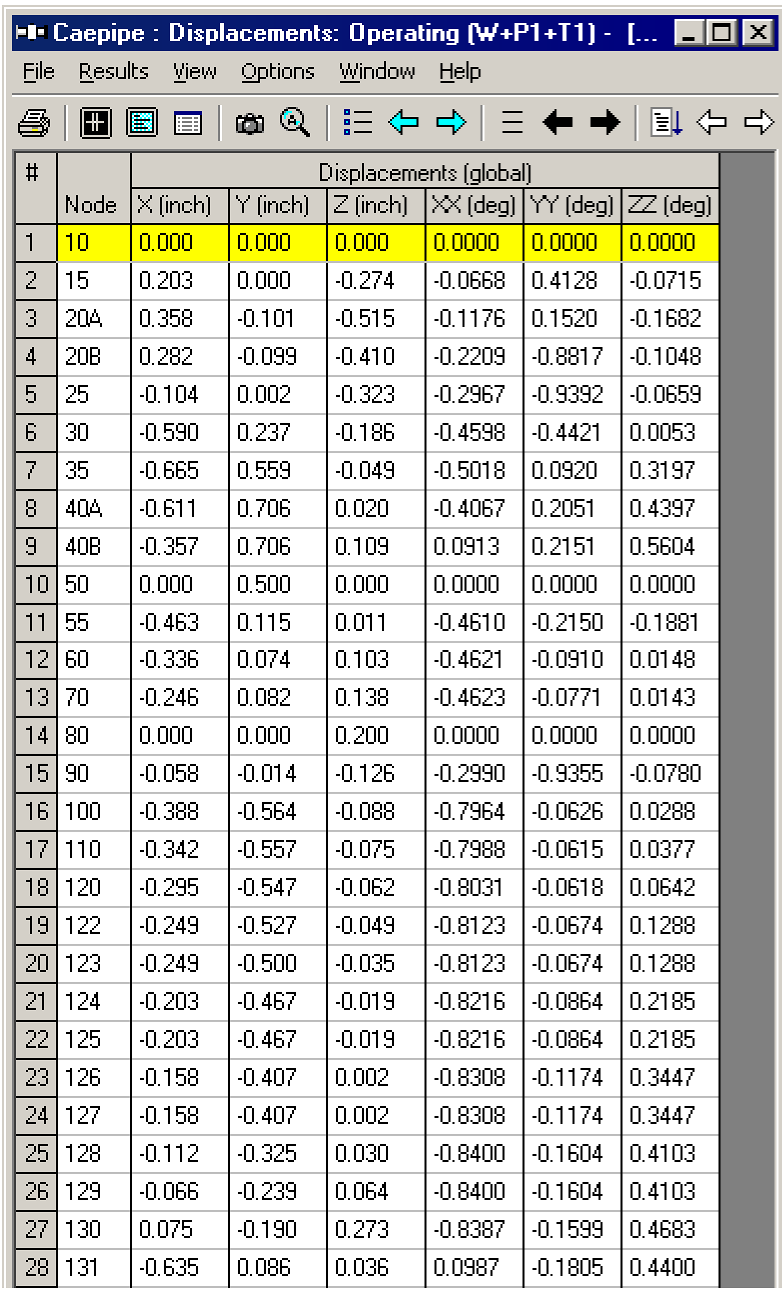


#	Hoop				Max Long				Min Long				Torsion			
	Node	Stress (psi)	Allow (psi)	Stress/ Allow	Node	Stress (psi)	Allow (psi)	Stress/ Allow	Node	Stress (psi)	Allow (psi)	Stress/ Allow	Node	Stress (psi)	Allow (psi)	Stress/ Allow
1	220A	-5696	1500	3.80	250	-3236	1500	2.16	220A	-5935	1500	3.96	220B	158	2500	0.06
2	220A	-5696	1500	3.80	240	-2833	1500	1.89	210	-4837	1500	3.22	240	158	2500	0.06
3	220B	-5696	1500	3.80	240	-2833	1500	1.89	220A	-4190	1500	2.79	230	158	2500	0.06
4	250	-5696	1500	3.80	230	-2751	1500	1.83	250	-4071	1500	2.71	240	158	2500	0.06
5	210	-5696	1500	3.80	230	-2751	1500	1.83	220B	-4009	1500	2.67	250	158	2500	0.06
6	230	-5696	1500	3.80	220B	-2669	1500	1.78	240	-3647	1500	2.43	230	158	2500	0.06
7	240	-5696	1500	3.80	220B	-2140	1500	1.43	240	-3647	1500	2.43	220B	158	2500	0.06
8	240	-5696	1500	3.80	220A	-1506	1500	1.00	230	-3564	1500	2.38	210	10	2500	0.00
9	220B	-5696	1500	3.80	210	-859	1500	0.57	230	-3564	1500	2.38	220A	10	2500	0.00
10	230	-5696	1500	3.80	220A	239	1500	0.16	220B	-3481	1500	2.32	220A	10	2500	0.00

The various FRP stresses such as hoop, maximum and minimum longitudinal and torsional stresses are sorted here by descending order of stress ratios assuming you have input the material allowables.

Displacements

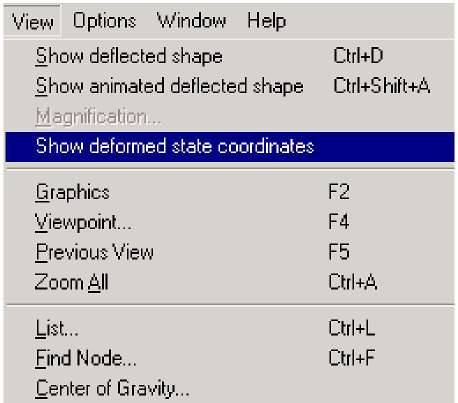
Displacements for all load cases can be shown (operating load case shown here). Click on the black left/right arrow to show displacements for other load cases. You can show the deflected shape in the graphics window for any load case by clicking on the button to the left of the “A” button.



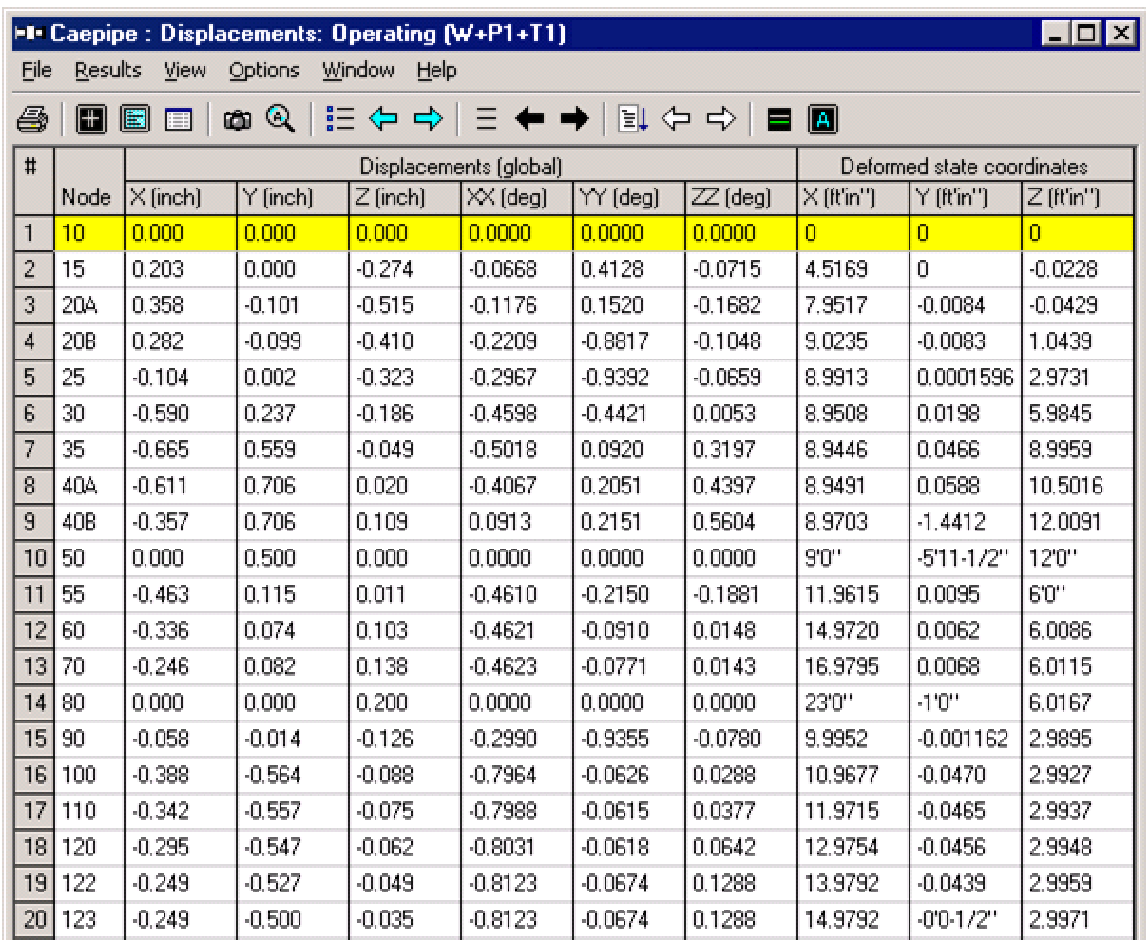
#	Node	Displacements (global) X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	15	0.203	0.000	-0.274	-0.0668	0.4128	-0.0715
3	20A	0.358	-0.101	-0.515	-0.1176	0.1520	-0.1682
4	20B	0.282	-0.099	-0.410	-0.2209	-0.8817	-0.1048
5	25	-0.104	0.002	-0.323	-0.2967	-0.9392	-0.0659
6	30	-0.590	0.237	-0.186	-0.4598	-0.4421	0.0053
7	35	-0.665	0.559	-0.049	-0.5018	0.0920	0.3197
8	40A	-0.611	0.706	0.020	-0.4067	0.2051	0.4397
9	40B	-0.357	0.706	0.109	0.0913	0.2151	0.5604
10	50	0.000	0.500	0.000	0.0000	0.0000	0.0000
11	55	-0.463	0.115	0.011	-0.4610	-0.2150	-0.1881
12	60	-0.336	0.074	0.103	-0.4621	-0.0910	0.0148
13	70	-0.246	0.082	0.138	-0.4623	-0.0771	0.0143
14	80	0.000	0.000	0.200	0.0000	0.0000	0.0000
15	90	-0.058	-0.014	-0.126	-0.2990	-0.9355	-0.0780
16	100	-0.388	-0.564	-0.088	-0.7964	-0.0626	0.0288
17	110	-0.342	-0.557	-0.075	-0.7988	-0.0615	0.0377
18	120	-0.295	-0.547	-0.062	-0.8031	-0.0618	0.0642
19	122	-0.249	-0.527	-0.049	-0.8123	-0.0674	0.1288
20	123	-0.249	-0.500	-0.035	-0.8123	-0.0674	0.1288
21	124	-0.203	-0.467	-0.019	-0.8216	-0.0864	0.2185
22	125	-0.203	-0.467	-0.019	-0.8216	-0.0864	0.2185
23	126	-0.158	-0.407	0.002	-0.8308	-0.1174	0.3447
24	127	-0.158	-0.407	0.002	-0.8308	-0.1174	0.3447
25	128	-0.112	-0.325	0.030	-0.8400	-0.1604	0.4103
26	129	-0.066	-0.239	0.064	-0.8400	-0.1604	0.4103
27	130	0.075	-0.190	0.273	-0.8387	-0.1599	0.4683
28	131	-0.635	0.086	0.036	0.0987	-0.1805	0.4400

You can animate the same deflected shape by clicking on Show animated deflected shape (last) button in the toolbar.

You have the option of showing the deformed state coordinates in the global coordinate system along with the displacements at each node. Is snapshot below updated from V8.0?

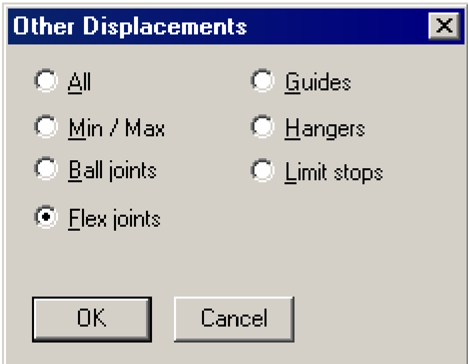


These deformed state coordinates are useful in locating hanger attachment points on the steel structures or concrete slabs/walls from which the hangers are to be hung, such that the hangers remain almost "vertical" (and not skewed) during normal operating load case.

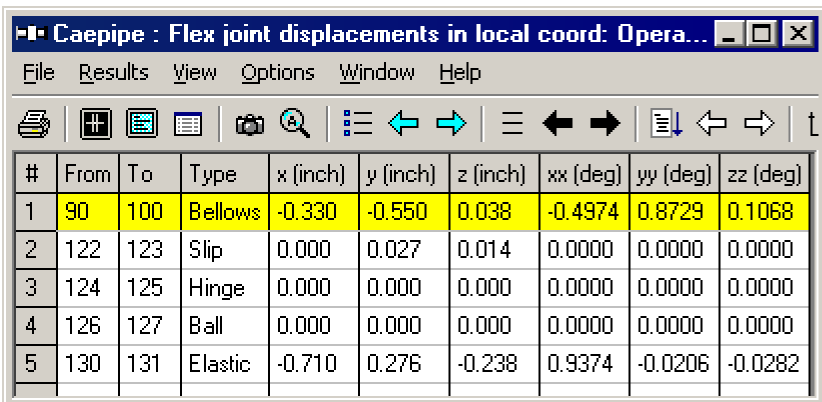


#		Displacements (global)						Deformed state coordinates		
	Node	X (inch)	Y (inch)	Z (inch)	XX (deg)	YY (deg)	ZZ (deg)	X (ft'in")	Y (ft'in")	Z (ft'in")
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000	0	0	0
2	15	0.203	0.000	-0.274	-0.0668	0.4128	-0.0715	4.5169	0	-0.0228
3	20A	0.358	-0.101	-0.515	-0.1176	0.1520	-0.1682	7.9517	-0.0084	-0.0429
4	20B	0.282	-0.099	-0.410	-0.2209	-0.8817	-0.1048	9.0235	-0.0083	1.0439
5	25	-0.104	0.002	-0.323	-0.2967	-0.9392	-0.0659	8.9913	0.0001596	2.9731
6	30	-0.590	0.237	-0.186	-0.4598	-0.4421	0.0053	8.9508	0.0198	5.9845
7	35	-0.665	0.559	-0.049	-0.5018	0.0920	0.3197	8.9446	0.0466	8.9959
8	40A	-0.611	0.706	0.020	-0.4067	0.2051	0.4397	8.9491	0.0588	10.5016
9	40B	-0.357	0.706	0.109	0.0913	0.2151	0.5604	8.9703	-1.4412	12.0091
10	50	0.000	0.500	0.000	0.0000	0.0000	0.0000	9'0"	-5'11-1/2"	12'0"
11	55	-0.463	0.115	0.011	-0.4610	-0.2150	-0.1881	11.9615	0.0095	6'0"
12	60	-0.336	0.074	0.103	-0.4621	-0.0910	0.0148	14.9720	0.0062	6.0086
13	70	-0.246	0.082	0.138	-0.4623	-0.0771	0.0143	16.9795	0.0068	6.0115
14	80	0.000	0.000	0.200	0.0000	0.0000	0.0000	23'0"	-1'0"	6.0167
15	90	-0.058	-0.014	-0.126	-0.2990	-0.9355	-0.0780	9.9952	-0.001162	2.9895
16	100	-0.388	-0.564	-0.088	-0.7964	-0.0626	0.0288	10.9677	-0.0470	2.9927
17	110	-0.342	-0.557	-0.075	-0.7988	-0.0615	0.0377	11.9715	-0.0465	2.9937
18	120	-0.295	-0.547	-0.062	-0.8031	-0.0618	0.0642	12.9754	-0.0456	2.9948
19	122	-0.249	-0.527	-0.049	-0.8123	-0.0674	0.1288	13.9792	-0.0439	2.9959
20	123	-0.249	-0.500	-0.035	-0.8123	-0.0674	0.1288	14.9792	-0'0-1/2"	2.9971

Also, you can show Minimum/Maximum displacements, displacements at flexible joints (if any) by clicking on the Other displacements button and selecting the item of interest (or by clicking on the left/right white arrows).

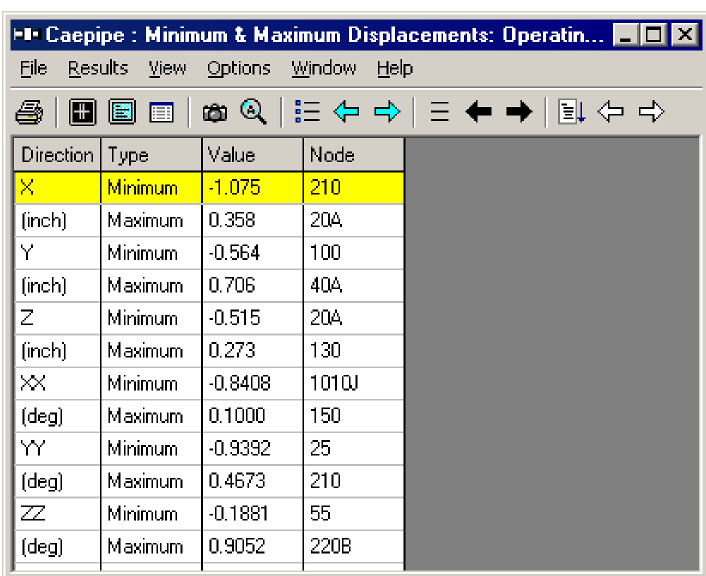


Displacements in local coordinates for all flexible joints (bellows, ball, hinge, etc.) are shown under Flex joints.



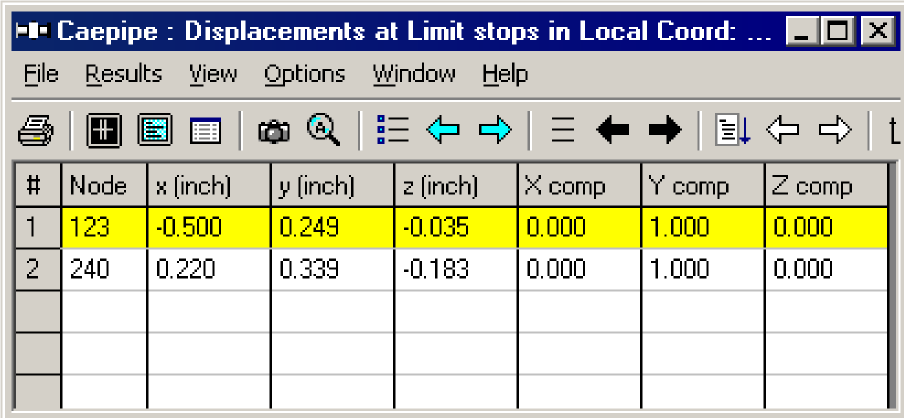
#	From	To	Type	x (inch)	y (inch)	z (inch)	xx (deg)	yy (deg)	zz (deg)
1	90	100	Bellows	-0.330	-0.550	0.038	-0.4974	0.8729	0.1068
2	122	123	Slip	0.000	0.027	0.014	0.0000	0.0000	0.0000
3	124	125	Hinge	0.000	0.000	0.000	0.0000	0.0000	0.0000
4	126	127	Ball	0.000	0.000	0.000	0.0000	0.0000	0.0000
5	130	131	Elastic	-0.710	0.276	-0.238	0.9374	-0.0206	-0.0282

Minimum/Maximum displacements for each load case can be shown (Operating load case shown below). You can show minimum/maximum displacements for other load cases by clicking on the black arrows.



Direction	Type	Value	Node
X	Minimum	-1.075	210
(inch)	Maximum	0.358	20A
Y	Minimum	-0.564	100
(inch)	Maximum	0.706	40A
Z	Minimum	-0.515	20A
(inch)	Maximum	0.273	130
XX	Minimum	-0.8408	1010J
(deg)	Maximum	0.1000	150
YY	Minimum	-0.9392	25
(deg)	Maximum	0.4673	210
ZZ	Minimum	-0.1881	55
(deg)	Maximum	0.9052	220B

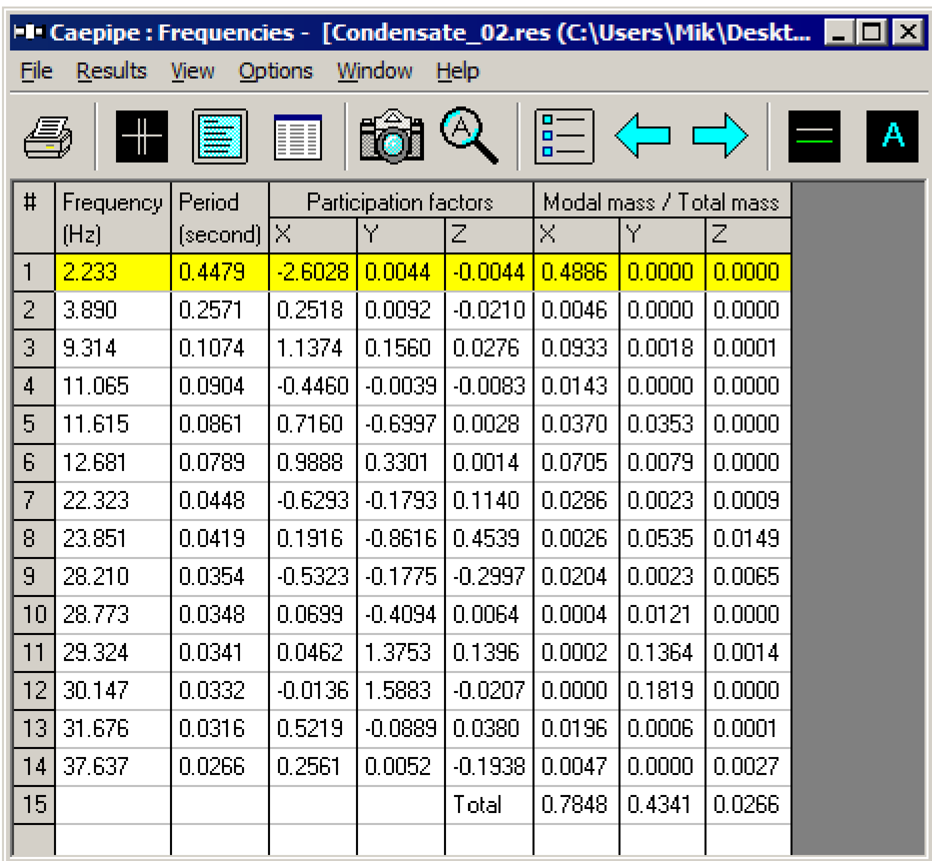
Displacements at “Other supports” can be shown (limit stops shown here).



#	Node	x (inch)	y (inch)	z (inch)	X comp	Y comp	Z comp
1	123	-0.500	0.249	-0.035	0.000	1.000	0.000
2	240	0.220	0.339	-0.183	0.000	1.000	0.000

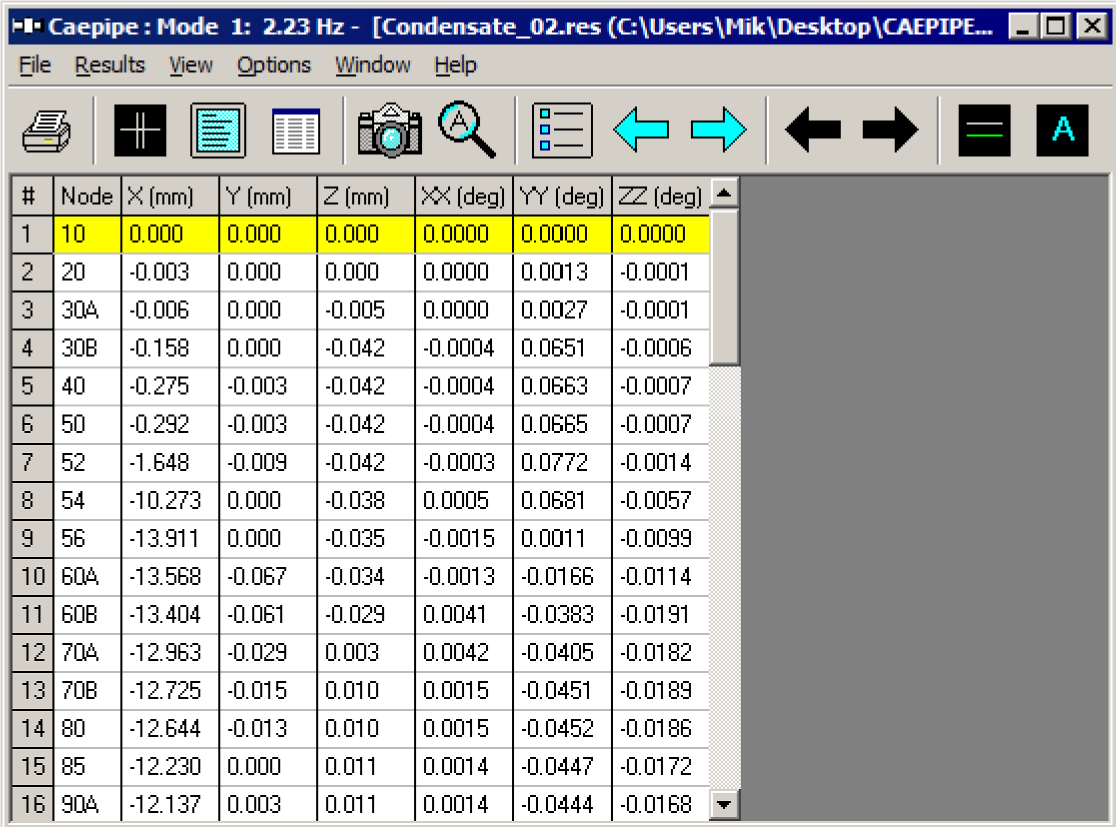
Frequencies

A list of natural frequencies, periods, modal participation factors and modal mass fractions is shown next. You can show each frequency’s mode shape graphically or animate it by clicking on Show mode shape or Show animated mode shape button in the toolbar.



#	Frequency (Hz)	Period (second)	Participation factors X	Participation factors Y	Participation factors Z	Modal mass / Total mass X	Modal mass / Total mass Y	Modal mass / Total mass Z
1	2.233	0.4479	-2.6028	0.0044	-0.0044	0.4886	0.0000	0.0000
2	3.890	0.2571	0.2518	0.0092	-0.0210	0.0046	0.0000	0.0000
3	9.314	0.1074	1.1374	0.1560	0.0276	0.0933	0.0018	0.0001
4	11.065	0.0904	-0.4460	-0.0039	-0.0083	0.0143	0.0000	0.0000
5	11.615	0.0861	0.7160	-0.6997	0.0028	0.0370	0.0353	0.0000
6	12.681	0.0789	0.9888	0.3301	0.0014	0.0705	0.0079	0.0000
7	22.323	0.0448	-0.6293	-0.1793	0.1140	0.0286	0.0023	0.0009
8	23.851	0.0419	0.1916	-0.8616	0.4539	0.0026	0.0535	0.0149
9	28.210	0.0354	-0.5323	-0.1775	-0.2997	0.0204	0.0023	0.0065
10	28.773	0.0348	0.0699	-0.4094	0.0064	0.0004	0.0121	0.0000
11	29.324	0.0341	0.0462	1.3753	0.1396	0.0002	0.1364	0.0014
12	30.147	0.0332	-0.0136	1.5883	-0.0207	0.0000	0.1819	0.0000
13	31.676	0.0316	0.5219	-0.0889	0.0380	0.0196	0.0006	0.0001
14	37.637	0.0266	0.2561	0.0052	-0.1938	0.0047	0.0000	0.0027
15					Total	0.7848	0.4341	0.0266

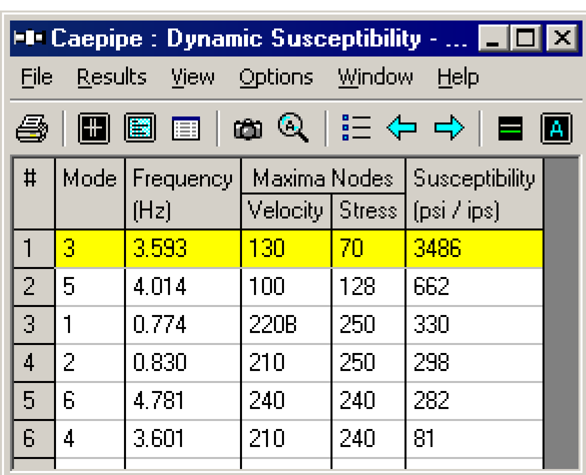
Each frequency's mode shape detail is shown in the next window. As in the earlier window, you can show graphically the mode shape or animate it by clicking on the appropriate button.



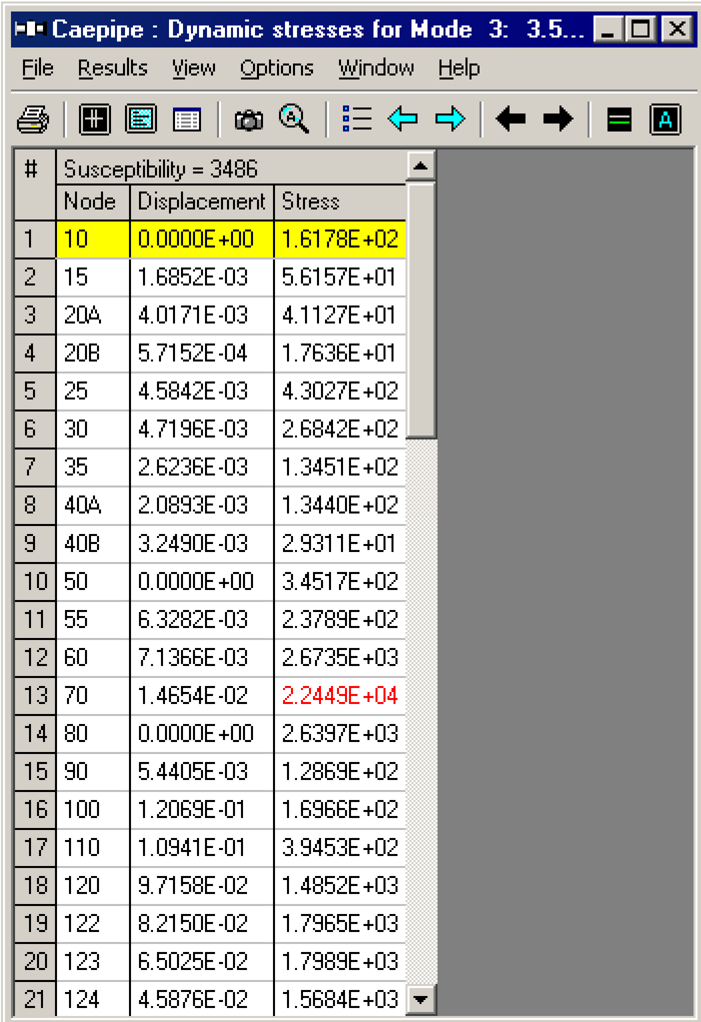
#	Node	X (mm)	Y (mm)	Z (mm)	XX (deg)	YY (deg)	ZZ (deg)
1	10	0.000	0.000	0.000	0.0000	0.0000	0.0000
2	20	-0.003	0.000	0.000	0.0000	0.0013	-0.0001
3	30A	-0.006	0.000	-0.005	0.0000	0.0027	-0.0001
4	30B	-0.158	0.000	-0.042	-0.0004	0.0651	-0.0006
5	40	-0.275	-0.003	-0.042	-0.0004	0.0663	-0.0007
6	50	-0.292	-0.003	-0.042	-0.0004	0.0665	-0.0007
7	52	-1.648	-0.009	-0.042	-0.0003	0.0772	-0.0014
8	54	-10.273	0.000	-0.038	0.0005	0.0681	-0.0057
9	56	-13.911	0.000	-0.035	-0.0015	0.0011	-0.0099
10	60A	-13.568	-0.067	-0.034	-0.0013	-0.0166	-0.0114
11	60B	-13.404	-0.061	-0.029	0.0041	-0.0383	-0.0191
12	70A	-12.963	-0.029	0.003	0.0042	-0.0405	-0.0182
13	70B	-12.725	-0.015	0.010	0.0015	-0.0451	-0.0189
14	80	-12.644	-0.013	0.010	0.0015	-0.0452	-0.0186
15	85	-12.230	0.000	0.011	0.0014	-0.0447	-0.0172
16	90A	-12.137	0.003	0.011	0.0014	-0.0444	-0.0168

Dynamic Susceptibility

Please refer to topic by this name in the Technical Reference Manual and also Annexure I of the Technical Reference Manual to understand the screen shots shown below.



#	Mode	Frequency (Hz)	Maxima Nodes Velocity	Maxima Nodes Stress	Susceptibility (psi / ips)
1	3	3.593	130	70	3486
2	5	4.014	100	128	662
3	1	0.774	220B	250	330
4	2	0.830	210	250	298
5	6	4.781	240	240	282
6	4	3.601	210	240	81

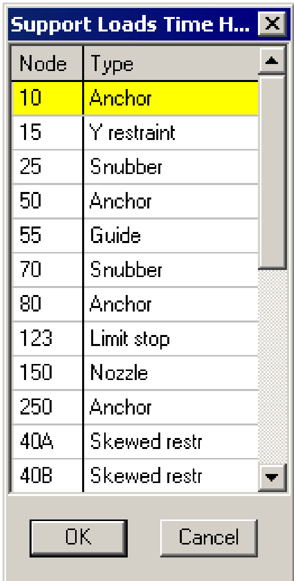


#	Susceptibility = 3486		
	Node	Displacement	Stress
1	10	0.0000E+00	1.6178E+02
2	15	1.6852E-03	5.6157E+01
3	20A	4.0171E-03	4.1127E+01
4	20B	5.7152E-04	1.7636E+01
5	25	4.5842E-03	4.3027E+02
6	30	4.7196E-03	2.6842E+02
7	35	2.6236E-03	1.3451E+02
8	40A	2.0893E-03	1.3440E+02
9	40B	3.2490E-03	2.9311E+01
10	50	0.0000E+00	3.4517E+02
11	55	6.3282E-03	2.3789E+02
12	60	7.1366E-03	2.6735E+03
13	70	1.4654E-02	2.2449E+04
14	80	0.0000E+00	2.6397E+03
15	90	5.4405E-03	1.2869E+02
16	100	1.2069E-01	1.6966E+02
17	110	1.0941E-01	3.9453E+02
18	120	9.7158E-02	1.4852E+03
19	122	8.2150E-02	1.7965E+03
20	123	6.5025E-02	1.7989E+03
21	124	4.5876E-02	1.5684E+03

Time History

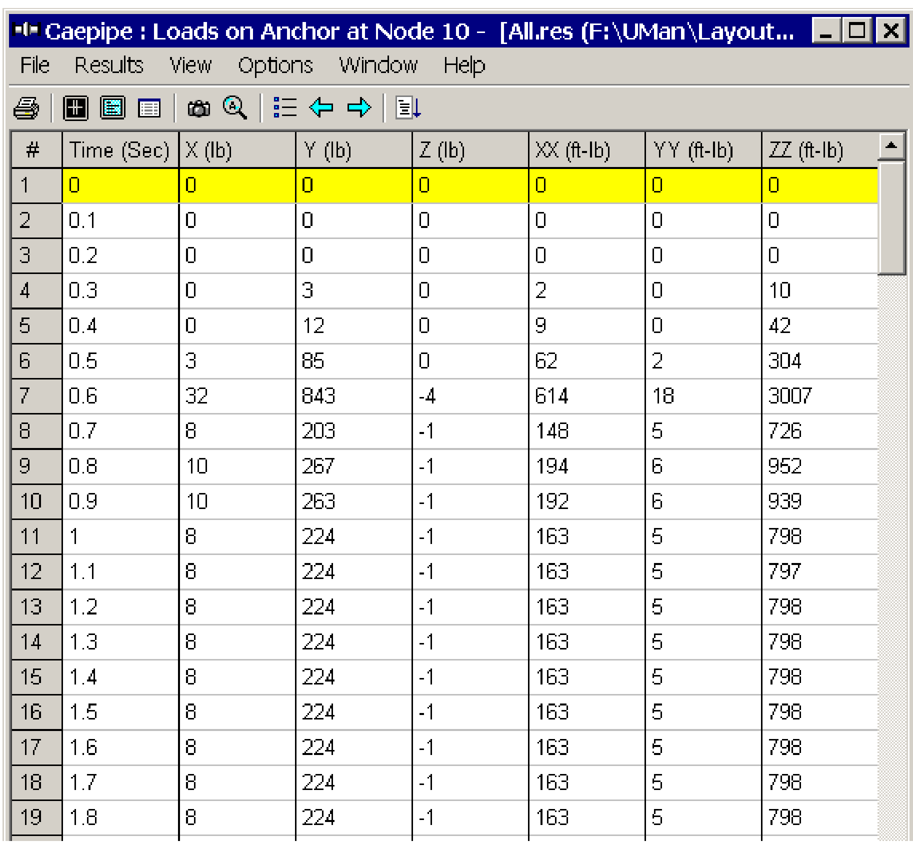
For time history results, you are shown the following dialog from which you need to select an item.

Then, you are shown a list of supports in the model from which you need to select one.



Node	Type
10	Anchor
15	Y restraint
25	Snubber
50	Anchor
55	Guide
70	Snubber
80	Anchor
123	Limit stop
150	Nozzle
250	Anchor
40A	Skewed restr
40B	Skewed restr

Once you select a support from the list, then you are shown the time history at that location.



#	Time (Sec)	X (lb)	Y (lb)	Z (lb)	XX (ft-lb)	YY (ft-lb)	ZZ (ft-lb)
1	0	0	0	0	0	0	0
2	0.1	0	0	0	0	0	0
3	0.2	0	0	0	0	0	0
4	0.3	0	3	0	2	0	10
5	0.4	0	12	0	9	0	42
6	0.5	3	85	0	62	2	304
7	0.6	32	843	-4	614	18	3007
8	0.7	8	203	-1	148	5	726
9	0.8	10	267	-1	194	6	952
10	0.9	10	263	-1	192	6	939
11	1	8	224	-1	163	5	798
12	1.1	8	224	-1	163	5	797
13	1.2	8	224	-1	163	5	798
14	1.3	8	224	-1	163	5	798
15	1.4	8	224	-1	163	5	798
16	1.5	8	224	-1	163	5	798
17	1.6	8	224	-1	163	5	798
18	1.7	8	224	-1	163	5	798
19	1.8	8	224	-1	163	5	798

While viewing these results, you can export time history results to a comma separated values (.csv) file that can be read by a spreadsheet program such as MS-Excel (see menu File > Export) for plotting, etc.

Hotkeys You Can Use

For keyboard operations, the following list of hotkeys (in addition to MS-Windows keys for open, print, exit, etc.) can make you more productive.

Tab	Next results item
(Shift) Tab	Previous results item
Right Arrow Key	Next load case or mode shape
Left Arrow Key	Previous load case or mode shape
Ctrl+Right Arrow Key	Next item of same type (e.g., next support)
Ctrl+Left Arrow Key	Previous item of same type (e.g., previous support)
Ctrl+F	Find node
Ctrl+L	List
Ctrl+U	Units
Ctrl+A	Graphics - Zoom all
F2	Move focus to Graphics window
F3	Move focus to Layout, List or Results window (wherever the focus was before)
F4	Graphics - Set Viewpoint
F5	Graphics - Previous view
F6	Show list of Other supports, forces, displacements, etc.
F7	Show forces, displacements in global/local coordinates
Ctrl+D	Show/Hide deflected shape*
Ctrl+Shift+A	Show/Hide animated deflected/mode shape*

* Works only while reviewing Displacement Results / Mode Shapes / Frequencies.

Menus and Toolbar

These items are explained in detail under Menus. A summary is given here.

File menu

This menu contains commands for file operations including starting a new model, opening a Results file, printing, etc.

Results menu

This is one of the menus that changes with the displayed result. At all times, it contains commands for

- Display list of Results
- Move forward to Next Result (Tab key)
- Move backward to Previous Result (Shift+Tab key)

An example of the additions to the Results menu:

When Support Loads are displayed, the menu displays, in addition to the above, the following commands:

- Other supports
- Next Support
- Previous Support
- Load cases
- Next Load case
- Previous Load case

All of the above commands also appear on the (context-sensitive) toolbar.

View menu

Commands for graphics operations are available from here. This menu, like the Results menu, changes with displayed results.

An example of the changes:

When Sorted Stresses are displayed, the menu displays, in addition to the above, the following commands:

- Show stresses in the Graphics window
- Show stress ratios in the Graphics window
- Set stress or stress ratio thresholds (to display stresses or stress ratios that are above a specific threshold value).

You do not have to move focus to the Graphics window to execute these commands.

Options menu

You can change units or font from the options menu. Any change here in units or font will affect all text windows. But the change in units is not saved when you close the Results window (they will be saved only when you change units in input mode and save the model).

Window menu

You can move focus to other windows. Use F2 to move between text and Graphics windows, and F3 to move focus between open text windows. This is helpful to those who work with maximized windows (perhaps, because of a lower monitor resolution).

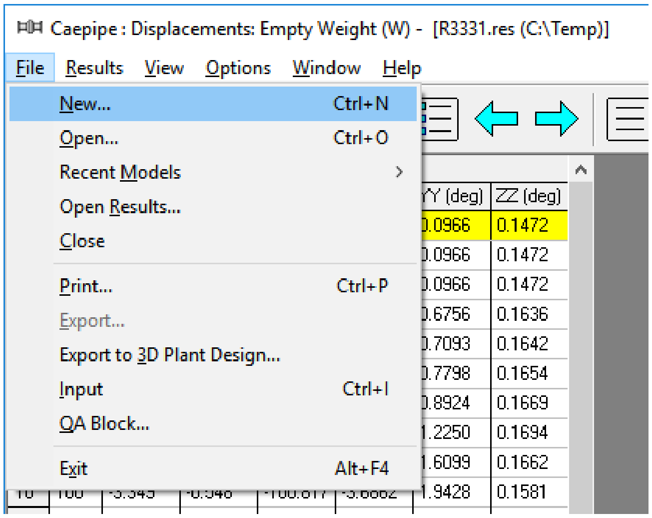
Open Other Windows

You can open the remaining text windows from here - List and Layout - to have all four windows open at the same time to enhance your understanding of the model results. Use the Window menu to open the other windows. They are displayed in non-editable mode.

Help menu

For on-line help and information pertaining to the remaining period of your yearly Maintenance, Enhancement and Support (ME&S) agreement with SST.

File Menu



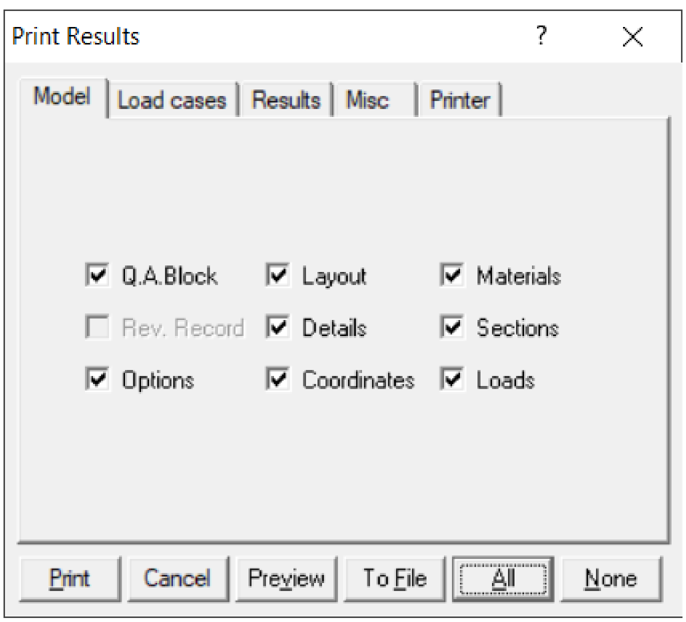
The File menu here is similar to the File menu in the Layout window. You can open or close any CAEPIPE model, material, spectrum library or CAEPIPE results file from here.

.Print (Ctrl+P).

This is a comprehensive command that lets you print all model input and computed results. This dialog has five tabs and six buttons.

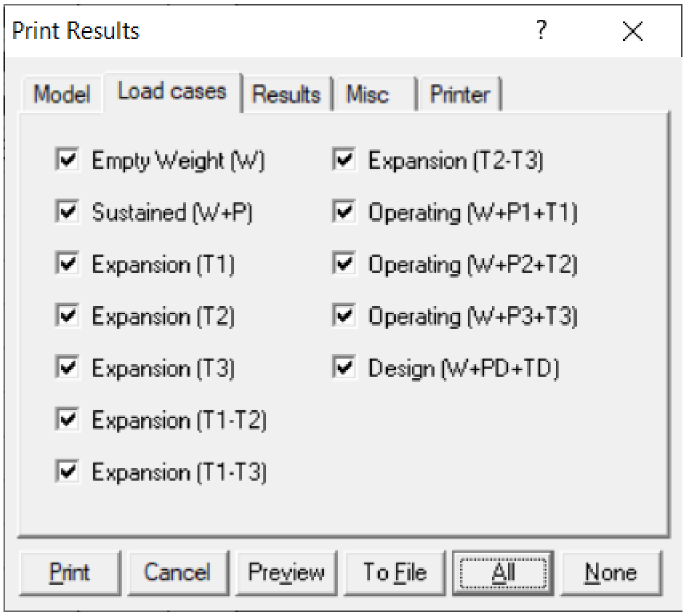
Model

On this tab, check those input data items that you would like to print.



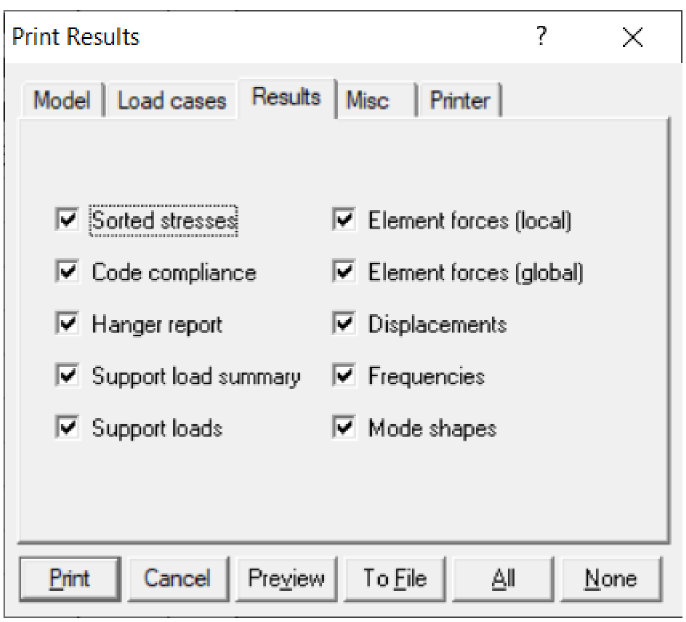
Load cases

On this tab, check the different load cases for which you want to print results.



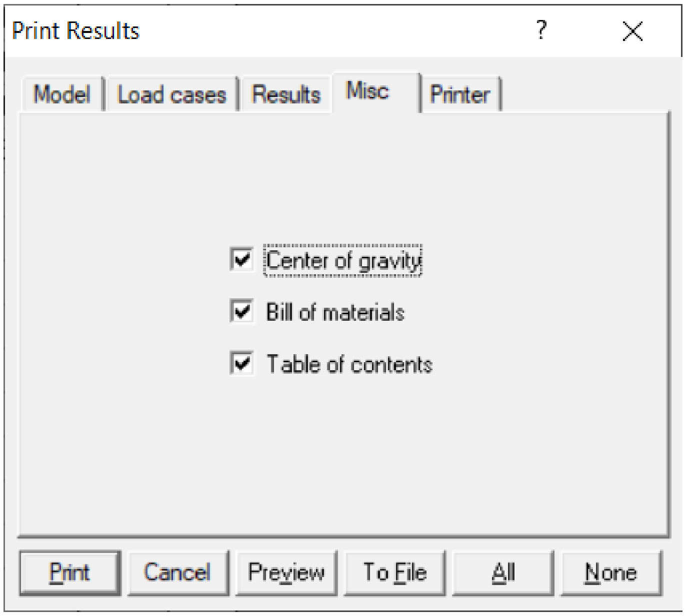
Results

On this tab, check the different results items that you want to print.



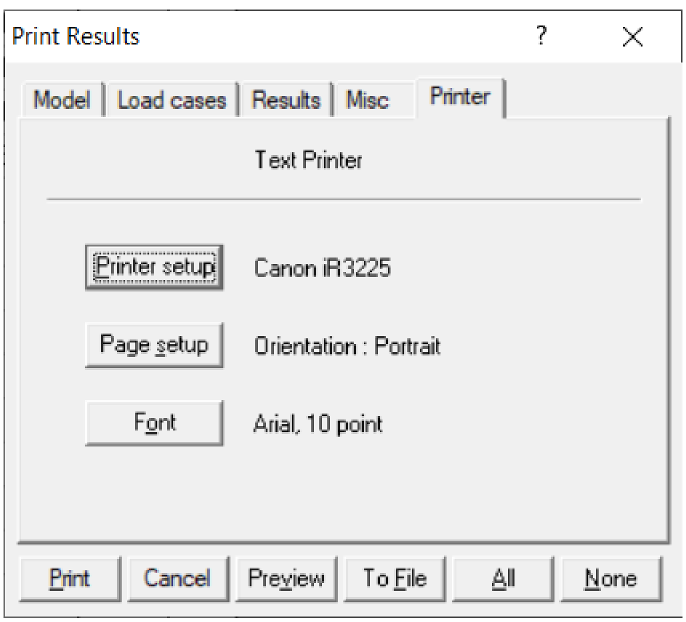
Misc

On this tab, select the items of interest that you want to print.



Printer

On this tab, you can customize your printer settings such as selecting a different printer, setting different margins, and the print font. By selecting a custom Adobe Acrobat-compatible printer (if available on your system), you can generate an Adobe PDF (Portable Document Format) file that contains the whole model input and output.



Print Dialog Buttons

Print

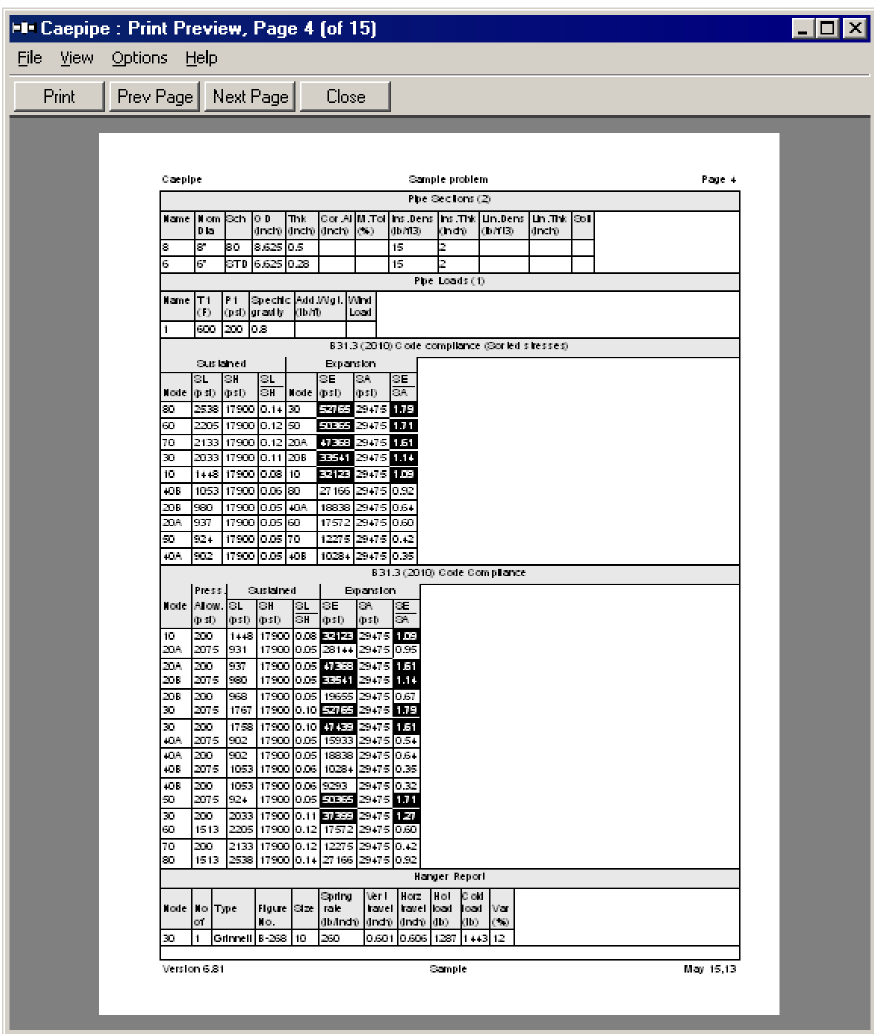
After selecting the items you want to print, click on Print button to send output to the printer selected under the Printer tab.

Cancel

You can click on Cancel to exit the dialog without printing.

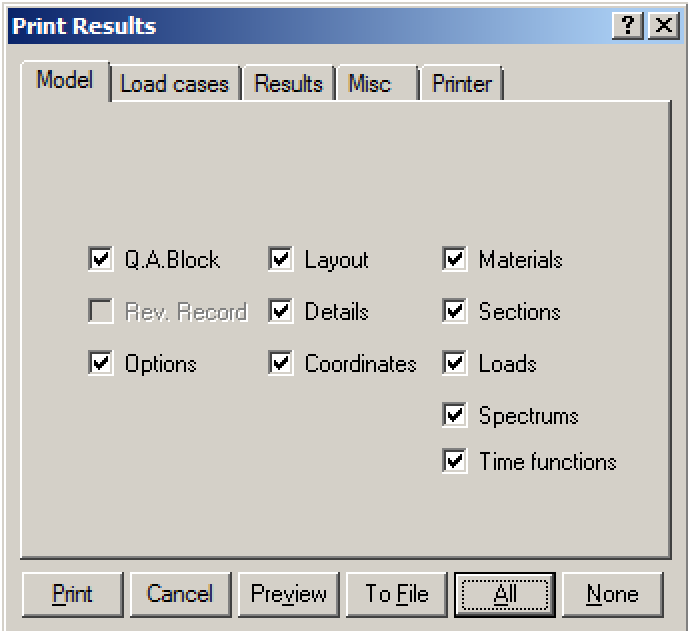
Preview

Before printing, you can preview how the printed output will appear by clicking on Preview. If necessary, you can adjust the printer settings (under the Printer tab) to change the print font and margins, and preview again.

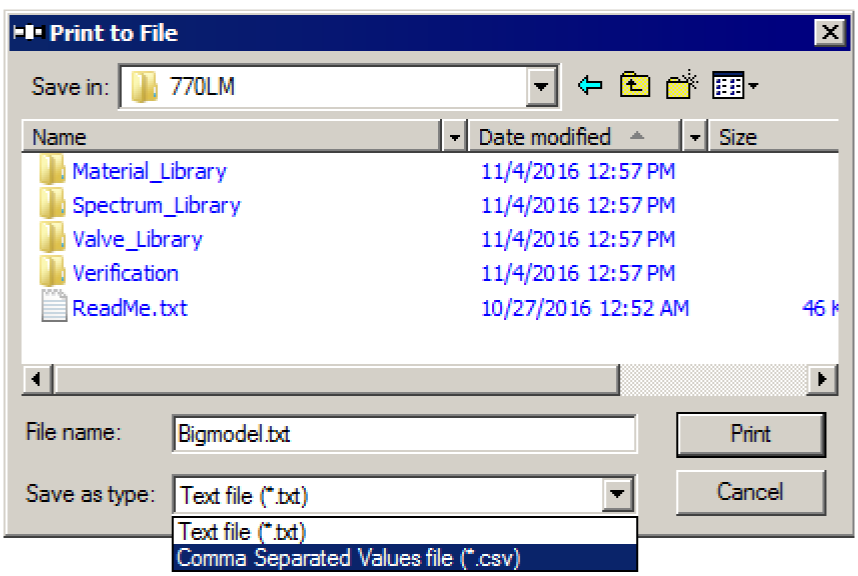


To File

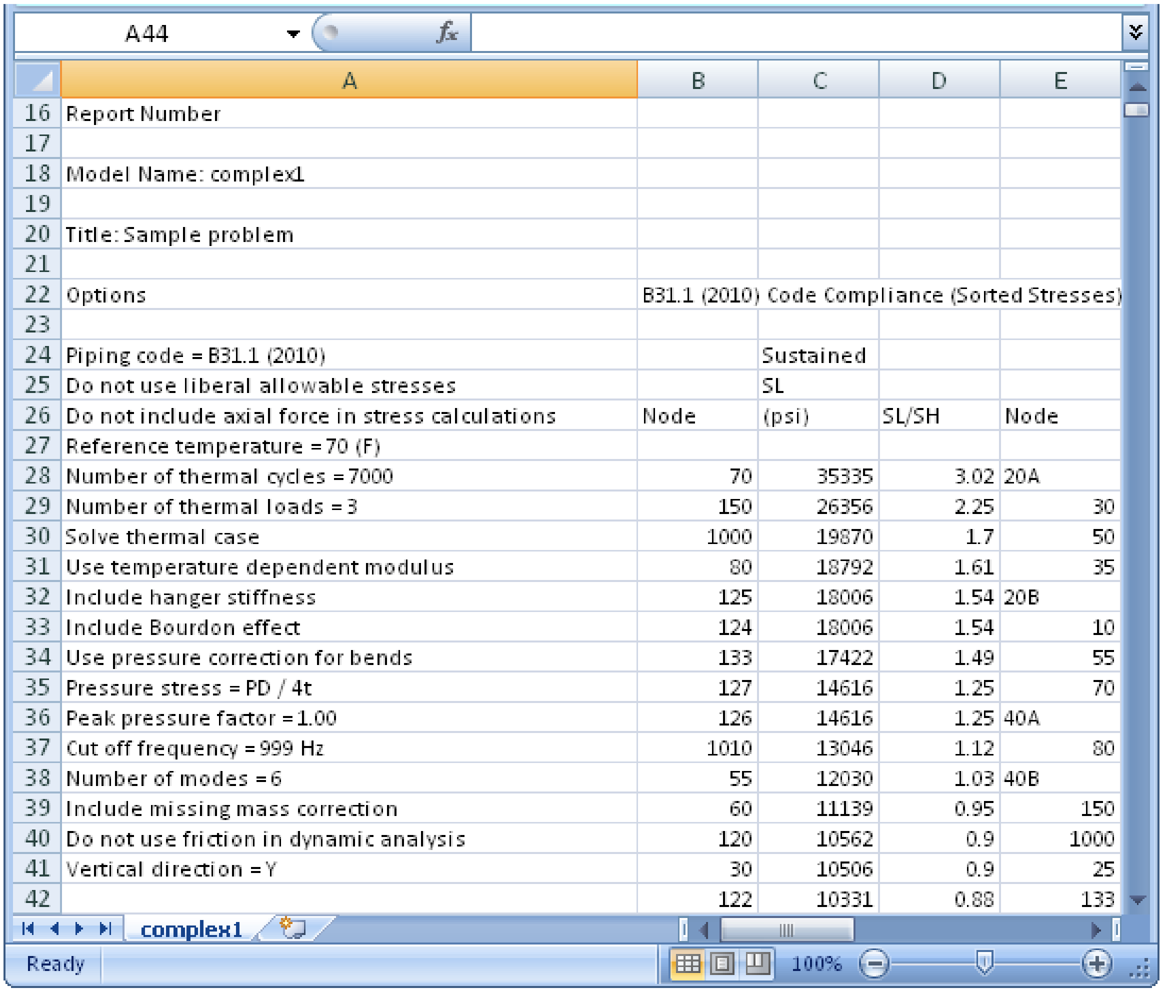
If you want to send all of your selected items (input and results) click on the To File button.



You can export the data to a .txt file or to a .csv file.



Here's an example of what the .csv file looks like with the exported data (reading from a spreadsheet software).



	A	B	C	D	E
16	Report Number				
17					
18	Model Name: complex1				
19					
20	Title: Sample problem				
21					
22	Options	B31.1 (2010) Code Compliance (Sorted Stresses)			
23					
24	Piping code = B31.1 (2010)		Sustained		
25	Do not use liberal allowable stresses		SL		
26	Do not include axial force in stress calculations	Node	(psi)	SL/SH	Node
27	Reference temperature = 70 (F)				
28	Number of thermal cycles = 7000	70	35335	3.02	20A
29	Number of thermal loads = 3	150	26356	2.25	30
30	Solve thermal case	1000	19870	1.7	50
31	Use temperature dependent modulus	80	18792	1.61	35
32	Include hanger stiffness	125	18006	1.54	20B
33	Include Bourdon effect	124	18006	1.54	10
34	Use pressure correction for bends	133	17422	1.49	55
35	Pressure stress = PD / 4t	127	14616	1.25	70
36	Peak pressure factor = 1.00	126	14616	1.25	40A
37	Cut off frequency = 999 Hz	1010	13046	1.12	80
38	Number of modes = 6	55	12030	1.03	40B
39	Include missing mass correction	60	11139	0.95	150
40	Do not use friction in dynamic analysis	120	10562	0.9	1000
41	Vertical direction = Y	30	10506	0.9	25
42		122	10331	0.88	133

All

To select all items, click on the All button.

None

To deselect all items, click on the None button.

.Export.

This command lets you export Time History results to .CSV file and Hanger Report to LICAD software using an ASCII file with extension (.cli).

.Export to 3D Plant Design.

This command lets you export deformed geometry to 3D Plant Design software E3D/PDMS and CADMATIC while being in Displacements results.

In order to comply with space constraints, piping designers check for interferences between piping and the adjacent structures, concrete buildings, equipment and reserve volumes for walkthrough, maintenance space etc. either manually on General Arrangement drawings or nowadays in a 3D Plant model. Unfortunately, such "Clash-check" is performed using only "as-designed" drawings/3D model. In reality, piping systems deform due to its own dead-weight during installed/shut-down condition, during hydrotest as well as during operating condition. Clash-checks for piping systems under such "deformed conditions" are usually never performed till date. This option transfers "deformed shapes" for the piping system for different loading conditions to the 3D Plant Design software.

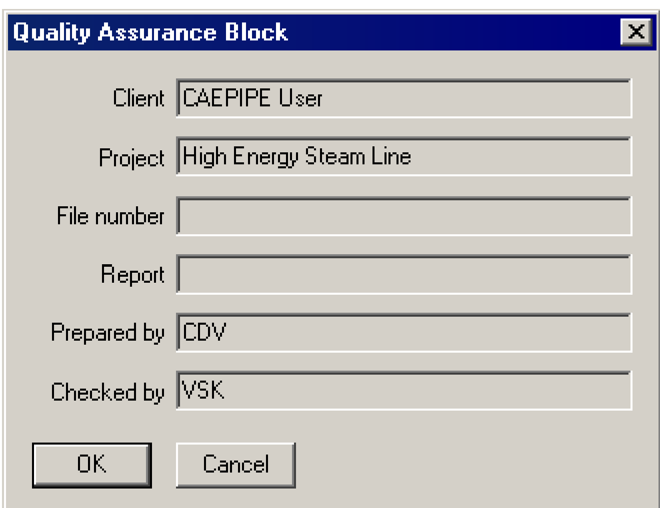
The 3D Plant Design software, in turn, could then check for interference under different loading conditions as well as under the traditional "as-designed" model condition by exporting the "deformed layouts for loading conditions of interest"through the option Results Window > File > Export 3D Plant Design.

.Input.

This useful command allows you to go to the model Layout window quickly without having to go through a model File > Open… command.

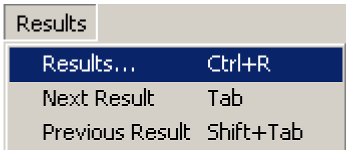
.QA Block.

This command shows you the QA information you input in the Layout window. As the shown information is not editable here, to make changes to it, you need to go back to the Layout window and use the same command under the File menu. After you make edits, you need to save the model and reanalyze.



Results Menu

.Results Navigation

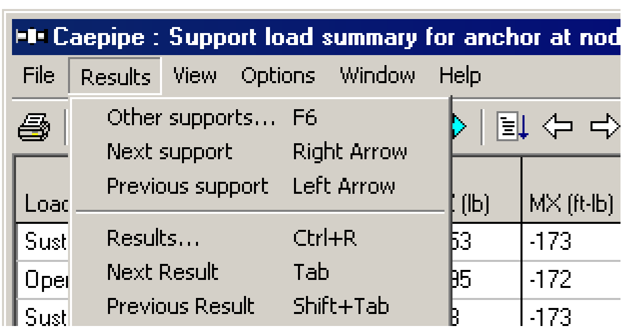


You can open the list of all results computed by selecting Results command (Ctrl+R). Press Tab or Shift+Tab to move forward or backward through the different results items, each of which will be shown in the same Results window.

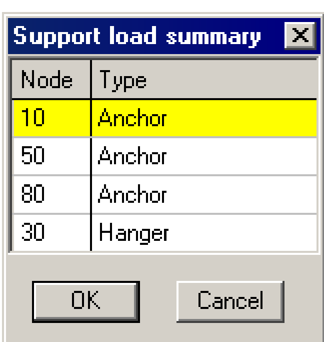
This menu may change with the displayed results item. For example, the menu shows like in the above figure for Sorted Stresses, Code Compliance and Hanger Report.

You can open "Support load summary" results by selecting Results command (Ctrl+R) and then the radio button "Support load summary".

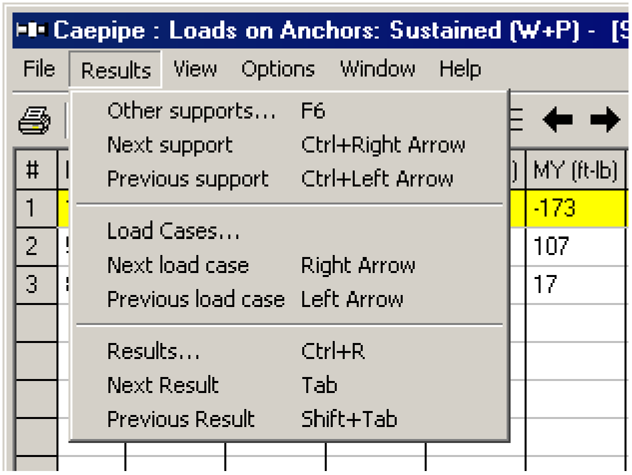
For Support load summary, the menu shows similar to the following figure. The first set of commands is for showing the support load summary for other supports, one at a time. The second set of commands is as before.



Pressing F6 key will display the window for "Other supports" as shown below.



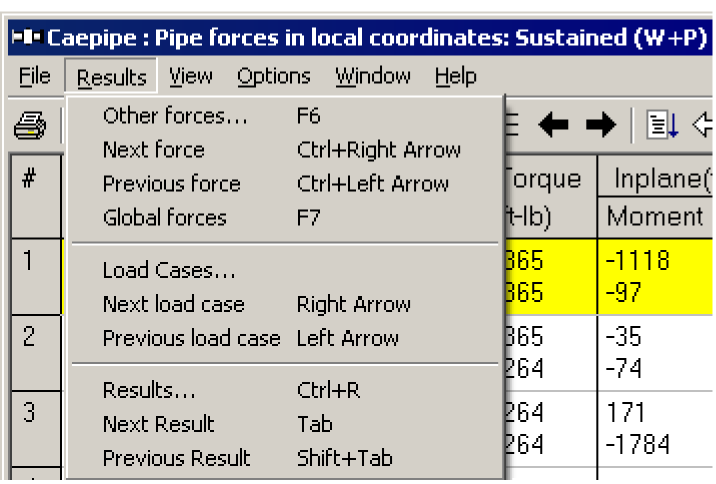
For Support Loads, the menu changes to show related commands for the new results item.



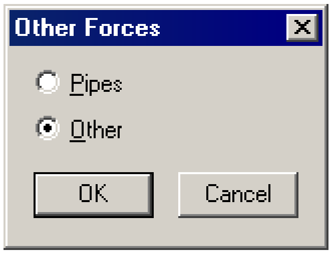
The first set of commands is for showing the support loads for other supports (hangers, limit stops, etc.), one at a time (you can also click on the white right or left arrow).

The second set of commands (Load cases, etc.) is for changing the load case for the displayed support. For example, the title bar of the figure shows Loads on Anchors: Sustained. You can change the load case to Expansion load case for this support by selecting the Next load case command (you can also click on the black right or left arrow). The last set of commands is as before, moving through different Results screens.

For Element forces, the menu changes to show related commands.

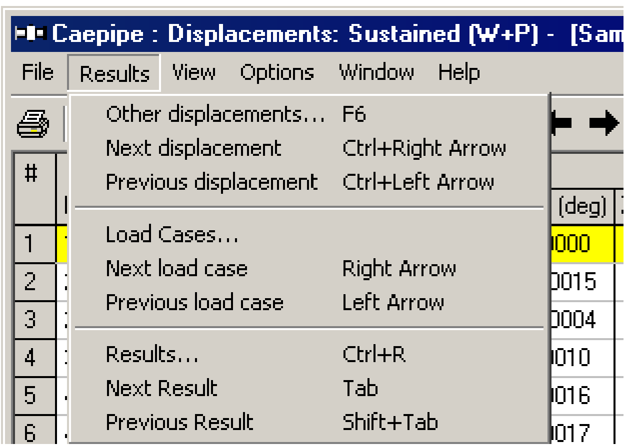


The first set of commands is for showing other forces (at other elements such as expansion joints and valves), one at a time (you can also click on the white right or left arrow).



The second set of commands (Load cases, etc.) is for changing the load case for the displayed Element forces. For example, if the title bar of the figure showed Sustained load case, then by selecting the Next Load case command, you can display forces for the Expansion load case (you can also click on the black right or left arrow). The last set of commands is as before, moving through different Results screens.

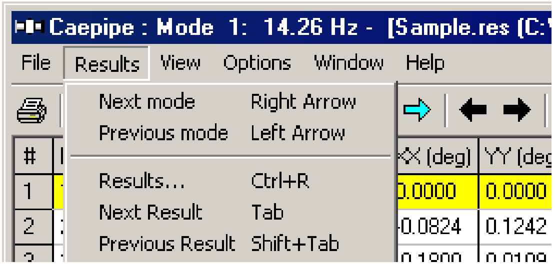
For Displacements, the menu changes to show related commands as follows:



The first set of commands is for showing other displacements (such as minimum and maximum displacements and displacements at Hangers, Guides, Limit Stops, etc.).You can also click on the white right or left arrow.

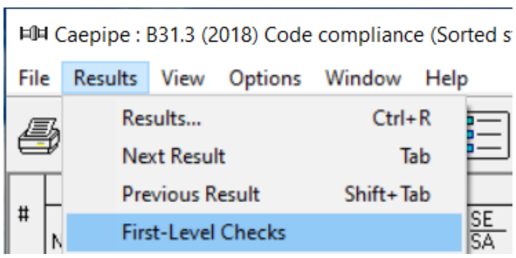
The second set of commands (Load cases, etc.) is for changing the load case for the displayed displacements. For example, if the title bar of the figure showed Sustained load case, then you can show displacements for the Expansion load case by selecting the Next Load case command. (You can also click on the black right or left arrow). The last set of commands is as before.

For Mode Shapes, the menu changes to show two commands specific to the new results.

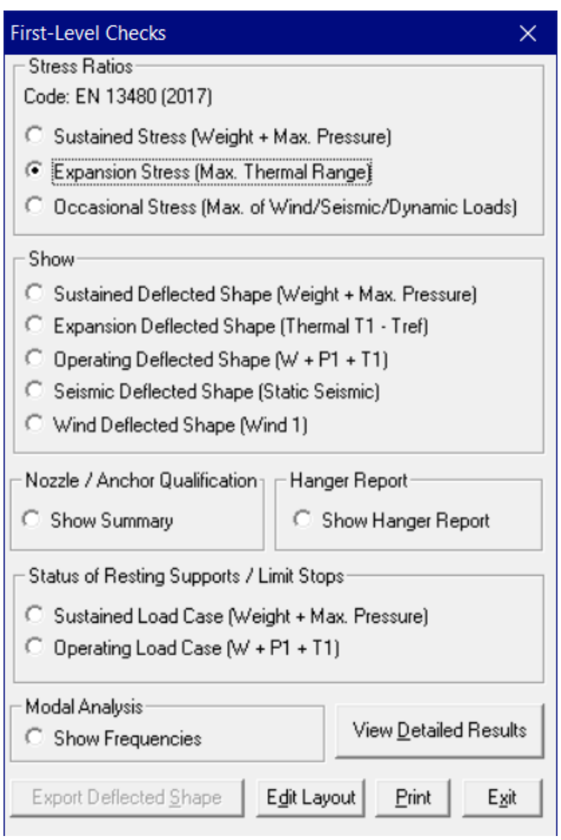


They are to move forward or backward through the available mode shape details. The last set of commands is as before.

So, depending on the displayed results item, the Results menu (Ctrl+R) will show different menus with at least one common set of commands for moving forward or backward through the different results.



.First-Level Checks.



This menu is available only when the load cases "Sustained (W+P)" and "Expansion (T1)" are turned ON for all piping codes excepting the "NONE" code. For NONE code, the load case "Static" should be turned ON.

Pipe stress engineers normally carry out a number of analysis iterations on each stress model to arrive at a suitable support scheme that would meet all required pipe stress criteria. When performing such analysis iterations, they would like to review only key results (summarized under one command) at the end of each analysis. Based on such key results, they decide their next course of action on each stress model. The menu "First-level Checks" provides an easy way for stress engineers to review key results for their analyses.

Key results that can be reviewed using this menu are as follows.

Contour plots of stress ratios for Sustained, Expansion and Occasional stresses

Deflected shapes for Sustained, Thermal, Operating, Seismic (static) and Wind loads

Support Load Summary for Nozzles and Anchors

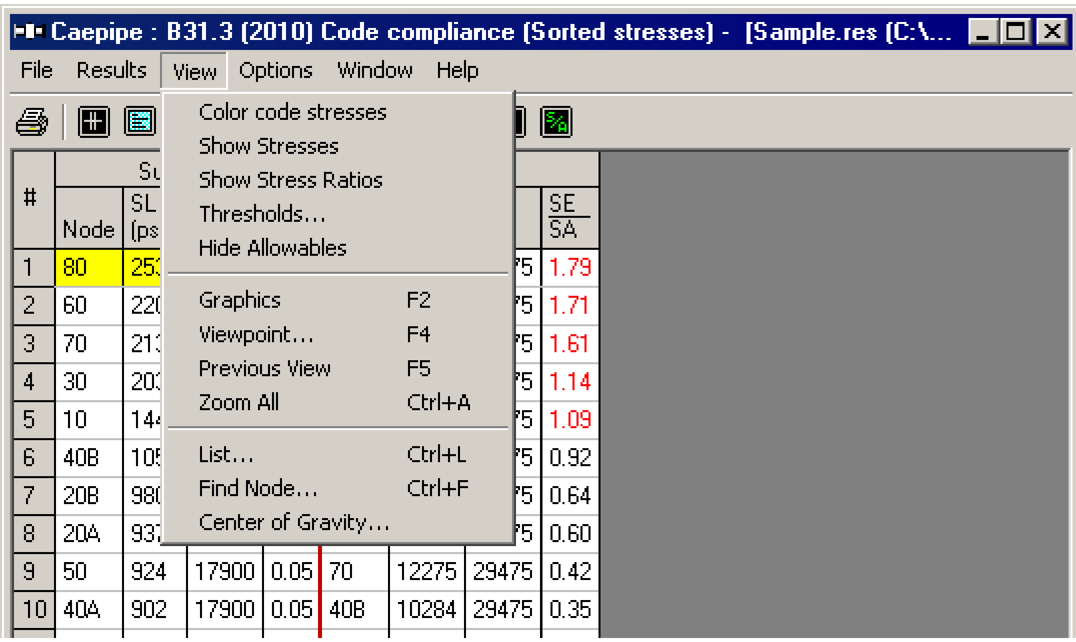
Hanger Report

Status of Piping at Resting Supports for Sustained and Operating loads (resting or lifted-off)

Frequencies and Mode Shapes

When piping designers use CAEPIPE 3D+ along with 3D Plant Design systems, they can use this menu "First-level Checks" to review key results that would guide them to arrive at piping layouts that would be flexible enough to absorb thermal expansion/contraction.

View Menu



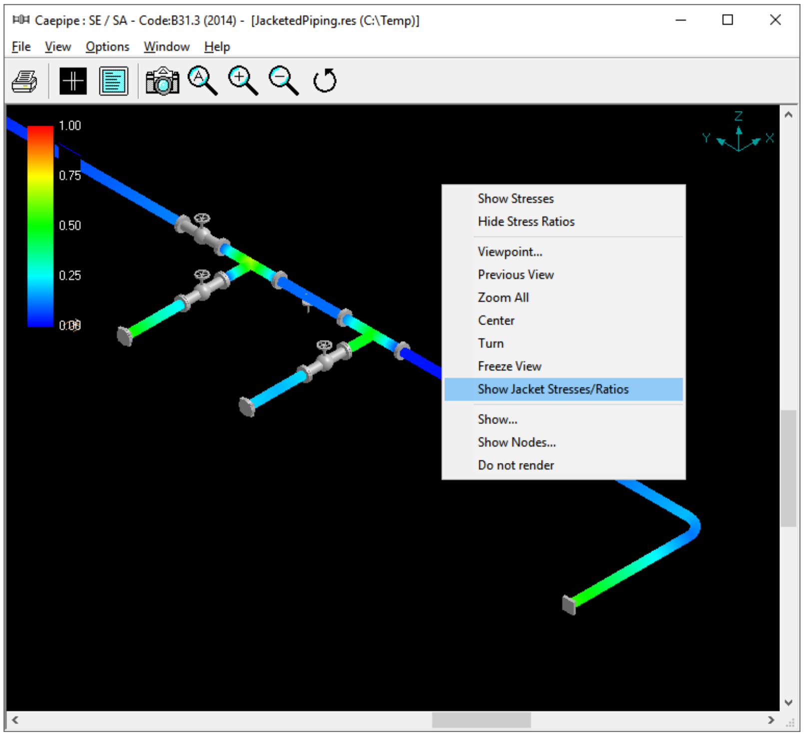
This menu contains commands mostly forgraphics operations. The List and Find Node commands function like they do in Layout window > View menu. Center of Gravity command is covered later in this section. As with the Results menu, this menu too changes its offerings depending on the results item being viewed.

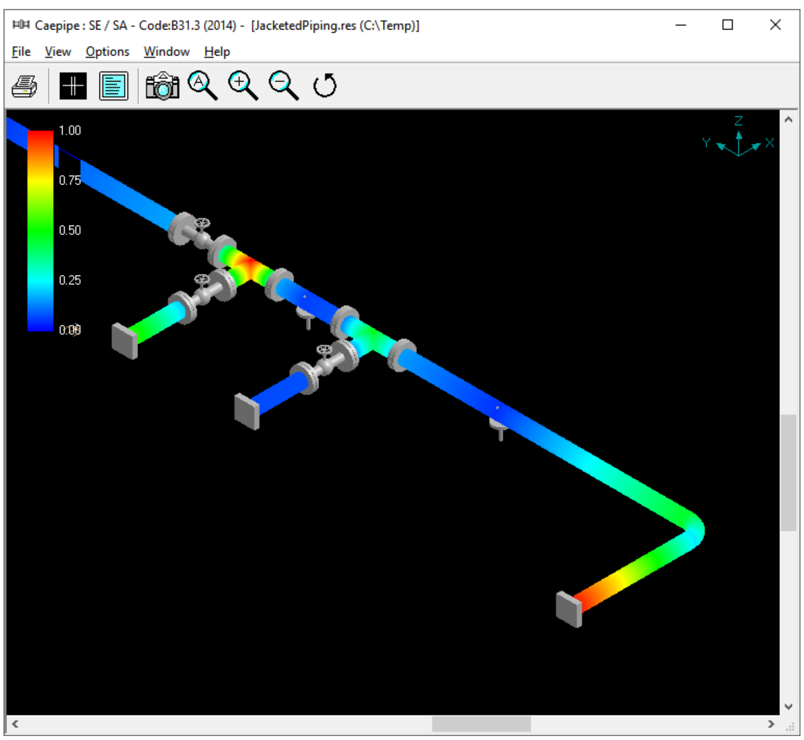
The first menu shown here is available from the Sorted Stresses results. The first set of commands helps you view stresses graphically in the Graphics window.

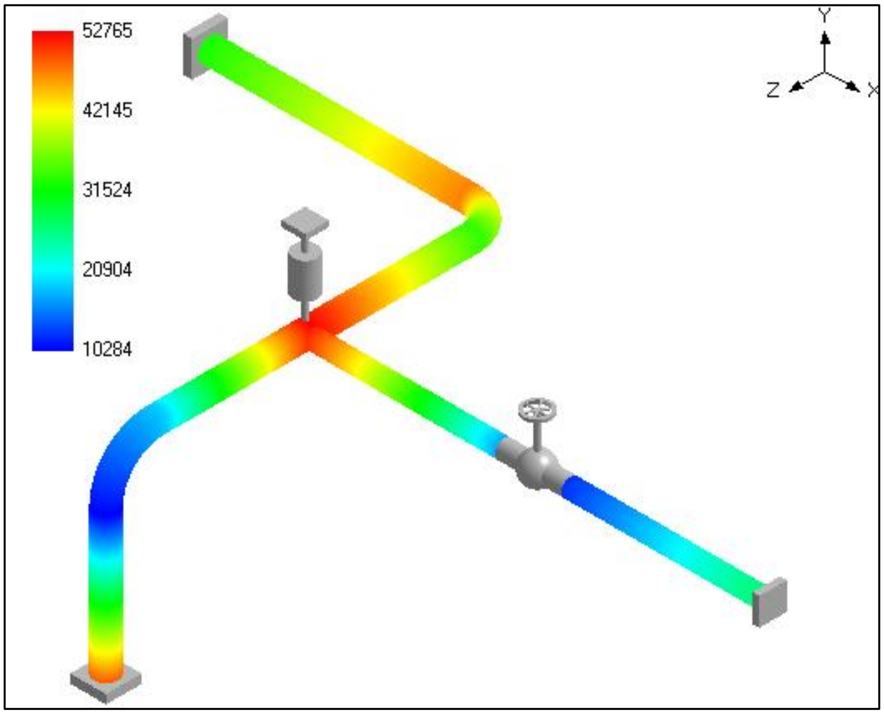
.Color code stresses./ .No color coding.

Displays or removes the color coded stress (or stress ratio) contour in the Graphics window. To display a color coded contour, place the yellow highlight on the type of stress you want to see graphically, for example, on Expansion stress. Then, select Color code stresses. Next, click on Show stresses or stress ratios button on the toolbar. The Graphics window will show the color coded contour plot.

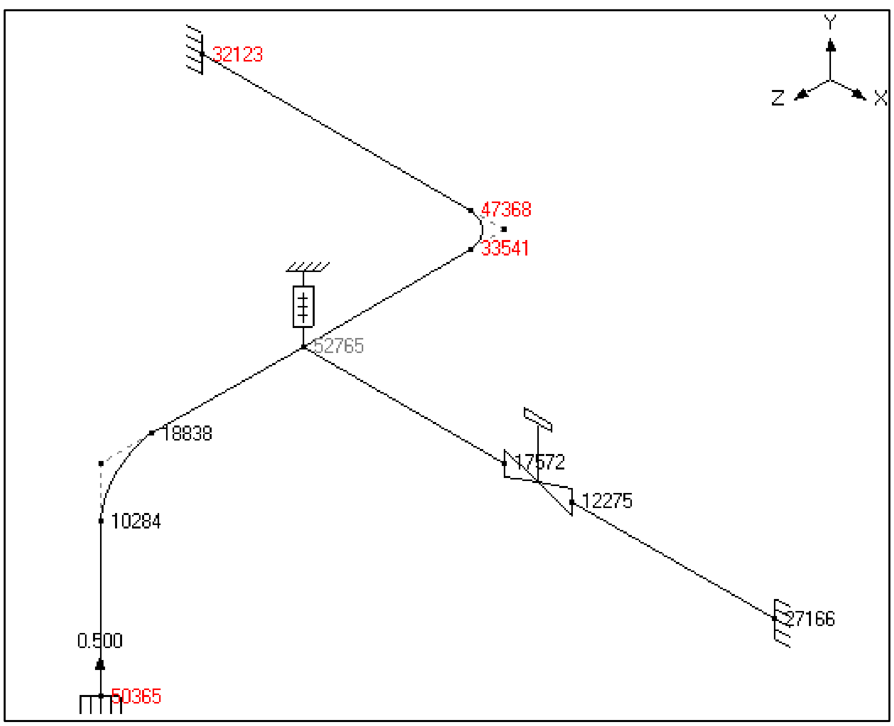
For Jacketed piping, the color coded contour plot shown is for the core pipe. You can also view the stress contour plot for the jacket piping using the command "Show Jacket stresses /Ratios" as shown in the graphics below.


Caepipe : SE / SA - Code:B31.3 (2014) - [JacketedPiping.res (C:\Temp)]
File View Options Window Help
1.00
0.75
0.50
0.25
0.00
Show Stresses
Hide Stress Ratios
Viewpoint...
Previous View
Zoom All
Center
Turn
Freeze View
Show Jacket Stresses/Ratios
Show...
Show Nodes...
Do not render


Caepipe : SE / SA - Code:B31.3 (2014) - [JacketedPiping.res (C:\Temp)]
File View Options Window Help
1.00
0.75
0.50
0.25
0.00



When you select No color coding, numeric values for stresses or stress ratios are shown (see next image).



.Thresholds.

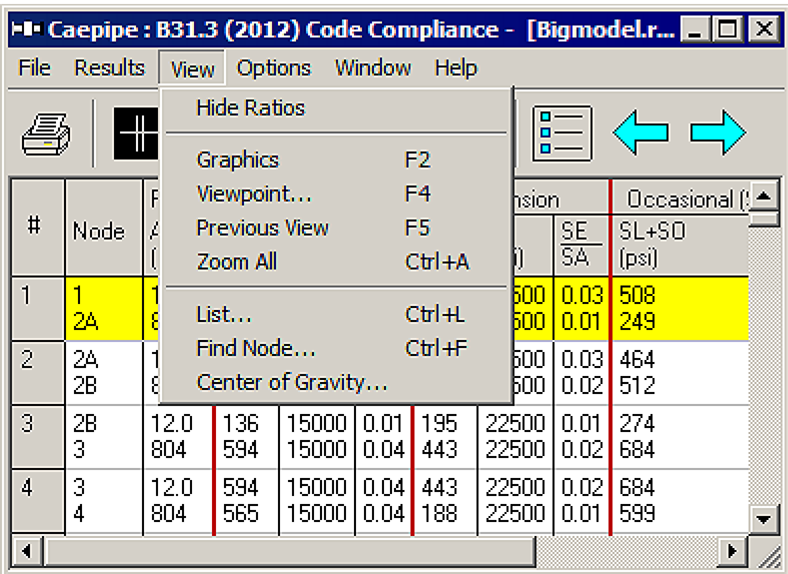
Use this command to display stresses (or stress ratios) over a specified stress or stress ratio value.

.Hide Allowables.

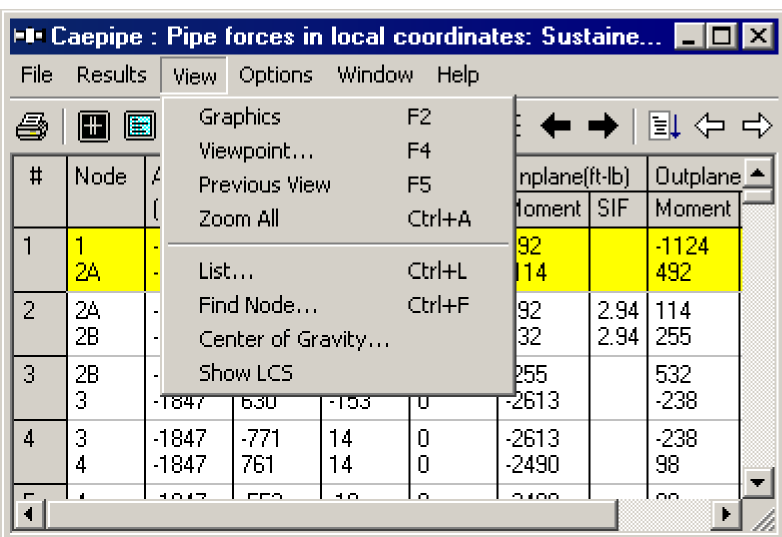
Selecting this command will remove the Allowables columns from the Results display. For some codes that have too many columns, this command helps by lessening the number of columns displayed.

All commands (except Hide Ratios and Center of Gravity) function like they do under the Layout window > View menu.

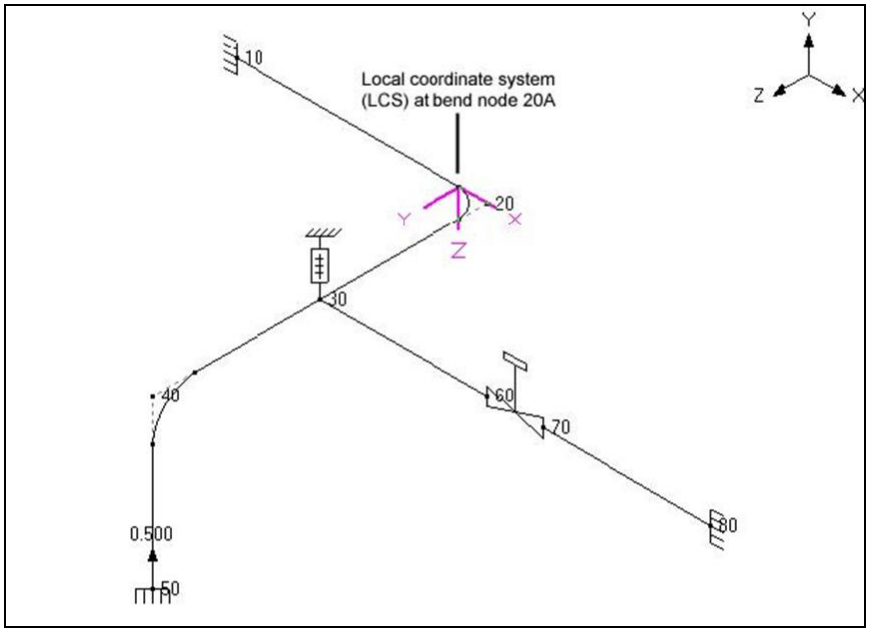
Under the Code Compliance results display, the View menu shows only one command to Hide (Stress) Ratios (such as SL/SH, SE/SA). The remaining commands are the same as before.



For the Pipe element (or Other element) forces display, the following View menu is shown.

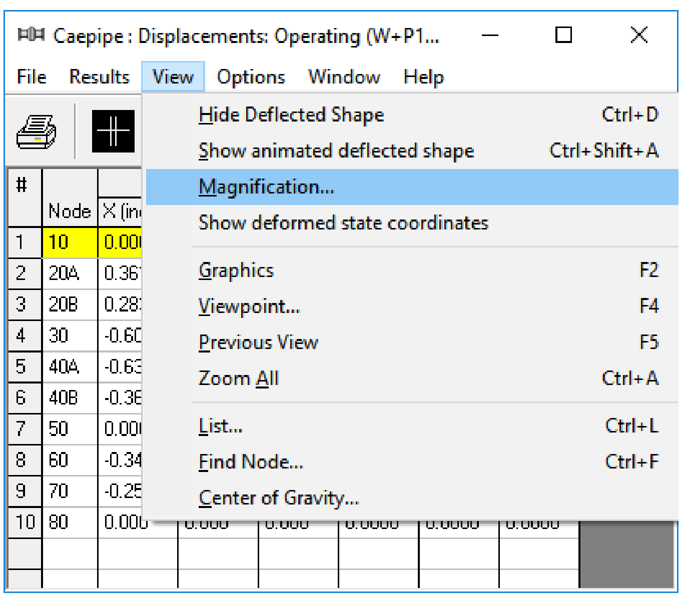


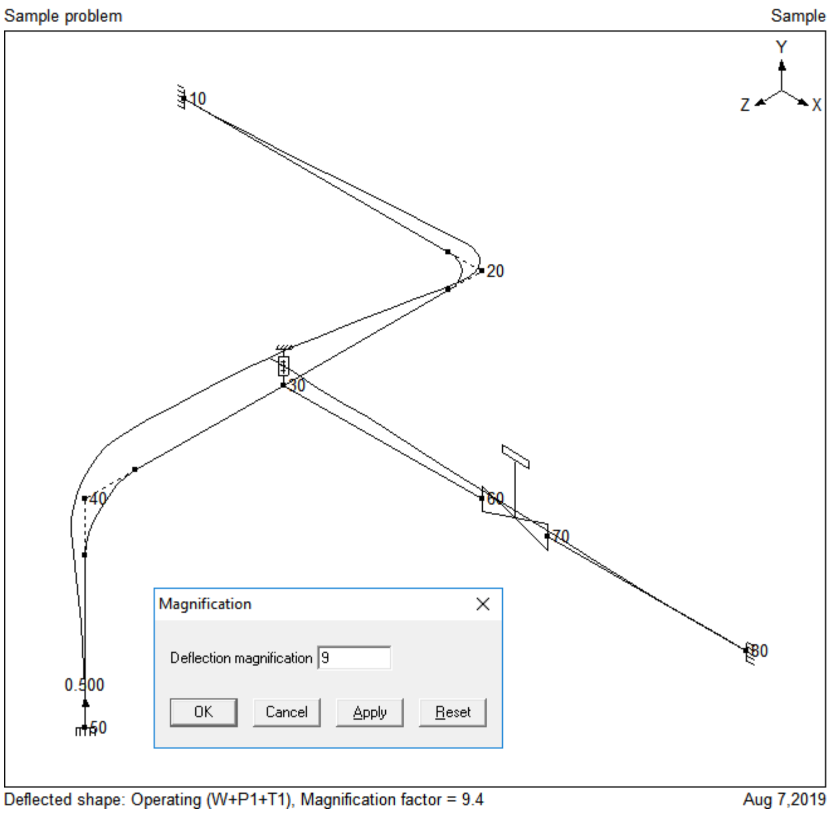
The only item new here is a command to Show LCS (Local Coordinate system)/Hide LCS in the Graphics window for each element. As you keep scrolling through the different elements in the results display, the corresponding element in the Graphics window will show a small coordinate symbol that depicts the element's LCS. The remaining commands are the same as before.



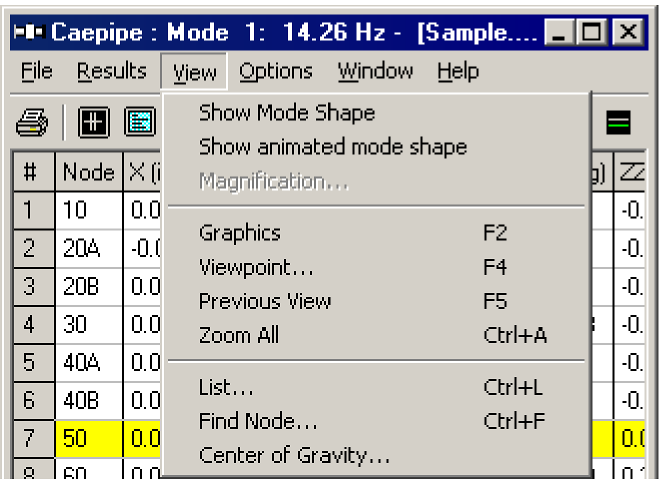
For Displacements results display, two commands to show and animate the deflected shape are shown. The remaining commands are the same as before.

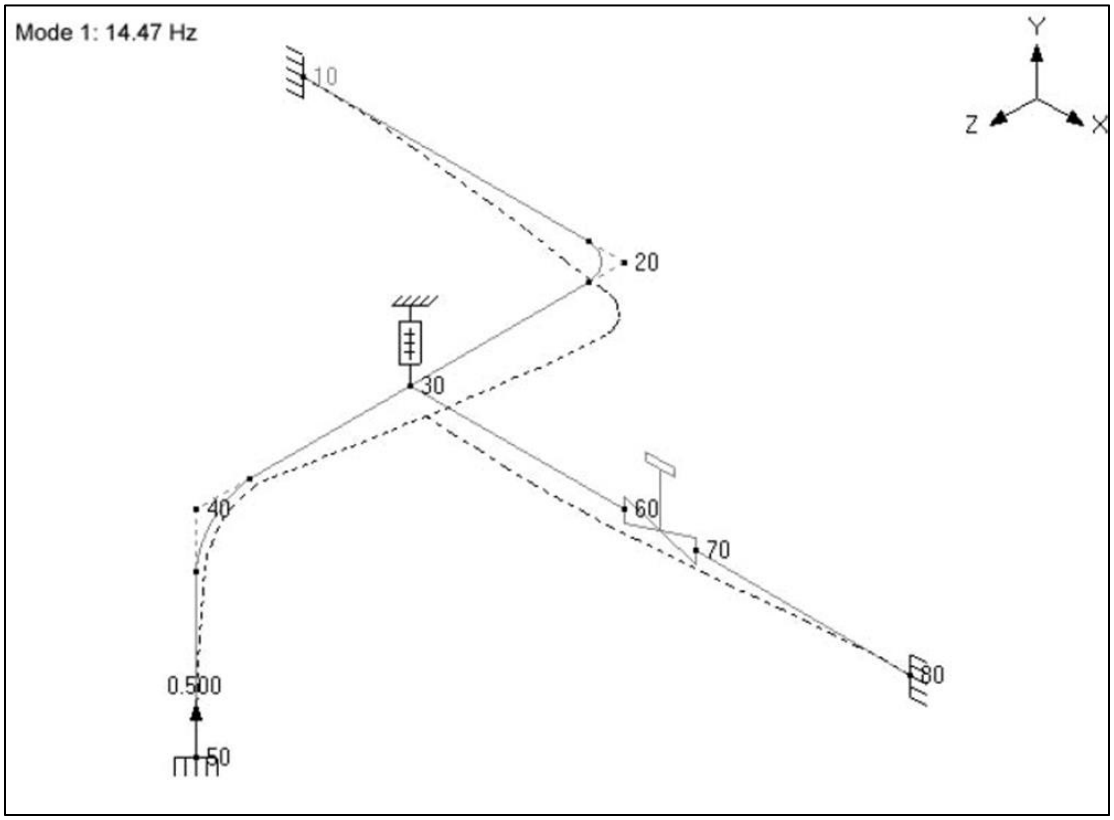
The deflected shape shown in CAEPIPE graphics is "Magnified" with a scale factor greater than 1.0. For the Deflected shape shown below for operating load case, CAEPIPE has magnified the deflected shape by a factor of 9. This magnification factor can be modified through Results Windows > View > Magnification… while being in deflected shape plot in CAEPIPE as shown below.





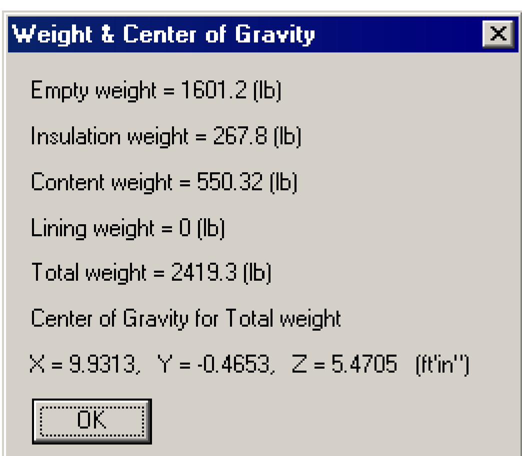
For Frequencies and Mode Shapes results display, two commands to show and animate the mode shapes are shown. The remaining commands are the same as before.





.Center of Gravity.

This command displays the center of gravity and total weight of the piping system as modeled. This information can be useful when you are comparing the weight of this model with total weight obtained from another source.



The Total weight is the sum of empty, insulation, content and lining weights.

This page is blank

Appendix A

Import / Export

Environmental Variable for Import / Export

It is possible to export model to MBF file with length units in "Inches" and moment units in "lb-in". This can be done by setting the following.

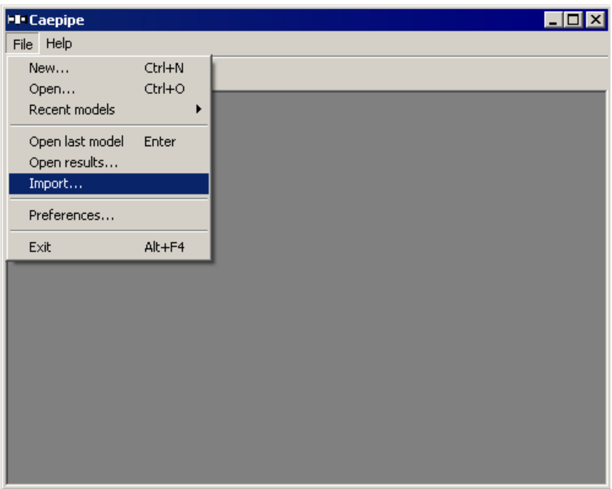
1. Length unit is defined in inches in .mod file and
2. An environmental variable with name "EXPORT_INCHES" with value is set to "YES".

This option is introduced to maintain the accuracy of the model by avoiding rounding-off issues while exporting and importing via MBF.

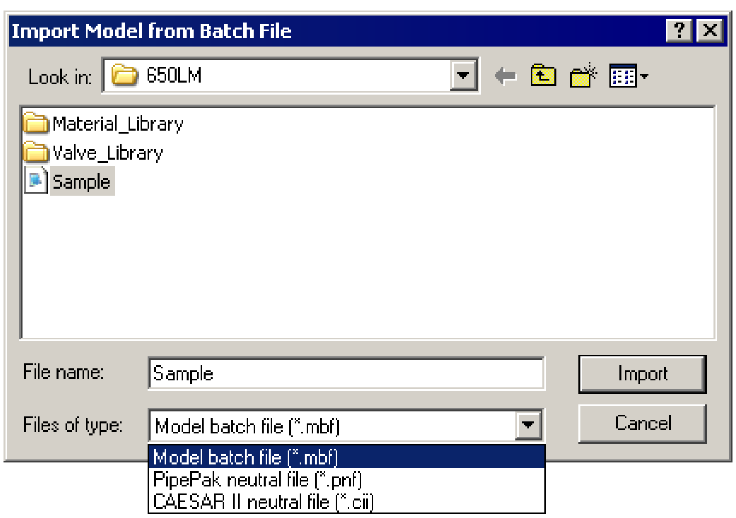
MBF Import

CAEPIPE can import model data from a text file, which may be created using a text editor and should have the extension: .mbf (model batch file). This text file may also be created for an existing model using the Export command from the Layout window.

To import a model, select the menu command File > Import from the Main window.



The Import Model dialog is shown.



Select the model batch file (.mbf) and then click on the Import button. The batch file will be read and the corresponding model file (.mod) will be created and then shown in the Layout window which can be further modified or analyzed.

For detailed information on the .pnf and .cii file import capabilities please contact us at support@sstusa.com.

Command line operation

Dragging and dropping an .mbf file on CAEPIPE.exe will import, analyze and produce the results in .csv format and close the program. On the other hand, dragging and dropping the .mod file on CAEPIPE.exe will open the .mod file. Using the command line, you may have CAEPIPE analyze and produce a CSV results file thus:

C:\CAEPIPE> caepipe.exe sample.mbf (Enter)

The above command will produce a "sample.csv" file in the same folder as the .mbf file.

Format of .MBF

The input data is given in the following order. Except for the heading the start of each section is indicated by a keyword. The data for that section follow. Only the first three characters of the keyword are significant. See an example MBF at the end of this appendix.

Note:

CAEPIPE input for Concentrated Mass, Ball Joint, Rigid Element, Valve, Bellows, Slip Joint etc. require the "Weight" of that item to be input in lbf, kgf etc. and NOT its mass. So, whenever mass is required for a calculation as in the case of forming Mass matrix for dynamic analysis, or in calculating inertia force as (mass x acceleration) for static seismic analysis, CAEPIPE internally computes the mass for each item to be equal to (weight / g-value).

Similarly, for density of pipe, insulation and lining materials, CAEPIPE requires "Weight Density" to be input in lbf/in3, kgf/m3 etc. and NOT its mass density.

Keyword	Data
	Heading
OPTIONS	Program options
MATERIAL	Material properties
SOILS	Soil properties
PIPE	Section properties
LOADS	Loads data
BMATERIALS	Beam Materials
BSECTIONS	Beam section properties
BLOADS	Beam loads
SPCMS	Spectrums (available in CAEPIPE Version 10.50 or later)
SPLVL	Spectrum Levels (available in CAEPIPE Version 10.50 or later)
LAYOUT	Layout data
PUMPS	Pump data
COMPRESSORS	Compressor data
TURBINES	Turbine data
SEISMIC	Seismic load

WIND	Wind load
ALLOWABLES	User allowable loads

Heading

Any number of heading lines may be given. Only the first will be used as Title in the CAEPIPE model.

OPTIONS

Hanger design

HGRA	Grinnell **(Default)**
HGRB	Bergen-Paterson
HGRC	Fee and Mason
HGRD	Basic Engineers
HGRE	Power Piping
HGRF	Nordon
HGRG	Carpenter and Paterson
HGRH	NPS
HGRI	PTP
HGRJ	Corner and Lada
HGRK	Elcen

Piping Code

B311	ASME B31.1
B311-67	USAS B31.1 (1967)
B313	ASME B31.3
B314	ASME B31.4
B315	ASME B31.5
B318	ASME B31.8
B319	ASME B31.9
B3112	ASME B31.12
ASME	ASME Section III, Class 2 (1980)
ASME-86	ASME Section III, Class 2 (1986)
ASME-92	ASME Section III, Class 2 (1992)
ASME-2015	ASME Section III, Class 2 (2015)
ASME-2017	ASME Section III, Class 2 (2017)
ASME-ND-2017	ASME Section III, Class 3 (2017)
BS806	British code
NORWEIGIAN-83	Norwegian code (1983)
NORWEIGIAN-90	Norwegian code (1990)
RCC-M	French code (1985)
SNCT	CODETI (1995)
SWEDISH	Swedish code (1978)
STOOMWEZEN	Dutch code (1989)
Z183	Z183 (1990)
Z184	Z183 (1992)
Z662	Z662
EUROPEAN	EN 13480

Units

SI SI units (default is English units)
RAD Radians for angles (default is degrees)

Vertical Axis

Z Vertical axis is Z (Default is Y)

OPTIONS Example

OPTIONS
HGRA,B311,RAD

[OPCODE]

[Options > Analysis > Code. Optional Section valid for CAEPIPE Version 10.10 and later]

This Section is Optional. If defined, then the syntax should be as given below. The values in square brackets ([...]) are optional.

Include axial force in stress calculation (0 = Off, 1 = On), Liberal allowable (0 = Off, 1 = On), [Use B31J (0 = Off, 1 = On. Valid only for ASME B31.x codes, for other codes, this field is ignored)], [Service level (valid for ASME Class 2 and Class 3 codes. 0 = B(Upset), 1 = C(Emergency) and 2 = D(Faulted). For other piping codes, leave this field as 0], [B31.8 Design Factor Index (F), Valid for B31.8 Code. Set 0 = 0.80, 1 = 0.72, 2 = 0.60, 3 = 0.5 and 4 = 0.4. For other piping codes, leave this field as 0], [Z183 Location Factor Index (L). Valid for Z183 Code. Set 0 = 1.0 and 1 = 0.8. For other piping codes, leave this field as 0], [Class Location (L). Valid for Z184 and Z662 Code. Value can be 1, 2, 3 or 4 for Z184 and **1 to 120 for Z662 [available in CAEPIPE Version 10.50 or later]**. For other codes, leave this field as 1], [Z184 Sour Service. Valid for Z184 Code. Set 0 = Nonsour service, 1 = Sour service. For other codes, leave this field as 0], [EN 13480 Seismic Factor (K). Valid only for EN 13480 code. Value should be > 1.00]

Example 1 [for ASME B31.3 Code]
OPCODE
1, 1, 1
Example 2 [for ASME Class 2 Code]
OPCODE
1, 0, 0, 2
Example 3 [for ASME B31.8 Code]
OPCODE
1, 1, 0, 0, 3
Example 4 [for Z183 Code]
OPCODE
1, 0, 0, 0, 0, 1
Example 5 [for Z184 Code]
OPCODE
1, 0, 0, 0, 0, 0, 4, 1
Example 6 [for EN 13480 Code]
OPCODE
1, 0, 0, 0, 0, 0, 1, 0, 1.33

[OPTEMP]

[Options > Analysis > Temperature. Optional Section valid for CAEPIPE Version 10.10 and later]

This Section is Optional. If defined, then the syntax should be as given below. The values in square brackets ([...]) are optional.

English units

Reference Temperature (F), [Number of Thermal Cycles], [Number of Thermal Loads (1, 2, 3 or 10)], [Solve Thermal Case (0 = Off, 1 = ON)], [Use Cold Modulus (0 = Off, 1 = ON. This field is ignored for ASME B31.3, B31.4, B31.5, 31.8, B31.12 and EN 13480]

SI units

Reference Temperature (C), [Number of Thermal Cycles], [Number of Thermal Loads (1, 2, 3 or 10)], [Solve Thermal Case (0 = Off, 1 = ON)], [Use Cold Modulus (0 = Off, 1 = ON. This field is ignored for ASME B31.3, B31.4, B31.5, 31.8, B31.12 and EN 13480]

Example 1 [English units]
OPTEMP
70
Example 2 [SI units]
OPTEMP
21.11, 7000, 1
Example 3 [English units with ASME B3.1 code]
OPTEMP
70, 7000, 0, 0

[OPPRES]

[Options > Analysis > Pressure. Optional Section valid for CAEPIPE Version 10.10 and later]

This Section is Optional. If defined, then the syntax should be as given below. The values in square brackets ([...]) are optional.

[Pressure Stress Option (0 or 1); 0 = Use PD/4t], [Bourdon effect (0 = Off, 1 = ON)], [Pressure Correction for Bends (0 = Off, 1 = On)], [Peak Pressure factor. Value should be >= 1.00]
Example 1
OPPRES
0, 1, 1, 1.50
Example 2
OPPRES

Note:
Leaving a BLANK line following the Section heading “OPPRES” will set the values as 0, 1, 1, 1.00.

[OPMISC]

[Options > Analysis >Dynamics &Misc. Optional Section valid for CAEPIPE Version 10.10 and later]

This Section is Optional. If defined, then the syntax should be as given below. The values in square brackets ([...]) are optional. Spectrum related options given below are available in CAEPIPE Version 10.50 or later

[Cutoff Frequency], [Number of modes], [Include Missing Mass (0 = Off, 1 = On)], [Include Friction in Dynamics (0 = Off, 1 = On)], [Include Hanger Stiffness (0 = Off, 1 = On)], [Spectrum Mode Sum (1 = Absolute, 2 = SRSS, 3 = Closely Spaced, 4 = NRL)], [Spectrum Direction Sum (1 = Absolute, 2 = SRSS)], [Spectrum Group Sum (1 = Absolute, 2 = SRSS)]

Example 1

OPMISC
100, 20, 1, 0, 1, 2, 2, 1

Example 2

OPMISC

Note:
Leaving a BLANK line following the Section heading "OPMISC" will set the values as 33, 20, 1, 1, 1, 2, 2, 1

SPCMS

[This section is available in CAEPIPE Version 10.50 or later]

Name of Spectrum (up to 32 characters).

First line: Name, No. of Spectrum entries, Abscissa Interpolation, Ordinate Interpolation, Abscissa Units, Ordinate Units.

Following lines: Name, Abscissa, Ordinate
.
.

SPLVL

[This section is available in CAEPIPE Version 10.50 or later]

Name of Spectrum Level (up to 8 characters).

Name, Abscissa Interpolation (0 = linear, 1 = log), Ordinate Interpolation (0 = linear, 1 = log), Abscissa Units(0 = frequency, 1 = period), Ordinate Units (0 = inch, 1 = mm, 2 = in/sec2, 3 = mm/sec2, 4 = g's).

MATERIAL

Name is Material Name (up to 5 characters). The values in square brackets ([...]) are optional.

English units

First line: Name, Density (lbf/in3), Poisson's ratio, [Long. joint factor], [circ. joint factor]

Following lines: Name, Temp (F), E (psi), alfa (in/in/F), [allowable stress (psi)], [yield stress (psi)], [rupture stress (psi)]

.

.

SI units

First line: Name, Density (kgf/m3), Poisson's ratio, [Long. joint factor], [circ.joint factor]

Following lines: Name, Temp (C), E (N/mm2), alfa (mm/mm/C), [allowable stress (N/mm2)], [yield stress (N/mm2)], [rupture stress (N/mm2)]

.

.

MATERIAL Example (English units)

```
MATERIAL
A53,0.283,0.300,1.00
A53,70,29.5E+6,6.07E-6,20000
A53,200,28.8E+6,6.38E-6,20000
A53,300,28.3E+6,6.60E-6,20000
A53,400,27.7E+6,6.82E-6,20000
A53,500,27.3E+6,7.02E-6,18900
ALU,0.098,0.330,1.00
ALU,70,10.0E+6,12.25E-6,10000
ALU,150,9.80E+6,12.67E-6,10000
ALU,200,9.60E+6,12.95E-6,9800
ALU,250,9.40E+6,13.12E-6,9000
ALU,300,9.20E+6,13.28E-6,6600
```

SOILS (Soil properties)

Soil properties are input as follows:

Note:

Name is Soil Name (up to 3 characters)
Type must be 0 or 1 (1 = Cohesive; 0 = Cohesive less)
If Type = 0, then Delta and Ks is required
If Type = 1, then Strength is required

English units

Name, Type (0 or 1), Density (lbf/ft3), Ground level (ft'in"), [Strength (psi) /Delta (deg or rad], [Ks]

SI units

Name, Type (0 or 1), Density (kgf/m3), Ground level (mm), [Strength (N/mm2) / Delta (deg or rad], [Ks]

PIPE (Section properties)

Section properties are input as follows for pipe sections:

Note:

Name is Section Name (up to 5 characters).

OD must be actual OD and not Nominal pipe size (e.g., input 2.125 and not 2.0 for 2" NPS)
Thk must be an actual numerical value and not a schedule.
Soil name is Name of the soil (up to 3 characters). It should be defined in the soil section before use in this section.

English units

Name, OD (inch), Thk (inch), Corrosion allowance (inch), Mill tolerance (%), Insulation density (lbf/ft3), Insulation Thk (inch), Lining density (lbf/ft3), Lining Thk (inch), Soil name

SI units

Name, OD (mm), Thk (mm), Corrosion allowance (mm), Mill tolerance (%), Insulation density (kgf/m3), Insulation Thk (mm), Lining density (kgf/m3), Lining Thk (mm), Soil name

PIPE Example (English units)

```
PIPE
8,8.625,0.5,0,0.0,11,2
6,6.625,0.28,0,0.0,11,2
```

LOADS (Load data)

Load data are input as follows:

Note:
Name is Load Name (up to 5 characters).
T2 through T10are Temperature 2 and through Temperature 10.
P2 through P10are Pressure 2 and through Pressure 10.
The values in square brackets ([...]) are optional.

English units

Name, Temperature 1 (F), Pressure 1 (psi), Specific Gravity, [T2,P2,T3,P3, Additional weight (lbf/ft), Design Temperature, Design Pressure, T4, P4, T5, P5, T6, P6, T7, P7, T8, P8, T9, P9, T10, P10, Wind]

SI units

Name, Temperature 1 (c), Pressure 1 (bar), Specific Gravity, [T2,P2,T3,P3, Additional weight (kgf/m), Design Temperature, Design Pressure, T4, P4, T5, P5, T6, P6, T7, P7, T8, P8, T9, P9, T10, P10, Wind 1, Wind 2, Wind 3, Wind 4]

Note:

Design Temperature, Design Pressure, T4, P4, T5, P5, T6, P6, T7, P7, T8, P8, T9, P9, T10, P10 are valid for CAEPIPE Version 10.10 or later.

Wind = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.20 or later

Wind 2 = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.40 or later

Wind 3 = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.40 or later

Wind 4 = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.40 or later

If Design Temperature and Design Pressure is not available then the Maximum of Operating Temperature and Maximum of Pressure input will be assigned as Design Temperature and Design Pressure respectively.

Example 1 [English Units]

LOADS

1,366,120,1.0,70,0,70,0,0

Example 2 [English Units]

LOADS

1,366,120,1.0,70,0,70,0,0,366,160

Example 2 [SI Units]

LOADS

C1,21.11,0,0.01,121.1,0.70,121.1,0.70,0,121.1,0.70,121.1,0.70,121.1,0.70,21.11,0,21.11,0,21.11,0, 21.11,0,21.11,0,1,0,0,1

BMATERIALS (Beam Materials)

Name is Material Name (up to 5 characters). The values in square brackets ([...]) are optional.

English units

Name, Young's Modulus (psi), [Poisson's ratio, Density (lbf/in3), Alfa (in/in/F)]

SI units

Name, Young's Modulus (Mpa), [Poisson's ratio, Density (kgf/m3), Alfa (mm/mm/c)]

BSECTIONS (Beam Sections)

Name is Beam Section name (up to 5 characters). The values in square brackets ([...]) are optional.

English units

Name, Axial area (in2), Major moment of inertia (in4), Minor moment of inertia (in4), [Torsional inertia (in4), Major shear area (in2), Minor shear area (in2), Depth (inch), Width (inch)]

SI units

Name, Axial area (mm2), Major moment of inertia (mm4), Minor moment of inertia (mm4), [Torsional inertia (mm4), Major shear area (mm2), Minor shear area (mm2), Depth (mm), Width (mm)]

BLOADS (Beam Loads)

Name is Beam Load name (up to 5 characters). The values in square brackets ([...]) are optional.

T2 through T10are Temperature 2 and through Temperature 10. Valid for CAEPIPE Version 10.10 or later.

Wind 1 = Include or Exclude Wind (1 = Include, 0 = Exclude)

Wind 2 = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.40 or later

Wind 3 = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.40 or later

Wind 4 = Include or Exclude Wind 1 (1 = Include, 0 = Exclude). Valid for CAEPIPE Version 10.40 or later

English units

Name, T1 (F), [T2 (F), T3 (F), Additional weight (lbf/ft), T4, T5, T6, T7, T8, T9, T10, Wind 1, Wind 2, Wind 3, Wind 4]

SI units

Name, T1 (c), [T2 (c), T3 (c), Additional weight (kgf/m), T4, T5, T6, T7, T8, T9, T10, Wind 1, Wind 2, Wind 3, Wind 4]

LAYOUT

The piping layout is entered as a (key letter-data) pair, which may be input in any order. The exceptions to this convention are the comments which must always appear last. The pairs are separated by commas. A data line which contains only continued comments (from the previous line) is acceptable. A list of key letters follows:

*	Model comments
F	From node
T	To node
H	Hydro test Load
L	Location node
K	Code for node
M	Material number
P	Pipe (Section) property number
J	Joint code
X	X offset
Y	Y offset
Z	Z offset
B	Bend radius
C	Comments (separated by commas)

LAYOUT Example (English units)

```
LAYOUT
F10,KA,M5,P12,CT=650,P=500
T20,KI,X10’8-1/2,B20
T30,Z8
T40,JR,X1’6,CWGT=250
```

*	The model comment section allows entry of notes relevant to analysis model. The model comment can be up to 70 characters.

Example: *Load cases considered for analysis are T1 and P1.

F (From) — From node is specified. When a new branch is started, the first node of the branch is specified as a "From" node. The X, Y and the Z fields are taken as coordinates rather than offsets from the previous node.

T (To) — To node is specified. This is a "To" node from the previous "From"node or the previous "To" node (but not from the previous "Location node").

L (Location) — Location node is used to input additional data at a node when the node has more than one data item such as a hanger/force, etc.

K (Code) — The following codes may be used:

Code	Description
A	Anchor
B	Branch connection
G	Generic Support (valid for CAEPIPE Version 10.30 or later)
H	Hinge (To node only)
M	Miter bend (To node only)
I	Tangent intersection (To node only)
T	Welding Tee
S	Sweepolet
W	Weldolet
F	Fabricated Tee
E	Extruded Tee
R	Radiused branch
P	Branch on thickened pipe

M (Material) — A material is retained until altered. Another material should be entered only when there is a change.

P (Pipe) — A pipe (section property) is retained until altered. Another section property should be entered only when there is a change.

J (Joint Code)

Code	Description
B	Ball joint
C	Cut pipe
D	Reducer
E	Expansion
I	Jacket bend
L	Elastic element
M	Beam
P	Jacket pipe
R	Rigid
S	Slip joint
T	Tierod
V	Valve

The weight of a rigid joint or a valve should be entered using aWGTcomment. The stiffnesses of the expansion joint should be entered using an ES comment and the pressure thrust area should be entered using the TA comment.

X, Y and Z — The offsets May be entered in combination of feet, inches and fractions of an inch for English units and mm for SI units.

Example (English units)

Entry	**Value**
-10	-10 ft
10'8 or 10-8	10 ft 8 in
0'8 or 0-8	8 in
10.5	10 ft 6 in
1'6-3/8 or 1-6-3/8	1 ft 6.375 in

B (Bend Radius) — The bend radius (inch or mm) is entered only if a tangent intersection(i.e., Bend, Jbend and Miter Bend) has been specified. The default is the long radius for the bend and jacketed bend.

C (Comment) — The comment section allows entry of data related to a particular node or element. For example, a pipe end specified as a hinge would have the rotational spring constant and the direction vector entered in the comment section. A line temperature can be entered as comment and is retained until changed. Multiple comments may be entered separated by commas. If the last comment is followed by a comma, the comment is continued on the next line.

The comments are as follows:

AMB — Ambient or reference temperature (F or C)
Default is 70 F.
Example: AMB=80

AXIAL — Set support axis along the local-x direction (pipe axis). Use only for Limit Stop, Skewed Restraint and Snubber to define support axis along the local-x direction (pipe axis). Available in CAEPIPE Version 10.50 or later.

AWGT — Additional weight for valve (lbf. or kgf)
Use only for valve.
Example: AWGT=100

BTHK — Bend thickness (inch or mm)
Use for Bend and Jbend (core).
Example: BTHK=6.01

BMAT — Name of Bend Material (Up to 5 characters)
Should be defined in Material section before use.
Example: BMAT=BM1

BK — Bending stiffness (in-lb/deg or Nm/deg)
Use only for Bellows
Example: BK=10000

BSIF — Bend SIF

	In-plane Out-plane Example: BSIF(1.5,2.0) For Piping code with one SIF, use in-plane=Out-plane=SIF Example: BSIF (2.0,2.0)
BETA	Beta angle for Beam (deg or rad) Use only for Beam Example: BETA=90
CONE	Cone angle for reducer (deg. or rad.)
CNOD	Connected to Node Use for Guide, Hanger, Limit Stop, Rod Hanger, Skewed Restraint, User Hanger, Snubber and Constant Support Example: CNOD=150
CLD	Cold load Use only for User Hanger. Example: CLD=1
CRTCH	Crotch radius for an extruded tee (inch or mm) Example: CRTCH=1.25
CS	Constant support spring Example: CS=2: Two constant support springs
CWGT	Concentrated weight (lbf or kgf) Use only for concentrated weight. Example: CWGT=200
D or DIS	Specified displacements (Inch or mm) Use for Anchor. Also valid for Nozzle (valid for CAEPIPE 10.10 or later). Note: Entry of zero is ignored and not treated as a specified displacement. Example: D(0.1,-0.25,0) or DIS(0.1,-0.25,0) DX=0.1,DY=-0.25 or DISX=0.1,DISY=-0.25
DV	Direction vector.DV(xcomp, ycomp, zcomp) Use for Hinge, Snubber and Nozzle Example: DV(1,-2,0)
DIS2	Specified Displacement for Temperature 2 (inch or mm) Use only for Anchor Also valid for Nozzle (valid for CAEPIPE 10.10 or later). Note: Entry of zero is ignored and not treated as a specified displacement. Example: DIS2(1,-2,0)
DIS3	Specified Displacement for Temperature 3 (inch or mm) Use only for Anchor. Also valid for Nozzle (from CAEPIPE 10.10 or later). Note: Entry of zero is ignored and not treated as a specified displacement. Example: DIS3(1,-2,0)

DISn	Specified Displacement for Temperature 4 through 10 (inch or mm). n can be 4 through 10. Valid for CAEPIPE 10.10 or later. Use only for Anchor and Nozzle Note: Entry of zero is ignored and not treated as a specified displacement. Example: DIS4(1,-2,0), DIS7(1,-1,0)
E	Young's modulus (psi or Mpa) Use only for Nozzle Example: E=28E6
ES	Expansion joint stiffnesses Axial(lb/in or N/mm) Lateral(lb/in or N/mm) Torsional(in-lb/deg or NM/deg) Example: ES(1000,5000,200)
F or FIXD	Translational restraint Example: FIXD(1,0,1) : Restrain X and Z translations FIXDX=1 or FX or FIXDX
FF	Bend Flexibility factor Use for Bends and Miter bends Example: FF=1.5
FIXR	Rotational restraint Example: FIXR(0,1,0) : Restrain Y rotation. FIXRY=1 or FIXRY
FLANGE	FLANGE = Type, where <u>Type</u> <u>Description</u> WN Weld neck flange SO Single welded slip on DW Double welded slip on SW Socket welded FW Fillet welded LJ Lap joint TH Threaded Example: FLANGE=TH (Threaded flange)
FOR	Force (lb or N) Use for Force and Harmonic load. Example: FOR(100,0,-200) or FORX=100,FORZ=-200
FLC	Apply Force defined to the specified load case. Available for CAEPIPE Version 10.30 or later Use 1 to add it to Sustained Load Case Use 2 to 11 to add it to T1 through T10 load cases. Use 12 to add it to Static Seismic Load case Example: FLC=12
FFOR	Friction force (lb or N)

	Use for Slip joint Example: FFOR=100
FTOR	Friction torque (ft-lb or Nm) Use for Slip joint Example: FTOR=100
FRCT	Bending and Torsional friction Torque (ft-lb or Nm). FCRT(Bending, Torsional) Use only for Ball joint Example: FCRT(100,150)
FREE	Free anchor during hanger design Example: FREE: Free all directions FREEY: Free Y direction
FRE	Frequency (Hz) Use only for Harmonic Load. Example: FRE=30
G	Guide
GRA1	Generic Support - Group A Stiffness for (1,1) (1,2) and (1,3) in (lb/in or N/mm). Available for CAEPIPE Version 10.30 or later. GRA1(Rigid, 0, 0)
GRA2	Generic Support - Group A Stiffness for (2, 2) (2, 3) and (3,3) in (lb/in or N/mm). Available for CAEPIPE Version 10.30 or later. GRA2(1E+10, 0, Rigid)
GRB1	Generic Support - Group B Stiffness for (1,4) (1,5) and (1,6) in (lb/deg or N/deg). Available for CAEPIPE Version 10.30 or later. GRB1(1000, 0, 0)
GRB2	Generic Support - Group B Stiffness for (2,4) (2,5) and (2,6) in (lb/deg or N/deg). Available for CAEPIPE Version 10.30 or later. GRB2(1000, -2000, 0)
GRB3	Generic Support - Group B Stiffness for (3,4) (3,5) and (3,6) in (lb/deg or N/deg). Available for CAEPIPE Version 10.30 or later. GRB3(1000, 0, 0)
GRC1	Generic Support - Group C Stiffness for (4,4) (4,5) and (4,6) in (in-lb/deg or Nm/deg). Available for CAEPIPE Version 10.30 or later. GRC1(Rigid, 0, 1000)
GRC2	Generic Support - Group C Stiffness for (5,5) (5,6) and (6,6) in (in-lb/deg or Nm/deg). Available for CAEPIPE Version 10.30 or later. GRC2(Rigid, 0, 1E+8)

GAP	Tension and Compression gap (inch or mm) GAP(Tension, Compression) Use only for Tie rod. Example: GAP(10,15)
GGAP	Guide Gap (inch or mm) Use only for guide Example: GGAP=5
HSG	Hydrotest Specific Gravity Use only for Hydrotest Load Example: HSG=0.7
HPRES	Hydrostatic Pressure (psi or bar) Use only for Hydrotest Load Example: HPRES=3
HTYP	Defines the Hanger Type. Refer to end of this appendix for details. Use only for Hanger Example: HTYP=16 (Grinnell)
IN1	Intermediate Node 1 for Bends. Node Number (>1 and < 99999) Angle (deg or rad) Use for Bend and Jbend Example: IN1(300,30)
IN2	Intermediate Node 2 for Bends. Node Number (>1 and < 99999) Angle (deg or rad) Use for Bend and Jbend Example: IN2(400,30)
INSF	Insulation factor Use only for Valve Example: INSF=3.0
IPS	User Sustained SCF - Pressure Use only for IGEM Code Example: IPS=1.3
IBS	User Sustained SCF - Bending Use only for IGEM Code Example: IBS=1.5
ITS	User Sustained SCF - Thrust Use only for IGEM Code Example: ITS=1.2
IQS	User Sustained SCF - Torsion Use only for IGEM Code Example: IQS=1.5
ISS	User Sustained SCF - Shear Use only for IGEM Code Example: ISS=2.0

IPC	User Cyclic SCF - Pressure Use only for IGEM Code Example: IPC=1.3
IBC	User Cyclic SCF - Bending Use only for IGEM Code Example: IBC=1.5
ITC	User Cyclic SCF - Thrust Use only for IGEM Code Example: ITC=1.2
IQC	User Cyclic SCF - Torsion Use only for IGEM Code Example: IQC=1.5
ISC	User Cyclic SCF - Shear Use only for IGEM Code Example: ISC=2.0
JCAP	Jacked End Cap Defines the data type Jacked End Cap Example: JCAP
JMAT	Jacket Material (up to 3 characters) Use for Jpipe and Jbend Example: JMAT=A53
JSEC	Jacket Section (up to 3 characters) Use for Jpipe and Jbend Example: JSEC=N10
JLOAD	Jacket Load (up to 3 characters) Use for Jpipe and Jbend Example: JLOAD=L3
JTHK	Jacket Thickness (inch or mm) Use only for Jbend
JR	Jacket Radius (inch or mm) Use only for Jbend. Example: JR=6.75
K	Translational stiffness (lb/inch or N/mm) Use for Skewed restraint Example: K=500,DV(1.5,-0.75,0.25)
KR	Rotational stiffness (in-lb/deg. or N-m/deg) Use for Skewed restraint Example: KR=1200,DV(1.2,2.5,0)
KTIE	Tension and Compression stiffness.KTIE(Tension, Compression) (lb/in or N/mm). Use only for Tie rod Example: KTIE(1000,1500)

LA	Anchor in Local Coordinate System (LCS) or Global Coordinate System (GCS). Valid for CAEPIPE Version 10.10 or later. Use only for Anchor. Example: LA=1 (Anchor in LCS), LA=0 (Anchor in GCS)
LD	Anchor Displacement in Local Coordinate System (LCS) or Global Coordinate System (GCS). Valid for CAEPIPE Version 10.10 or later. Use only for Anchor. Example: LD=1 (Anchor Displacement in LCS), LD=0 (Anchor Displacement in GCS)
LS	Limit stop LS(M1,M2) M1=allowable movement in negative direction (in. or mm) M2=allowable movement in positive direction (inch or mm) Example : LS(-1.0,1.5), DV(0,1,0), MU=0.3
LOAD	Beam load reference (up to 5 characters) Note: Beam load should be defined in BLOADS section before use. Example: LOAD=B1
LEN	Length (inch or mm) Use only for Branch SIF with type Branch on Thickened Pipe Example: LEN=5
L1	Length 1 (ft-in or mm) Use to define "L" for API 650 Nozzle and "L1" for WRC 297 Nozzle Example: L1=3'0" or L1=900
L2	Length 2 (ft-in or mm) Use only for WRC 297 Nozzle to define "L2" Example: L2=4'0" or L2=1200
LONG	Cut long (inch or mm) Use only for Cut pipe. Example: LONG=100
LTAG	Level Tag for Supports. Use only when Spectrum and Spectrum Level defined. LTAG is available in CAEPIPE Version 10.50 or later.
LXAX	Local X axis.LXAX(xcomp, ycomp, zcomp) Use only for Elastic Element Example: LXAX(1,0,0)
LYAX	Local Y axis.LYAX(xcomp,ycomp,zcomp) Use only for Elastic Element Example: LYAX(0,1,0)
MAT	Beam material reference (up to 5 characters) Note: Beam material should be defined in Beam material (BMATERIALS) section before use. Example: MAT=M1

MM	Mismatch (inch or mm) Use only for weld Example: MM=5
MLV	Maximum load variation (%) in hanger design Default is 25%. Example: MLV=30
MOM	Moment (ft-lb or NM) Example: MOM(200,-100,0) or MOMX=200, MOMY=-100
MU	Friction co-efficient Example: MU=0.3
NOD	Nozzle outside diameter (inch or mm) Example: NOD=104
NTHK	Nozzle thickness (inch or mm) Example: NTHK=6.01
NOZZLE	Defines the Nozzle data type. (650, 297or 5500). Example: NOZZLE=650 (API 650) or NOZZLE=297 (WRC 297) Example: NOZZLE=5500 (valid for CAEPIPE 10.10 or later)
OD1	Outer diameter at from end for the reducer (inch or mm)
OD2	Outer diameter at to end for reducer (inch or mm)
OFFSET	Offset of concentrated weight from node or additional weight of valve from the center of valve (inch or mm) OFFSET(X offset, Y offset, Z offset). Example: OFFSET(0,18,0)
PAD	Thickness of reinforcement for fabricated tee (inch or mm) Example: PAD=0.25
PH	Phase (deg or rad) Use only for Harmonic Load Example: PH=10
P or PRES	Pressure (psig or bar) Example: P=500
ROT	Specified rotation (deg or rad) Note: Entry of zero is ignored and not treated as a specified rotation. Example: ROT(1.5,0,-0.25) ROTX=1.5,ROTZ=-0.25
RLIM	Rotation limit (deg or rad) Use only for Hinge Example: RLIM=10
ROTL	Rotational limit in Bending and Torsion ROTL(Bending, Torsion) (deg or rad)

	Use only for Ball joint Example: ROTL(10,20)
ROTK	Rotational stiffness. ROTK(kxx,kyy,kzz) (in-lb/deg. or N-m/deg) Use for Anchor and Elastic element Example: ROTK(5,6,3)
ROT2	Rotational Displacement for Temperature 2. (deg or rad) ROT2(kxx,kyy,kzz) Use only for Anchor Example: ROT2(10,15,10)
ROT3	Rotational Displacement for Temperature 3. (deg or rad) ROT3(kxx,kyy,kzz) Use only for Anchor Example: ROT3(10,15,10)
ROTn	Rotational Displacement for Temperature 4 through 10. (deg or rad). "n" can be 4 through 10. Valid for CAEPIPE 10.10 or later. ROT3(kxx,kyy,kzz) Use only for Anchor Example: ROT5(10,15,10), ROT10(11,5,3)
R	Fillet radius (inch or mm) Use only for Branch SIF (Radiused Branch and Branch Connection) Example: R=10
RPAD	Reinforcing Pad (0 or 1) Use only for Nozzle (API 650) Example RPAD=1
RK	Rotational Stiffness in Bending and Torsion. RK(Bending, Torsion) (ft-lb or Nm) Use only for Ball joint Example: RK(100,150)
SG	Specific Gravity Example: SG=0.8
SIF	Stress intensification factor at node. SIF=value or SIF(in-plane, Out-plane) Example: SIF=1.3 or SIF(1.5,2.0)
SIFA	Axial Stress intensification factor. Valid for CAEPIPE 10.10 or later. Use this field to define Axial SIF for applicable codes (ex. B31J, B31.3, etc.). Will be ignored, if not applicable for the analysis code defined.
SIFT	Torsional Stress intensification factor. Valid for CAEPIPE 10.10 or later. Use this field to define Torsion SIF for applicable codes (ex. B31J, B31.3, etc.). Will be ignored, if not applicable for the analysis code defined.
SHORT	Cut short (inch or mm). Use only for Cut pipe. Example: SHORT=100

SHEARY	Set support axis in the local-y directionof pipe. Use only for Limit Stop, Skewed Restraint and Snubber to define support axis in the local-y direction of pipe. Available in CAEPIPE Version 10.50 or later.
SHEARZ	Set support axis in the local-z directionof pipe. Use only for Limit Stop, Skewed Restraint and Snubber to define support axis in the local-z direction of pipe. Available in CAEPIPE Version 10.50 or later.
SEC	Beam section reference (up to 5 characters) Note: Beam section should be defined in Beam section (BSECTIONS) before use. Example: SEC=BS1
SEIS	Seismic displacement (inch or mm)(x,y,z) Example: SEIS (-0.25, 0.00, -0.25)
SERO	Seismic rotation (deg or rad) (rx,ry,rz) Example: SERO (0.25, 0.00, 0.25)
SETT	Settlement displacement (inch or mm)(x,y,z) Example: SETT (-0.25, -0.25, -0.25)
SPIDER	Defines the data type SPIDER Example: SPIDER
SR	Turn on the option Short range. Use only for Hanger. Example: SR
STIFF	Stiffness (lb/in or N/mm) Use for Guide, Limit stop and Snubbers Example: STIFF=1000
STRO	Settlement rotation (deg or rad) (rx,ry,rz) Example: STRO (0.15, 0.15, 0.15)
T or TEMP	Temperature (F or C) Example: T=650
TA	Pressure thrust area for bellows and Slip joints (in2 or mm2). Example: TA=12.3
TAG	Support Tag (up to 14 characters). Valid for CAEPIPE 10.10 or later. Use for Anchor, Guide, Hanger, Limit Stop, Nozzle, Restraint, Rod hanger, Skewed restraint and User Hanger. Example: TAG=GUID1S01
THK	Thickness (inch or mm) Use only for Branch SIF (Radiused Branch & Branch on Thickened Pipe) Example: THK=10
THK1	Thickness at from end for reducer (inch or mm)

THK2	Thickness at to end for reducer (inch or mm)
THKF	Thickness factor Use only for Valve. Example: THKF=3.0
TRAK	Translational Stiffness (lb/in or N/mm).TRAK(kx,ky,kz) Use for Anchor and Elastic element. Example: TRAK(1000,1500,2000)
TJOINT	Defines the Threaded Joint Example: TJOINT
U or UNIF	Uniform load (lbf/ft or Kgf/m) Example: U=200
US	User-defined spring hanger US(No.of hangers, spring rate(lb/inch or N/mm), hot load(lb or N)) Examples: US(2,600,1540) US(1,0,2300) : Constant support
VS	Variable spring hanger Example : VS, VS=2: two variable spring hangers
VOD	Vessel outside diameter (inch or mm). For Nozzle attached to Spherical Vessel, this field can be used to define Vessel Radius. Use only for Nozzle. Example: VOD=250
VTHK	Vessel thickness (inch or mm) Use only for Nozzle Example: VTHK=10
VWGT	Valve weight (lbf or kgf) Example: VWGT=100
WGT	Weight of an item (ball joint, flange, Slip joint, etc.) (lbf or kgf) Example: WGT=50
WS	Widely Spaced Use only for Miter bend Example: WS
WTYPE	Weld type Example: WTYPE=1 (1 = Butt weld, 2 = Fillet weld, 3 = Concave fillet weld, 4 = Tapered Transition)

PUMPS

Description (up to 16 characters)
Suction/Discharge location (0 = Top, 1 = Side, 2 = End)
Shaft axis (xcomp, ycomp, zcomp)
Center of pump (x, y, z)

English units

Desc, Horizontal / Vertical inline (0 or 1), Shaft axis, Center of pump (ft-in), Suction node, Suction location, Discharge Node, Discharge Location

SI units

Desc, Horizontal / Vertical inline (0 or 1), Shaft axis, Center of pump (mm), Suction node, Suction location, Discharge Node, Discharge Location

Horizontal Pumps

Desc, 0, Shaft axis, Center of pump, Suction node, Suction location, Discharge Node, Discharge Location

Vertical Inline Pumps

Desc, 1, Suction node, Discharge Node

Example

```
PUMPS
427BSOUTH,1,350
427ANORTH,0,0.0000,1.0000,0.0000,-3.8299,-13'8”,44.33,390,2,410,2
427WEST,1,300
```

COMPRESSORS

Description (up to 16 characters)
Shaft axis (xcomp, ycomp, zcomp)

Description, Shaft axis direction, Inlet Node, Exhaust Node, Extraction Node 1, Extraction Node 2.

Example

```
COMPRESSORS
Compressor1,1.0000,0.0000,0.0000,210,300
```

TURBINES

Description (up to 16 characters)
Shaft axis (xcomp, ycomp, zcomp)

Description, Shaft axis direction, Inlet Node, Exhaust Node, Extraction Node 1, Extraction Node 2.

Example

```
TURBINES
TURBINE1,1.0000,0.0000,0.0000,250,360
```

SEISMIC

X,Y,Z Static equivalent seismic loads in g’s, [Seismic Combination 1 = Absolute Sum, 2 = SRSS].

The value in square brackets ([...]) is optional.

SEISMIC Example: Static seismic loads of 0.25 and 0.3 g's in X and Z directions.

```
SEISMIC
0.25,0.00,0.30
```

[ASCES]

Static Seismic Loads (g's) as per ASCE/SEI 7-16 that can be defined through Options > Analysis > Static Seismic > Use ASCE.

This Section is Optionaland is valid for CAEPIPE Version 10.10 and later. If defined, then the syntax should be as given below.

Occupancy Category [0 = I, 1 = II, 2 = III and 3 = IV], Site Class [0 = A, 1 = B, 2 = C, 3 = D, and 4 = E], MCE Spectral Acceleration, Component height(ft-in or mm), Structure height(ft-in or mm), Component Amplication Factor (Ap) [1.0 to 2.50], Response Modification Factor (Rp) [1.0 to 12.0], Importance Factor (Ip) [1.0 to 1.5], Allowable Stress Design Factor, Seismic Combination.

Example

ASCES

2, 3, 10.00, 50'0", 40'0", 2.50, 12.00, 1.00, 0.70, 2

Note:
If both SEISMIC and ASCES definition are defined, then CAEPIPE will ignore SEISMIC definition.

WIND

This section is compatible with all earlier version of CAEPIPE up to and including 10.0.

See ASCEW/1991W, WIND1, WIND2, WIND3 and WIND4 for CAEPIPE Version 10.10 or later.

Wind velocity (mph or m/s), Shape factor (defaults to 0.6), X, Y, Z components of wind direction.

WIND Example: A wind load of 100 mph in the X direction.

```
WIND
100,0.600,1.000,0.000,0.000
```

[ASCEW]

Wind Profile as per ASCE/SEI 7-16 that can be defined through Layout Window > Misc > Wind – ASCE/SEI 7-16.

This Section is Optionaland is valid for CAEPIPE Version 10.10 and later. If defined, then the syntax should be as given below.

Occupancy Category [0 = I, 1 = II, 2 = III and 3 = IV], Basic Wind Speed (mph or m/s), Wind Directionality Factor, Exposure Category (0 = B, 1 = C and 2 = D], Hill Type (0 = No Hill, 1 = 2D Ridge, 2 = 2D Escarpment and 3 = 3D Axisymmetric Hill], Hill height (ft-in or mm), Crest Distance (ft-in or mm), Height above Ground Level (ft-in or mm), Distance from Crest to Site (ft-in or mm), Type of Surface (0 = Moderately Smooth, 1 = Rough, 2 = Very Rough), Gust Factor.

Example

ASCEW

2, 100, 0.9500, 0, 0, 0, 0, 0, 0, 0, 0.8500

[1991W]

Wind Profile as per EN 1991-1-4 (2010) that can be defined through Layout Window > Misc > Wind – EN 1991-1-4 (2010).

This Section is Optional and is valid for CAEPIPE Version 10.30 and later. If defined, then the syntax should be as given below.

Basic Wind Speed (mph or m/s), Air Density(lbf/ft3 or kgf/m3), Terrain Category (0 = 0, I = I, 2 = II, 3 = III and 4 = IV), [Direction Factor, Season Factor, Terrain Orography, Turbulence Factor]

Example

1991W

50,0.001,2,1.0000,1.0000,1.0000,1.0000

[WIND1/WIND2/WIND3/WIND4]

Wind Profile that can be defined through Layout Window >Loads > Wind1 / Wind 2/ Wind 3 / Wind 4.

These Sections are Optional and are valid for CAEPIPE Version 10.10 and later. If defined, then the syntax should be as given below.

First line: Use WindCode (0 = None, 1 = ASCE and 2 = EN 1991-1-4. Can be defined as 1 or 2, when ASCEW/1991W is defined), Shape factor, X, Y, Z components of wind direction, Elevation Units (0 = feet and 1 = m), Pressure Units (0 = psi, 1 = kPa and 2 = kg/cm2), Velocity Units (0 = mph, 1 = kmh and 2 = m/s), Profile Type (0 = Pressure vs Elevation and 1 = Velocity vs Elevation)

Second line: Number of Wind Profiles

Following lines: Elevation, Velocity / Pressure

.

Example 1 [Manual Wind Profile]

WIND1

0,0.6000,1.0000,0.0000,0.0000,0,0,0,1

2

0,100

100,150

Example 2 [ASCE Wind Profile]

WIND2

1,0.6000,-1.0000,0.0000,0.0000

Example 3 [EN1991-1-4 Wind Profile]

WIND3

2, 0.6000, -1.0000, 0.0000, 0.0000

ALLOWABLES

Node, FX/P (lb or N), FY/VL (lb or N), FZ/VC (lb or N), MX/MT (ft-lb or Nm), MY/MC (ft-lb or Nm), MZ/ML (ft-lb or Nm).

Note:

For Nozzle, enter Radial (P), y shear (VL), z shear (VC), Torque (MT), Circumferential Moment (MC) and Longitudinal Moment (ML)

For Anchor, enter Global forces FX, FY &FZ and Global Moments MX, MY and MZ

Example:

ALLOWABLES

10,84817.63,848176.31,848176.31,454771.2,45477.12,45477.12

Example MBF Format:

```
Sample MBF

VERSION
10.50
OPTIONS
HGRA,B319
OPCODE
0,1,0,0,0,0,1,0,1.20
OPTEMP
40,7000,1,0,0
OPPRES
0,1,1,1.00
OPMISC
50,50,1,1,1,4,2,2
MATERIAL
A53,0.283,0.300,1.00
A53,-325,31.4E+6,5.00E-6,20000
A53,-200,30.8E+6,5.35E-6,20000
A53,-100,30.2E+6,5.65E-6,20000
A53,70,29.5E+6,6.07E-6,20000
A53,200,28.8E+6,6.38E-6,20000
A53,300,28.3E+6,6.60E-6,20000
A53,400,27.7E+6,6.82E-6,19900
A53,500,27.3E+6,7.02E-6,19000
A53,600,26.7E+6,7.23E-6,17900
A53,650,26.1E+6,7.33E-6,17300
A53,700,25.5E+6,7.44E-6,16700
A53,750,24.8E+6,7.54E-6,13900
A53,800,24.2E+6,7.65E-6,11400
A53,850,23.3E+6,7.75E-6,8700
A53,900,22.4E+6,7.84E-6,5900
A53,950,21.4E+6,7.91E-6,4000
A53,1000,20.4E+6,7.97E-6,2500
A53,1050,19.2E+6,8.05E-6,1600
A53,1100,18.0E+6,8.12E-6,1000
API,0.283,0.300,1.00
API,-325,31.4E+6,5.00E-6,20000
```

```
API,-200,30.8E+6,5.35E-6,20000
API,-100,30.2E+6,5.65E-6,20000
API,70,29.5E+6,6.07E-6,20000
API,200,28.8E+6,6.38E-6,20000
API,300,28.3E+6,6.60E-6,20000
API,400,27.7E+6,6.82E-6,19900
API,500,27.3E+6,7.02E-6,19000
API,600,26.7E+6,7.23E-6,17900
API,650,26.1E+6,7.33E-6,17300
API,700,25.5E+6,7.44E-6,16700
API,750,24.8E+6,7.54E-6,13900
API,800,24.2E+6,7.65E-6,11400
API,850,23.3E+6,7.75E-6,8700
API,900,22.4E+6,7.84E-6,5900
API,950,21.4E+6,7.91E-6,4000
API,1000,20.4E+6,7.97E-6,2500
API,1050,19.2E+6,8.05E-6,1600
API,1100,18.0E+6,8.12E-6,1000
PIPE
36I,36,0.375,0.075,0.0,13,2
36O,36,0.375,0.075,0.0,13,2.5
54O,54,0.375,0.075,0.0,13,2.5
54I,54,0.375,0.075,0.0,13,2
LOADS
36O,100,125,1.0,40,0,40,0,77.2,100,125,,,,,,,,,,,,,,,1,0,0,0
36I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
30O,100,125,1.0,40,0,40,0,65.9,100,125,,,,,,,,,,,,,,,1,0,0,0
30I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
24O,100,125,1.0,40,0,40,0,54.6,100,125,,,,,,,,,,,,,,,0,0,0,0
24I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
20O,100,125,1.0,40,0,40,0,48.9,100,125,,,,,,,,,,,,,,,1,0,0,0
20I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
18O,100,125,1.0,40,0,40,0,45.2,100,125,,,,,,,,,,,,,,,1,0,0,0
18I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
16O,100,125,1.0,40,0,40,0,41.4,100,125,,,,,,,,,,,,,,,1,0,0,0
16I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
14O,100,125,1.0,40,0,40,0,37.6,100,125,,,,,,,,,,,,,,,1,0,0,0
14I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
12O,100,125,1.0,40,0,40,0,35.3,100,125,,,,,,,,,,,,,,,1,0,0,0
12I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
10O,100,125,1.0,40,0,40,0,31.5,100,125,,,,,,,,,,,,,,,1,0,0,0
10I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
8O,100,125,1.0,40,0,40,0,27.5,100,125,,,,,,,,,,,,,,,1,0,0,0
8I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
6O,100,125,1.0,40,0,40,0,23.8,100,125,,,,,,,,,,,,,,,1,0,0,0
6I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
42O,100,125,1.0,40,0,40,0,88.5,100,125,,,,,,,,,,,,,,,1,0,0,0
42I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
48O,100,125,1.0,40,0,40,0,99.8,100,125,,,,,,,,,,,,,,,1,0,0,0
48I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
54O,100,125,1.0,40,0,40,0,111.1,100,125,,,,,,,,,,,,,,,1,0,0,0
54I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
60O,100,125,1.0,40,0,40,0,122.4,100,125,,,,,,,,,,,,,,,1,0,0,0
60I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
72O,100,125,1.0,40,0,40,0,144.9,100,125,,,,,,,,,,,,,,,1,0,0,0
72I,100,125,1.0,40,0,40,0,0,100,125,,,,,,,,,,,,,,,0,0,0,0
SPCMS
Malta--N-Y-3,14,0,0,1,4
```

```
Malta--N-Y-3,0,0.0856
Malta--N-Y-3,0.037,0.1284
Malta--N-Y-3,0.074,0.1712
Malta--N-Y-3,0.11,0.214
Malta--N-Y-3,0.331,0.214
Malta--N-Y-3,0.551,0.214
Malta--N-Y-3,1.914,0.061665
Malta--N-Y-3,3.276,0.036023
Malta--N-Y-3,4.638,0.025443
Malta--N-Y-3,6,0.019667
Malta--N-Y-3,8,0.011063
Malta--N-Y-3,10,0.00708
Malta--N-Y-3,20,0.00177
Malta--N-Y-3,30,0.000787
Malta--N-Y_b-1,23,0,0,1,4
Malta--N-Y_b-1,0,0.0856
Malta--N-Y_b-1,0.00011,0.0857
Malta--N-Y_b-1,0.0011,0.0869
Malta--N-Y_b-1,0.0368,0.1284
Malta--N-Y_b-1,0.074,0.1712
Malta--N-Y_b-1,0.11,0.214
Malta--N-Y_b-1,0.331,0.214
Malta--N-Y_b-1,0.551,0.214
Malta--N-Y_b-1,1.23,0.0957
Malta--N-Y_b-1,1.91,0.0617
Malta--N-Y_b-1,2.59,0.0455
Malta--N-Y_b-1,3.28,0.036
Malta--N-Y_b-1,3.96,0.0298
Malta--N-Y_b-1,4.64,0.0254
Malta--N-Y_b-1,5.32,0.0222
Malta--N-Y_b-1,6,0.0197
Malta--N-Y_b-1,8,0.0111
Malta--N-Y_b-1,10,0.0071
Malta--N-Y_b-1,20,0.0018
Malta--N-Y_b-1,30,0.0008
Malta--N-Y_b-1,50,0.0002832
Malta--N-Y_b-1,100,7.08E-5
Malta--N-Y_b-1,1000,7.08E-7
SPLVL
LVL-0,Malta--N-Y-3,Malta--N-Y-3,Malta--N-Y_b-1,1.000,0.043,1.000
LVL-1,Malta--N-Y-3,Malta--N-Y-3,Malta--N-Y_b-1,1.000,0.043,1.000
LAYOUT
F10,KA,CTAG=,LTAG=LVL-0
T20,KI,B81,MAPI,P54I,Z11'0",CL=54I
T30,Y-20'0",CVS=1,HTYP=27,MLV=25,TAG=,LTAG=LVL-0
T40,KI,B81,Y-10'9"
T50,Z9'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T60,P54O,Z2'0",CL=54O
T65,Z10'0"
T70,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T75,Z12'0"
T80,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T85,Z12'0"
T90,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
```

```
T100,Z11'0"
T110,KA,Z3'0",CTAG=,LTAG=LVL-0
T120,Z3'0"
T130,JD,Z3'0",COD1=54,THK1=0.375,OD2=36,THK2=0.375
T140,KI,B54,P36I,Z3'8",CL=36I
T150,KI,B54,X2'11",Y-4'6",Z4'6"
T160,Z6'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T165,Z7'0"
T170,Z7'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T180,KI,B54,Z6'0"
T190,X7'3",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T200,X12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T210,X12'0"
T220,X12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T230,KI,B54,X4'5"
T240,Z4'5",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T250,Z12'0"
T260,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T270,Z12'0"
T280,Z12'0",CG,TAG=,LTAG=LVL-0,STIFF=Rigid
T290,Z12'0"
T300,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T310,Z12'0"
T320,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T330,Z16'0",CG,TAG=,LTAG=LVL-0,STIFF=Rigid
T340,Z12'0"
T350,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T360,Z12'0"
T370,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T380,Z12'0"
T390,Z12'0",CG,TAG=,LTAG=LVL-0,STIFF=Rigid
T400,Z12'0"
T410,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T420,Z12'0"
T430,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T440,Z12'0"
T450,Z12'0",CG,TAG=,LTAG=LVL-0,STIFF=Rigid
T460,Z12'0"
T470,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T480,Z12'0"
T490,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T500,Z12'0"
T510,KA,Z12'0",CTAG=,LTAG=LVL-0
```

```
T520,Z12'0"
T530,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T540,Z12'0"
T550,Z12'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T560,KI,B54,Z4'5"
T570,X-4'5",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T580,X-12'0"
T590,X-11'7",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T600,X-12'0"
T610,X-12'0",CG,TAG=,LTAG=LVL-0,STIFF=Rigid
T620,X-12'0"
T630,KA,X-12'0",CTAG=,LTAG=LVL-0
T640,X-12'0"
T650,X-12'0",CG,TAG=,LTAG=LVL-0,STIFF=Rigid
T660,X-12'0"
T670,X-10'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T680,X-13'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T690,KI,B54,X-9'0"
T700,Z7'11",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T710,KA,Z12'0",CTAG=,LTAG=LVL-0
*Branch Pipe
F100
T720,KI,B54,P36O,Y10'9",CL=36O
T730,Z7'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T740,KI,B54,Z9'0"
T750,Y16'9",CVS=1,HTYP=27,MLV=25,TAG=,LTAG=LVL-0
T760,Y26'11",CVS=1,HTYP=27,MLV=25,TAG=,LTAG=LVL-0
T770,KI,B54,Y5'8"
T780,Z6'0",CTAG=,LTAG=LVL-
0,LS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T790,KA,Z12'0",CTAG=,LTAG=LVL-0
WIND1
0,0.6000,0.0000,0.0000,1.0000,0,0,0
5
0,15
15,15
30,15
45,15
60,15
```

Example MBF 6x Format:

An example of a model batch file (.mbf file) is shown below:

```
Desalter Pumps

OPTIONS
HGRA,B313
MATERIAL
A53,0.283,0.300,1.00
A53,-325,31.4E+6,5.00E-6,20000
A53,-200,30.8E+6,5.35E-6,20000
A53,-100,30.2E+6,5.65E-6,20000
A53,70,29.5E+6,6.07E-6,20000
A53,200,28.8E+6,6.38E-6,20000
A53,300,28.3E+6,6.60E-6,20000
A53,400,27.7E+6,6.82E-6,20000
A53,500,27.3E+6,7.02E-6,18900
A53,600,26.7E+6,7.23E-6,17300
A53,650,26.1E+6,7.33E-6,17000
A53,700,25.5E+6,7.44E-6,16500
A53,750,24.9E+6,7.54E-6,13000
A53,800,24.2E+6,7.65E-6,10800
A53,850,23.3E+6,7.75E-6,8700
A53,900,22.4E+6,7.84E-6,6500
A53,950,21.4E+6,7.91E-6,4500
A53,1000,20.4E+6,7.97E-6,2500
A53,1050,19.2E+6,8.05E-6,1600
A53,1100,18.0E+6,8.12E-6,1000
PIPE
4,4.5,0.237,0,12.5,11,2
6,6.625,0.28,0,12.5,11,2
8,8.625,0.322,0,12.5,11,2
10,10.75,0.365,0,12.5,11,2
12,12.75,0.406,0,12.5,11,2
LOADS
1,400,280,0.85,400,280,350,190
1N,400,280,0.85,100,280,350,190
1S,100,280,0.85,400,280,350,190
1C,100,280,0.85,100,280,100,190
4,400,600,0.85,400,600,285,450
4N,400,600,0.85,100,600,285,450
4S,100,600,0.85,400,600,285,450
SO1,350,190,0.85,70,0,70,0
SOA,230,145,0.85,70,0,70,0
SD1,400,280,0.85,70,0,70,0
DO1,285,450,0.85,70,0,70,0
DOA,225,485,0.85,70,0,70,0
DD1,400,600,0.85,70,0,70,0
LAYOUT
F10,CNOZZLE=297,L2=10'0”,NOD=17.5,NTHK=1.3125,VOD=144,VTHK=1.375,
L1=10'0”,E=29.0E+6,DV(0.0000,0.0000,1.0000)
L10,CFLANGE=WN,WGT=220

T20,JV,MA53,P10,Y1'6-1/4”,CL=1,VWGT=625,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=220
T30,Y0'4-5/8”
T40,JD,P8,Y0'7”,COD1=10.75,THK1=0.365,OD2=8.625,THK2=0.322
T50,KI,B12,Y6'0-1/8”
```

```
T60,KI,B12,X-1.0017,Z1.0017
T70,KI,B12,X-5'8-15/16
T80,KI,B12,Y-11'3”
T90,Z6'6”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T100,Z20'0”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T110,JI,B6.75,P4,Z17'6”,CJMAT=A53,JSEC=12,JLOAD=1,JR=19.125
T120,P8,X10'11”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T130,KI,B12,X2'0”
*
T240,KT,JE,Z6'4-9/16”,CES(100,400,300),TA=500,BK=11459
*
T160,X-2'0”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T170,KI,B12,X-8'0”
T180,Y-8'2-13/16,CFLANGE=WN,WGT=152
T190,JV,Y-1'4-3/4”,CVWGT=590,THKF=3.0000,INSF=1.7500,FLANGE=WN,
WGT=152
T200,KI,B12,Y-1'4-3/8”,CL=1N
T210,JS,Z-1'0”,CFFOR=1000
T220,JD,P6,Y0'0-15/16”,Z-0'6”,COD1=8.625,THK1=0.322,OD2=6.625,
THK2=0.28,FRE=30.000
L200A,CUS(1,900,2250)
*
F240
T250,KT,P8,Z1'2”,CL=1
*
T260,KT,Z1'2”
T270,Z0'11”
*
F260
T280,X-2'0”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T290,KI,B12,X-8'0”
T300,Y-8'2-13/16,CFLANGE=WN,WGT=96
T310,JV,Y-1'4-3/4”,CVWGT=590,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=152
T320,KI,B8,Y-1'4-3/8”,CL=1S
T330,Z1'0”
T340,JD,Y0'0-
15/16”,Z0'6”,COD1=8.625,THK1=0.322,OD2=6.625,THK2=0.28
*Node 230 = P-427B Suction
T350,KA,P6,Z1'5-3/16”,CDIS(0,0.027,0),DIS2(0,0.027,0),
DIS3(0,0.027,0)
L350,CFLANGE=WN,WGT=96
*Node 550 = P-427A Suction
F220
T390,KA,Z-2'7”,CL=1N,DIS(0,0.027,0),DIS3(0,0.027,0)
L390,CFLANGE=WN,WGT=96

T400,Z-1'6-3/8”
T410,JL,Z-1'5-3/8”,CTRAK(1,1,1),ROTK(1,1,1),
LXAX(1.0000,0.0000,0.0000),LYAX(0.0000,1.0000,0.0000)
*From node 200 2'11.5” in the Z direction
F370,KA,X-3.8300,Y-13'8”,Z59.6267,CDIS(0,0.027,0),
DIS2(0,0.027,0),DIS3(0,0.027,0)
L370,CFLANGE=WN,WGT=58
T420,P4,Z0'3-3/8”,CL=4S
T430,JD,P6,Z0'5-1/2”,COD1=4.5,THK1=0.237,OD2=6.625,THK2=0.28
T440,KI,B9,Z2'0-5/8”
```

```
T450,Y1'0-7/8”,CFLANGE=WN,WGT=96
T460,JV,Y1'5-5/8”,CVWGT=204,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=96
T470,JV,Y1'4-1/8”,CVWGT=334,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=96
T480,KI,B9,Y7'0-3/8”
T490,X8'0”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T500,KT,X3'6”
*
*From node 340 -2'11.5” in the -Z direction
F410,KA,CDIS(0,0.027,0),DIS3(0,0.027,0)
L410,CFLANGE=WN,WGT=58
T510,P4,Z-0'3-3/8”,CL=4N
T520,JD,P6,Z-0'5-1/2”,COD1=4.5,THK1=0.237,OD2=6.625,THK2=0.28
T530,KI,B9,Z-4'0-5/8”
T540,Y1'0-7/8”,CFLANGE=WN,WGT=96
T550,JV,Y1'5-5/8”,CVWGT=204,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=96
T560,JV,Y1'4-1/8”,CVWGT=334,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=96
T570,KI,B9,Y7'0-3/8”
T580,X8'0”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T590,KI,B9,X3'6”
T500
L530B,CUS(1,520,1392)
F500
T600,KI,B9,Z1'7”,CL=4
T610,X-3'6”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T620,KI,B9,X-10'11”
T630,KI,B9,Y-4'6-11/16
T640,KI,B9,Z7'4”
T650,Y1'1”,CFLANGE=WN,WGT=96
T660,Y0'11”
T670,JD,P8,Y0'6”,COD1=6.625,THK1=0.28,OD2=8.625,THK2=0.322
T680,KT,Y0'7”
*
F250
T685,P4,X-2'0”,CL=1C,LS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T690,KI,B6,X-10'11”
T700,KI,B6,Y-1'5-11/16
T710,Z1'10”,CFLANGE=WN,WGT=58

*
F680
T720,Z-1'6”,CL=4,FLANGE=WN,WGT=58
T730,JV,Z-1'0-1/4”,CVWGT=170,THKF=3.0000,
INSF=1.7500,FLANGE=WN,WGT=58
T735,Z-6'6-3/4”,CL=1C,LS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T710,CL=1
*
F680
T800,KI,B12,P8,Y3'0”,CL=4
T810,Z6'1-5/8”,CFLANGE=WN,WGT=152
T820,Z0'4-3/8”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),
STIFF=Rigid
T830,KI,B12,Z1'0”
```

```
T840,KI,B12,X7'0”
T850,KI,B12,X1'6”,Y-1'6”
T860,X6'0”,CFLANGE=WN,WGT=152
T870,JV,X1'4-3/4”,CVWGT=410,THKF=3.0000,INSF=1.7500,
FLANGE=WN,WGT=152
T880,KT,X1'4”
*
T890,KI,B12,Y1'7”
T900,KI,B12,Z2'0”
T910,Y-1'1-1/4”,CFLANGE=WN,WGT=152
T920,JV,Y-1'4-3/4”,CVWGT=410,THKF=3.0000,INSF=1.7500,FLANGE=WN,WGT=152
T930,KI,B12,Y-1'1-1/4”
T940,KT,Z-2'0”
*
T950,X1'0”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T960,X1'0”
*
F940
T970,X-1'4”
*
F880
T980,KI,B12,X10'6”
T990,KI,B12,Y4'6”
T1000,X1'10”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1010,X26'8”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1020,KI,B12,X6'2”
T1030,KI,B12,Z2'0”
T1040,Y4'2”
T1050,JD,P10,Y0'7”,COD1=6.625,THK1=0.28,OD2=8.625,THK2=0.322
T1060,KI,B15,Y1'3”
T1070,Z7'2”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1080,Z18'8”,CLS(0.000,None),DV(0.0000,1.0000,0.0000),STIFF=Rigid
T1090,KI,B15,Z15'2”
T1100,KI,B15,Y-25'6”
T1110,KI,B15,X-3.8971,Z-2'3”
T1120,Y1'7-7/8”,CFLANGE=WN,WGT=220

PUMPS
427BSOUTH,1,350
427ANORTH,0,0.0000,1.0000,0.0000,3.8299,13'8”,44.3351,390,2,410,2
COMPRESSORS
Compressor_1,1.0000,0.0000,0.0000,210,300
SEISMIC
0.25,0.00,0.30
WIND
100,0.600,1.000,0.000,0.000
```

Hanger Type

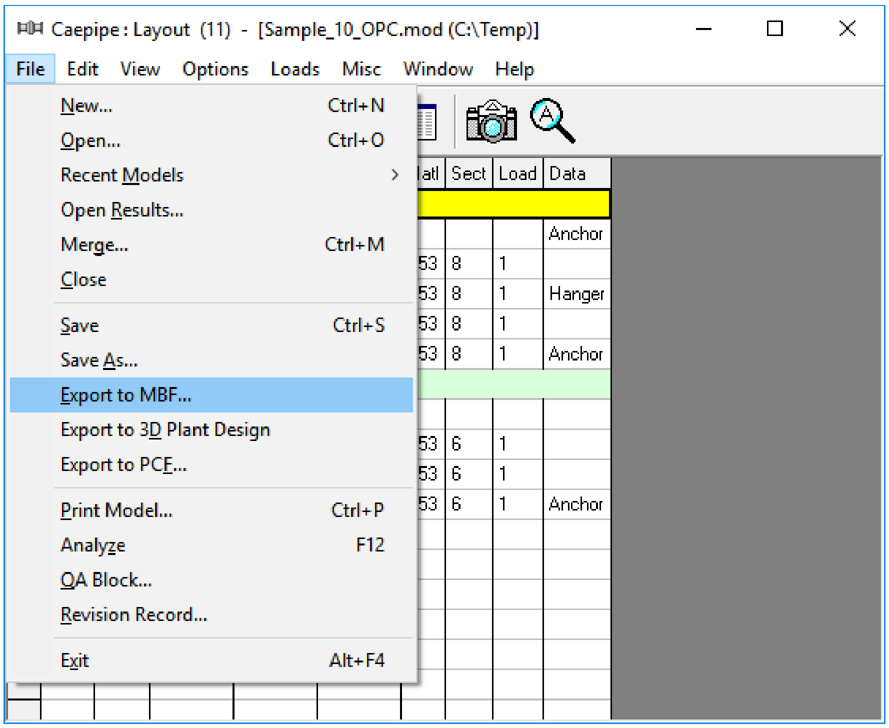
This section describes the hanger type (HTYP under comments section) number to be used for defining variable spring hanger type.

Hanger Description Type	Value
ABB-PBS	1
Basic Engineers	2
Bergen Paterson	3
Bergen Paterson (L)	4
BHEL Hyderabad	5
BHEL Trichy	6
Borrello	7
Carpenter & Paterson	8
Corner & Lada	9
Dynax	10
Elcen	11
Fee and Mason	12
Flexider (30-60-120)	13
Flexider (50-100-200)	14
Fronek	15
Grinell	16
Hydra	17
Lisega	18
Mitsubishi (30-60-120)	19
Mitsubishi (80-160)	20
Myricks	21
NHK (30-60-120)	22
NHK (80-160)	23
Nordon	24
NPS Industries	25
Piping Services	26
Piping Tech & Products	27
Power Piping	28
Sanwa Tekki (30-60-120)	29
Sanwa Tekki (85-170)	30
Sarathi	31
Spring Supports	32
SSG	33
Comet	34
Gradior	35

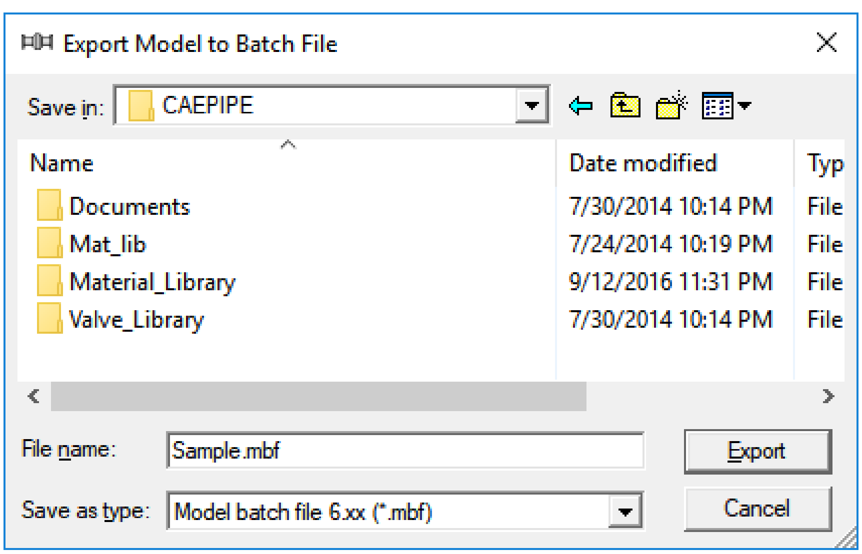
MBF Export

CAEPIPE can export model data from inside the Layout window to a text file with the extension .mbf (model batch file), which may be edited using a text editor. The edited text file may then be read back into CAEPIPE by using the Import feature. But note that the existing .mod file will be overwritten. So, you must rename the edited text file before importing it back into CAEPIPE.

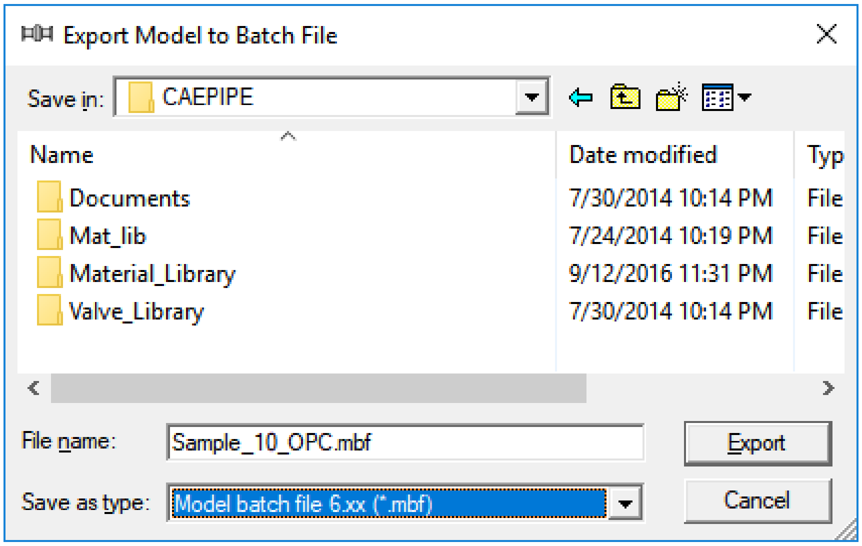
To export a model, select the menu command File > Export from the Layout window.



The Export Model dialog is shown. Selecting the "Save as type" option as "Model batch file (*.mbf)" will export the stress model in CAEPIPE latest version. Once done, click on Export button to write to an .mbf file.



Select the "Save as type" as "Model batch file 6.xx (*.mbf)" to export the stress model in 6.xx format. This will allow the user to have the backward compatibility with CAEPIPE Versions 6.81 through 10.00.



This section brings out the details on how each element and data from CAEPIPE are exported to .pcf (Piping Component File) file.

Vertical Axis

The stress layout developed in CAEPIPE with Y axis vertical is automatically translated to be Z axis as vertical when the layout is exported to PCF file.

Limitations

The Element types and Data types listed below are not transferred to PCF at this time.

Element Types

1. Ball Joint
2. Beam
3. Elastic Element
4. Hinge Joint
5. Tie Rod
6. Comment and
7. Hydrotest Load

Data Types

1. Concentrated Mass
2. Force
3. Harmonic Load
4. Jacket End Cap
5. Spider
6. Threaded Joint
7. Time Varying Load
8. User SIF and
9. Weld

Units

The stress system will be exported to PCF in SI units, when the Length unit selected in CAEPIPE for a stress layout is “m” or “mm”. Otherwise, the stress layout will be exported to PCF in English units. The table below provides the details on English and SI units used while exporting to PCF.

Sl. No.	Description of Units	English Units	SI Units
1.	Length	inch	mm
2.	Dimension	inch	mm
3.	Angle	degree	degree
4.	Weight	lbf	kgf
5.	Density	lbf/in3	Kgf/m3
6.	Temperature	deg. F	deg. C
7.	Pressure	psi	bar
8.	Stiffness	lb/inch	N/mm
9.	Rotational Stiffness	in-lb/deg	Nm/deg

Basic Header Information

The Basic Header Information attributes that defines the control file identifier and various Units that are used to specify Bores, Co-ordinates, Bolt Diameters, Bolt Lengths and Weights while exporting PCF file are provided below for both English and SI Units.

ISOGEN-FILES ISOGEN.FLS
UNITS-BORE INCH / MM
UNITS-CO-ORDS INCH / MM
UNITS-WEIGHT LBS/KGS
UNITS-BOLT-DIA INCH/MM
UNITS-BOLT-LENGTH INCH/MM

In addition to the above, the layout of stress system is always exported to PCF as a System Isometric using the identifier "SYSTEM-ISOMETRIC-REFERENCE" as the stress system may generally contain one or more individual pipelines that are physically connected in such a way that they form a network.

Additional optional attributes which relate to the "System" are not written to the PCF file at this time.

Pipeline Header Information

Each "From" node defined in CAEPIPE is exported to PCF as a PIPELINE-REFERENCE attribute with its value set to "P" followed by the From Node number used in CAEPIPE.

The Temperature (T1) and Pressure (P1) defined in CAEPIPE for the element followed by the From node is written to PCF using the attributes "PIPELINE-TEMP" and ATTRIBUTE1 respectively in English or SI units as shown below.

For example, a From Node 10 in CAEPIPE with Temperature (T1 = 200 C) and Pressure (P1 = 10 bar) of the element following the "From" node is exported to PCF as

PIPELINE-REFERENCE P10
PIPELINE-TEMP 200
ATTRIBUTE1 10

Component Identifiers

Each piping component from CAEPIPE is exported to PCF in a self-contained data block that consists of a "Component Identifier" together with a list of attributes that help to identify component location, size and specific requirements related to a physical component in a pipeline.

This section provides in details on how each piping component (element) from CAEPIPE is transferred to PCF along with their list of attributes. Each component from CAEPIPE is exported to PCF with Mandatory attributes, Material Information Attributes and Supplementary Information Attributes. The supplementary information attributes of each component will have eight (8) COMPONENT-ATTRIBUTES as shown in the table below.

Sl. No.	Component Attribute in PCF	CAEPIPE Attribute	English Units	SI Units
1.	COMPONENT-ATTRIBUTE1	Section Outer Diameter	'inch'	'mm'
2.	COMPONENT-ATTRIBUTE2	Section Wall Thickness	'inch'	'mm'
3.	COMPONENT-ATTRIBUTE3	Section Corrosion Allowance	'inch'	'mm'
4.	COMPONENT-ATTRIBUTE4	Section Mill tolerance	%	%
5.	COMPONENT-ATTRIBUTE5	Section Insulation Density	'lbf/in3'	'kgf/m3'
6.	COMPONENT-ATTRIBUTE6	Section Insulation Thickness	'inch'	'mm'
7.	COMPONENT-ATTRIBUTE7	Temperature T1	'deg. F'	'deg. C'
8.	COMPONENT-ATTRIBUTE8	Pressure P1	'psi'	'bar'

The co-ordinates for all components and supports are transferred in 'inch' for English Units and 'mm' for SI Units. Similarly, size attribute is transferred in 'inch' for English Units and 'mm' for SI Units. The material type and grade for each component is transferred to PCF through ITEM CODE.

Element types from CAEPIPE

Pipe

Pipe element from CAEPIPE is transferred to PCF as "PIPE" along with their attributes as shown below.

The absolute co-ordinate corresponding to "From" and "To" node is written to "END-POINT" attribute.

For standard pipe sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) defined for the element via section property is written to "Size" attribute in "inch" for English Units and "mm" for SI Units. For non-standard pipe sizes, OD defined for the element via section property is written to "Size" attribute.

The material properties (Name and Grade) defined for the element via "Material" property is written to PCF using the "ITEM-CODE" attribute.

Section properties of pipe element such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to "COMPONENT-ATTRIBUTE1" through "COMPONENT-ATTRIBUTE6" as explained in the Table above.

Temperature T1 and Pressure P1 defined for the pipe element via "Load" property is written to "COMPONENT-ATTRIBUTE7" and "COMPONENT-ATTRIBUTE8" respectively as explained in the Table above.

The element properties other than those listed above are ignored and not transferred to PCF at this time.

Component Identifier

PIPE

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data

Jacketed Pipe

The Core Pipe of Jacketed Pipe element of CAEPIPE is transferred to PCF as “PIPE” along with their attributes as explained above. The Jacketed Pipe details are not transferred to PCF at this time as there is no provision available in PCF.

Bend

Bend element from CAEPIPE is transferred to PCF as “ELBOW” along with their attributes as shown below.

The co-ordinate corresponding to “Near” and “Far End” nodes (referred in CAEPIPE as Node number suffixed with A and B) are written to “END-POINT” attributes. The co-ordinate corresponding to “TIP” from CAEPIPE is written to “CENTRE-POINT” attribute of PCF.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) defined for the element via section property is written to “Size” attribute in “inch” for English Units and “mm” for SI Units. For non-standard component sizes, OD defined for the element via section property is written to “Size” attribute.

The material properties (Name and Grade) defined for the element via “Material” property is written to PCF using the “ITEM-CODE” attribute.

The value of SKEY is written as “ELBW”.

Section properties of element (defined via section property) such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to “COMPONENT-ATTRIBUTE1” through “COMPONENT-ATTRIBUTE6” as explained above.

Temperature T1 and Pressure P1 defined for the bend element via “Load” property is written to “COMPONENT-ATTRIBUTE7” and “COMPONENT-ATTRIBUTE8” respectively as explained above.

Bend radius [inch/mm] and Angle [deg] defined for bend element in CAEPIPE are transferred to "BEND-RADIUS" and "BEND-ANGLE" respectively.

The properties other than those explained above are ignored and not transferred to PCF at this time.

Component Identifier

BEND

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
CENTRE-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	
SKEY ELBW	

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data
BEND-RADIUS	data
BEND-ANGLE	data

Miter Bend

Miter Bend element from CAEPIPE is transferred to PCF as "ELBOW" along with their attributes as explained above. In addition, the value of SKEY is written as "BEBW" instead of "ELBW".

Jacketed Bend

The Core Bend of Jacketed Bend element of CAEPIPE is transferred to PCF as "ELBOW" along with their attributes as explained above. The Jacketed Bend details are not transferred to PCF at this time as no provision is available in PCF.

Valve

Valve element from CAEPIPE is transferred to PCF as "VALVE" along with their attributes as shown below.

The co-ordinate corresponding to "From" and "To" node is written to "END-POINT" attributes.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) defined for the valve element via section property is written to "Size" attribute in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD defined for the valve element via section property is written to "Size" attribute.

The material properties (Name and Grade) defined for the valve element via "Material" property is written to PCF using the "ITEM-CODE" attribute.

The value of SKEY is written as "VG**" for Gate and Globe Valve, "CK**" for Check Valve, "ZB**" for Butterfly Valve, "VC**" for Control Vale, "VB**" for Ball Valve and "VP**" for Plug Valve. For valve types other than those explained above, the SKEY is written as "VS**".

Section properties of valve element (defined via section property) such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to "COMPONENT-ATTRIBUTE1" through "COMPONENT-ATTRIBUTE6" as explained above.

Temperature T1 and Pressure P1 defined for the valve element via "Load" property is written to "COMPONENT-ATTRIBUTE7" and "COMPONENT-ATTRIBUTE8" respectively as explained above.

Empty weight of valve is transferred to "WEIGHT" attribute in 'lbf' for English Units and 'kgf' for SI Units.

The valve properties other than those explained above are ignored and not transferred to PCF at this time.

Component Identifier

VALVE

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	data

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data
WEIGHT	data

Reducer

Reducer element from CAEPIPE is transferred to PCF as "REDUCER-CONCENTRIC" along with their attributes as shown below.

The co-ordinate corresponding to "From" and "To" node is written to "END-POINT" attributes.

For standard component sizes, the Nominal Sizes (NS) corresponding to Outer Diameter 1 (OD1) and Outer Diameter 2 (OD2) of Reducer element are written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD1 and OD2 of reducer element are written to "Size" attributes.

The material properties (Name and Grade) defined for the reducer element via "Material" property is written to PCF using the "ITEM-CODE" attribute.

The value of SKEY is written as "RCBW".

Section properties defined for the Reducer element (via section property) such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to "COMPONENT-ATTRIBUTE1" through "COMPONENT-ATTRIBUTE6" as explained above.

Temperature T1 and Pressure P1 defined for the valve element via "Load" property is written to "COMPONENT-ATTRIBUTE7" and "COMPONENT-ATTRIBUTE8" respectively as explained above.

The reducer properties other than those explained above are ignored and not transferred to PCF at this time.

Component Identifier

REDUCER-CONCENTRIC

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	RCBW

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data

Bellows / Slip Joint / Cut-Pipe

Bellow / Slip Joint / Cut-pipe from CAEPIPE are transferred to PCF as "MISC-COMPONENT" along with their attributes as shown below.

The co-ordinate corresponding to "From" and "To" node of the element is written to "END-POINT" attributes.

The mid-point computed using the "From" and "To" node of the element is written to "CENTRE-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element is written to "Size" attribute.

The material properties (Name and Grade) defined for the element via "Material" property is written to PCF using the "ITEM-CODE" attribute.

The SKEY attribute is written as EXPJ, SLIP and CUTP for Bellows, Slip Joint and Cut-Pipe respectively.

Section properties defined for the element (via section property) such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to "COMPONENT-ATTRIBUTE1" through "COMPONENT-ATTRIBUTE6" as explained above.

Temperature T1 and Pressure P1 defined for the element via "Load" property is written to "COMPONENT-ATTRIBUTE7" and "COMPONENT-ATTRIBUTE8" respectively as explained above.

Empty weight of bellow is transferred to "WEIGHT" attribute in 'lbf' for English Units and 'kgf' for SI Units.

The properties other than those explained above are ignored and not transferred to PCF at this time for Bellows / Slip Joint / Cut-Pipe.

Component Identifier

MISC-COMPONENT

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
CENTRE-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	EXPJ / SLIP / CUTP

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data
WEIGHT	data [only for Bellows]

Flange

Rigid element (with length < OD) and Flange from CAEPIPE is transferred to PCF as "FLANGE" along with their attributes as shown below.

For rigid element, the co-ordinate corresponding to "From" and "To" node is written to "END-POINT" attributes. On the other hand, for flange, the co-ordinate corresponding to flange node is written to "END-POINT" attributes. As the length of flange is zero in CAEPIPE (being a nodal property), both "END-POINT" attributes will have the same values.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the rigid element / flange defined via section property is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the rigid element / flange is written to "Size" attribute.

The material properties (Name and Grade) defined for the rigid element / flange via "Material" property is written to PCF using the "ITEM-CODE" attribute.

The SKEY attribute is written as "FL**".

Section properties defined for the rigid element / flange (via section property) such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to "COMPONENT-ATTRIBUTE1" through "COMPONENT-ATTRIBUTE6" as explained above.

Temperature T1 and Pressure P1 defined for the rigid element / flange via "Load" property is written to "COMPONENT-ATTRIBUTE7" and "COMPONENT-ATTRIBUTE8" respectively as explained above.

Empty weight of rigid element / flange is transferred to "WEIGHT" attribute in 'lbf' for English Units and 'kgf' for SI Units.

The properties other than those explained above are ignored and not transferred to PCF at this time

Component Identifier

FLANGE

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	FL**

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data
WEIGHT	data

Rigid

Rigid element from CAEPIPE is transferred to PCF as "INSTRUMENT" along with their attributes as shown below when the length of the rigid element is greater than the outer diameter (OD) of the element. On the other hand, when the length is less than OD of the element, then the same is transferred as "FLANGE" to PCF as explained above.

The co-ordinate corresponding to "From" and "To" node of the element is written to "END-POINT" attributes.

The mid-point computed using the "From" and "To" node of the element is written to "CENTRE-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element is written to "Size" attribute.

The material properties (Name and Grade) defined for the element via "Material" property is written to PCF using the "ITEM-CODE" attribute.

The SKEY attribute is written as "INST".

Section properties defined for the element (via section property) such as OD, Wall Thickness, Corrosion Allowance, Mill tolerance, Insulation Density and Insulation Thickness are written to "COMPONENT-ATTRIBUTE1" through "COMPONENT-ATTRIBUTE6" as explained above.

Temperature T1 and Pressure P1 defined for the valve element via "Load" property is written to "COMPONENT-ATTRIBUTE7" and "COMPONENT-ATTRIBUTE8" respectively as explained above.

Empty weight of rigid element is transferred to "WEIGHT" attribute in 'lbf' for English Units and 'kgf' for SI Units.

The properties other than those explained above are ignored and not transferred to PCF at this time.

Component Identifier

INSTRUMENT

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
CENTRE-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	INST

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data
COMPONENT-ATTRIBUTE5	data
COMPONENT-ATTRIBUTE6	data
COMPONENT-ATTRIBUTE7	data
COMPONENT-ATTRIBUTE8	data
WEIGHT	data

Data Types from CAEPIPE

Anchor

Anchor from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below. The co-ordinate value corresponding to Anchor node is written to "END-POINT" attribute.

The SKEY attribute is written as "ANCH".

The stiffnesses and specified displacements defined in CAEPIPE for Anchor are ignored at this time.

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	ANCH

Branch SIF

Branch SIF with their sub-types are transferred from CAEPIPE to PCF as given below.

Sl. No.	Sub-Type of Branch SIF	Component Identifier in PCF
1.	Welding TEE	TEE
2.	Reinforced Fabricated Tee	TEE-SET-ON
3.	Unreinforced Fabricated Tee	TEE-SET-ON
4.	Others	TEE-STUB

As Branch SIF in CAEPIPE is assigned to a node, the co-ordinate value corresponding to "Branch SIF" node is written to "END-POINT", "CENTRE-POINT" and "BRANCH1-POINT" attributes.

For standard component sizes, the Nominal Size (NS) corresponding to Run Pipe Outer Diameter (OD1) and Branch Pipe Outer Diameter (OD2) of the element is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD1 and OD2 of the element are written to "Size" attributes.

The material properties (Name and Grade) defined for the element via "Material" property on which the Branch SIF is located is written to PCF using the "ITEM-CODE" attribute.

Component Identifier

TEE

Mandatory Attributes

END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
END-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
CENTRE-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	
BRANCH1-POINT	*E/W co-ords*	*N/S co-ords*	*Elevation co-ords*	*Size*

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	TEBW

Reinforced / Unreinforced Fabricated Tee

Component Identifier

TEE-SET-ON

Mandatory Attributes

BRANCH1-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
CENTRE-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	TESO

Other Tees

Component Identifier

TEE-STUB

Mandatory Attributes

BRANCH1-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	Size
CENTRE-POINT	E/W co-ords	N/S co-ords	Elevation co-ords	

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	TSSO

Guide

Guide from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Guide node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the guide is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The Friction, Stiffness [lb/in or N/mm] and Gap [in or mm] defined at Guide are transferred to "COMPONENT-ATTRIBUTE1", "COMPONENT-ATTRIBUTE2" and "COMPONENT-ATTRIBUTE3" respectively.

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	GUI

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data

Hanger

Hanger from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Hanger node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the hanger is located is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The SKEY attribute is written as "SPRG".

The hanger type and number of hangers entered in CAEPIPE are transferred to "COMPONENT-ATTRIBUTE1" and "COMPONENT-ATTRIBUTE2 respectively.

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	SPRG

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data

User Hanger

User Hanger from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to User Hanger node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the user hanger is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The SKEY attribute is written as "HANG".

The Spring Stiffness [lb/in or N/mm], Number of hangers, Load [lb or N] and load type [HOT or COLD] defined at User Hanger are transferred to "COMPONENT-ATTRIBUTE1", "COMPONENT-ATTRIBUTE2", "COMPONENT-ATTRIBUTE3" and "COMPONENT-ATTRIBUTE4" respectively.

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	HANG

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data

Rod Hanger

Rod Hanger from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Rod Hanger node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the rod hanger is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The SKEY attribute is written as "ROD".

The Number of hangers defined at Rod hanger of CAEPIPE is transferred to "COMPONENT-ATTRIBUTE1".

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	ROD

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data

Constant Support

Constant Support from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Constant Support node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the constant support is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The SKEY attribute is written as "CS".

The Number of hangers defined at this support in CAEPIPE is transferred to "COMPONENT-ATTRIBUTE1".

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	CS

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data

Limit Stop

Limit Stop from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Limit Stop node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the limit stop is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The Upper Limit [in or mm], Lower Limit [in or mm], Stiffness [lb/in or N/mm] and Friction defined at Limit Stop are transferred to "COMPONENT-ATTRIBUTE1", "COMPONENT-ATTRIBUTE2", "COMPONENT-ATTRIBUTE3" and "COMPONENT-ATTRIBUTE4" respectively. The value of SKEY attribute is written as LSX when the direction is defined as (1.0, 0.0, 0.0), LSY when the direction is defined as (0.0,1.0, 0.0), LSZ when the direction is defined as (0.0, 0.0, 1.0) and LIM for other directions.

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	LIM / LSX /LSY / LSZ

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data
COMPONENT-ATTRIBUTE2	data
COMPONENT-ATTRIBUTE3	data
COMPONENT-ATTRIBUTE4	data

Nozzle

Nozzle from CAEPIPE is transferred to PCF as "Nozzle" along with the attributes as shown below.

The co-ordinate value corresponding to Nozzle node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the nozzle is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

Component Identifier

NOZZLE

Mandatory Attributes

CO-ORDS E/W co-ords N/S co-ords Elevation co-ords Size

Restraint

Restraint from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Restraint node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the restraint is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

The SKEY value will be filled with type of "Restraint" defined in CAEPIPE. For example, X and Z defined at a "Restraint" in CAEPIPE will be written to SKEY of PCF as "XZ"

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS E/W co-ords N/S co-ords Elevation co-ords Size

Material Information Attributes

ITEM-CODE data
FABRICATION-ITEM data
SKEY data

Skewed Restraint

Skewed Restraint from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Skewed Restraint node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the skewed restraint is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

Translational Stiffness [lb/in or N/mm] or Rotational Stiffness [in-lb/deg or N-m/deg] defined in CAEPIPE will be transferred to COMPONENT-ATTRIBUTE1 of PCF. The value of SKEY is filled as TX, TY and TZ for Translational Restraint defined in X, Y and Z directions respectively. Similarly, for Rotational Restraint, the value of SKEY is filled as RX, RY and RZ for Rotational Restraint defined in X, Y and Z directions respectively. For Translational and Rotational Restraint defined in directions other than X, Y and Z, the value of SKEY is filled as "SKEW".

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	TX/TY/TZ / RX/RY/RZ/SKEW

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data

Snubber

Snubber from CAEPIPE is transferred to PCF as "Support" along with the attributes as shown below.

The co-ordinate value corresponding to Snubber node is written to "END-POINT" attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the snubber is defined is written to "Size" attributes in "inch" for English Units and "mm" for SI Units. For non-standard component sizes, OD of the element are written to "Size" attribute.

Stiffness [lb/in or N/mm] defined in CAEPIPE will be transferred to COMPONENT-ATTRIBUTE1 of PCF. The value of SKEY is filled as XSNB, YSNB and ZSNB for snubber defined in X, Y and Z directions respectively. For direction of snubber other than X, Y and Z, the value is written as "SNUB".

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS	E/W co-ords	N/S co-ords	Elevation co-ords	Size

Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	XSNB / YSNB / ZSNB /SNUB

Supplementary Information Attributes

COMPONENT-ATTRIBUTE1	data

Generic Support

Generic Support from CAEPIPE is transferred to PCF as “Support” along with the attributes as shown below.

The co-ordinate value corresponding to Generic Support node is written to “END-POINT” attribute.

For standard component sizes, the Nominal Size (NS) corresponding to Outer Diameter (OD) of the element in which the generic support is defined is written to “Size” attributes in “inch” for English Units and “mm” for SI Units. For non-standard component sizes, OD of the element are written to “Size” attribute.

The stiffnesses defined in CAEPIPE for Generic Support are ignored at this time

Component Identifier

SUPPORT

Mandatory Attributes

CO-ORDS E/W co-ords N/S co-ords Elevation co-ords Size

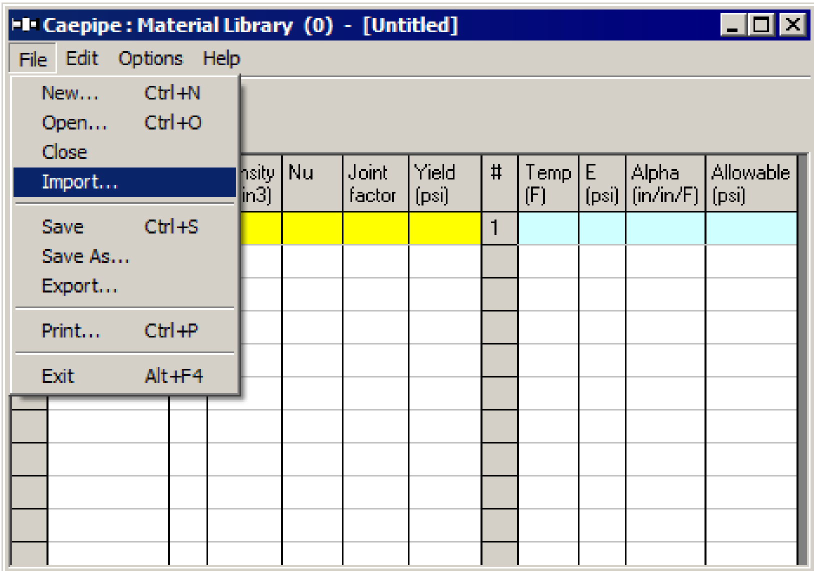
Material Information Attributes

ITEM-CODE	data
FABRICATION-ITEM	data
SKEY	GNSP

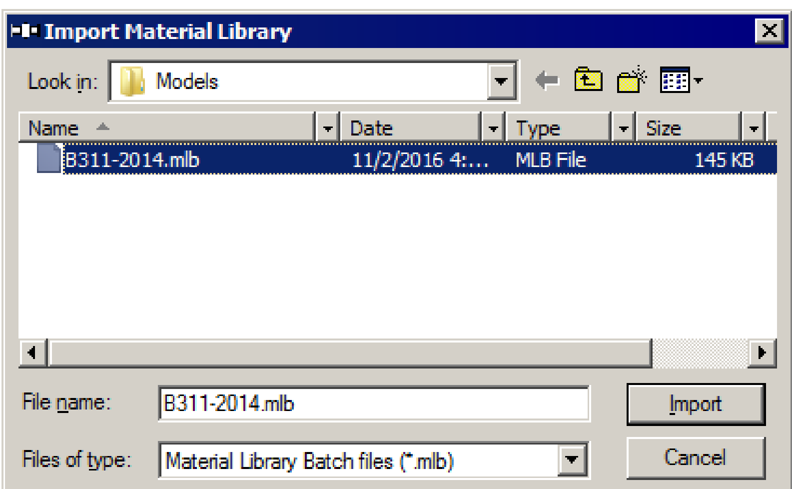
Import Material Library

CAEPIPE can import material properties created using a text file (batch file) into the Material Library. The text file may be created using a text editor and should have the extension: .mlb (material library batch file). The text file may also be created for an existing material library using the Export command from the Material Library window.

To import a material properties into the material library, select the menu command File > New > Material Library. From the Material Library window > File > Import…



The Import Material Library dialog is shown.



Select the material library batch file (.mlb) and then click on the Import button. The batch file will be read and the material properties thus imported are then shown in the Layout window, which can be further modified.

The input data is given in the following order. The start of each section is indicated by a keyword. The data for that section follow. Only the first five characters of the keyword are significant.

Format of Material Library Batch file (.mlb) is given below.

OPTIONS

Piping Code, Units

Piping Codes that can be entered are given below. Example, enter B311 to define the Piping Code as "ASME B31.1".

Units can be "SI" or "English". If this field is left blank, then CAEPIPE will set the default units as "English"

OPTIONSExample

```
OPTIONS
B311,SI
```

Piping codes

B311	ASMEB31.1
B311-67	USASB31.1(1967)
B313	ASME B31.3
B314	ASME B31.4
B315	ASME B31.5
B318	ASME B31.8
B319	ASME B31.9
B3112	ASME B31.12
ASME	ASMESectionIII,Class2(1980)
ASME-86	ASMESectionIII,Class2(1986)
ASME-92	ASME Section III, Class 2 (1992)
ASME-2015	ASME Section III, Class 2 (2015)
ASME-2017	ASME Section III, Class 2 (2017)
ASME-ND-2017	ASME Section III, Class 3 (2017)
BS806	Britishcode
NORWEGIAN-83	Norwegiancode (1983)
NORWEGIAN-90	Norwegiancode (1990)
RCC-M	Frenchcode (1985)
SNCT	CODETI (1995)
SWEDISH	Swedish code (1978)
STOOMWEZEN	Dutchcode (1989)
Z183	Z183 (1990)
Z184	Z184 (1992)
Z662	Z662
EUROPEAN	EN 13480

MATERIAL

English units

First line: Description (32 Characters), Density (lbf/in3), Poisson's ratio, [Long. joint factor], [circ. joint factor], [material type], [tensile strength (psi)]

Following lines:

Description, Temp (F), E (psi), alfa (in/in/F), [allowable stress (psi)], [yield stress (psi)], [rupture stress (psi)], [hoop modulus (psi)], [shear modulus (psi)]

.

.

SIunits

First line: Description (32 Characters), Density (kgf/m3), Poisson's ratio, [Long. joint factor], [circ.joint factor], [material type], [tensile strength (N/mm2)]

Following lines: Description, Temp (C), E (N/mm2), alfa (mm/mm/C), [allowable stress (N/mm2)], [yield stress (N/mm2)], [rupture stress (N/mm2)], [hoop modulus (N/mm2)], [shear modulus (N/mm2)]

.

.

MATERIALExample(SIunits)

```
OPTIONS
EUROPEAN,SI
MATERIAL
EN 1.0345 (P235GH) max 60 mm,7850,0.300,1.00,1.00,CSS,360.0
EN 1.0345 (P235GH) max 60 mm,20,212000,11.90E-6,120.0,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,50,209500,12.20E-6,120.0,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,100,207000,12.50E-6,120.0,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,150,203000,12.75E-6,120.0,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,200,199000,13.00E-6,113.0,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,250,195500,13.30E-6,100.0,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,300,192000,13.60E-6,86.70,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,350,188000,13.85E-6,80.00,0.000,0.000,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,400,184000,14.10E-6,74.70,0.000,94.00,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,410,183100,14.14E-6,74.16,0.000,85.30,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,420,182200,14.18E-6,73.62,0.000,76.00,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,430,181300,14.22E-6,66.70,0.000,66.70,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,440,180400,14.26E-6,58.70,0.000,58.70,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,450,179500,14.30E-6,51.30,0.000,51.30,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,460,178600,14.34E-6,44.00,0.000,44.00,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,470,177700,14.38E-6,37.30,0.000,37.30,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,480,176800,14.42E-6,31.30,0.000,31.30,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,490,175900,14.46E-6,26.00,0.000,26.00,0.000,0.000
EN 1.0345 (P235GH) max 60 mm,500,175000,14.50E-6,21.30,0.000,21.30,0.000,0.000
```

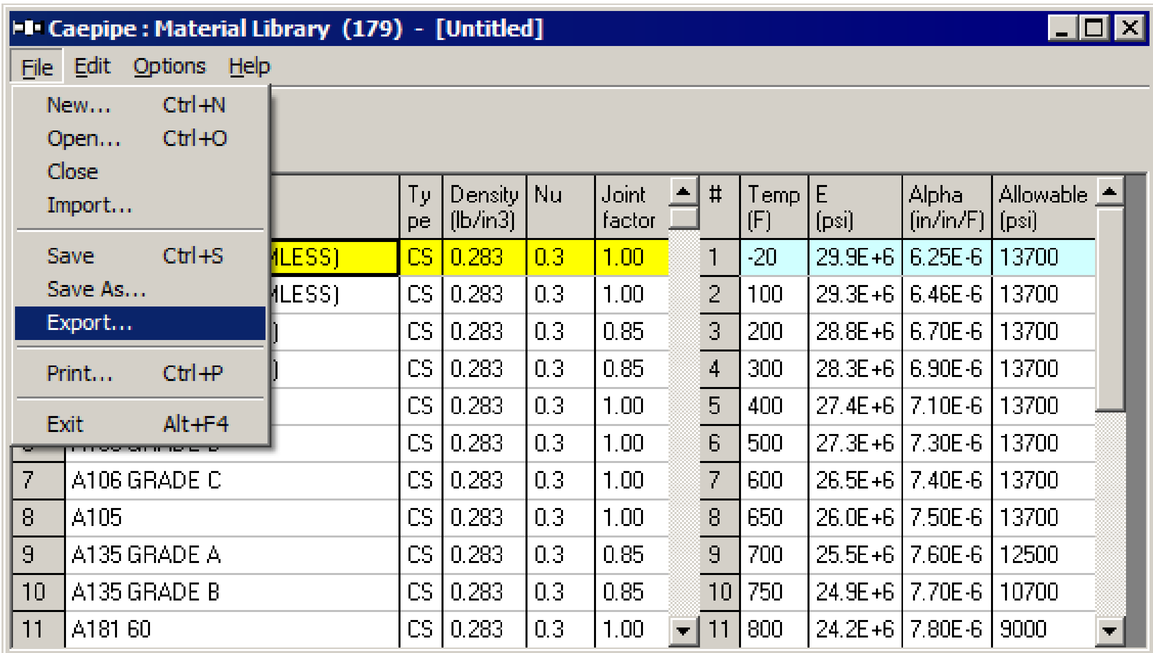
MATERIALExample(Englishunits)

```
OPTIONS
B311
MATERIAL
A53 GRADE A (SEAMLESS),0.283,0.300,1.00,1.00,CS,29994
A53 GRADE A (SEAMLESS),-20,29.9E+6,6.25E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),100,29.3E+6,6.46E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),200,28.8E+6,6.70E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),300,28.3E+6,6.90E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),399.9,27.4E+6,7.10E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),500,27.3E+6,7.30E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),600.1,26.5E+6,7.40E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),649.9,26.0E+6,7.50E-6,13700,0,0,0,0
A53 GRADE A (SEAMLESS),700,25.5E+6,7.60E-6,12499,0,0,0,0
A53 GRADE A (SEAMLESS),750,24.9E+6,7.70E-6,10699,0,0,0,0
A53 GRADE A (SEAMLESS),800.1,24.2E+6,7.80E-6,9000,0,0,0,0
```

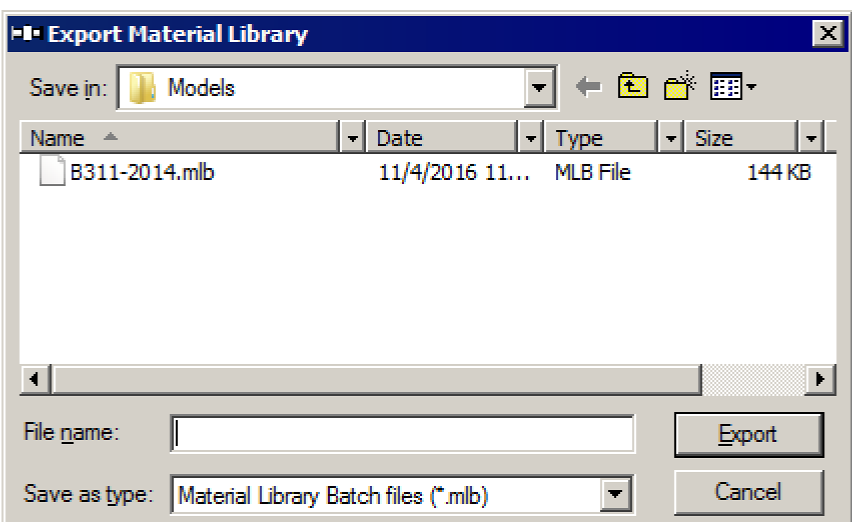
Export Material Library

CAEPIPE can export material properties to a batch text file with the extension .mlb (material library batch file). The text file may be edited using a text editor / using MS Excel. The edited text file may be read back into CAEPIPE by using the Import feature.

To export the material properties, select the menu command File > Export from the Material Library Layout window.



The Export Material Library dialog is shown. Click on the Export button to write to the batch file.



Appendix B

Response Spectrum Libraries

Response Spectrum Libraries

CAEPIPE provide fourteen (14) response spectra for your convenience. These spectra are stored in 3 separate Response Spectrum Library files under the directory "Spectrum_Library". These fourteen (14) spectra are added as per the following.

- EL Centro May 18, 1940
- Uniform Building Code (UBC) 1991 Edition and
- Nuclear Regulatory Commission (NRC) Guide 1.60

EL Centro

This spectrum data can be accessed by selecting the file "ELCentro_NS_May18_1940.spe" available in the folder "SpectrumLibrary" through Layout window >Misc> Spectrums > File > Library.

This predefined data is taken from "J'Bigs, Introduction to Structural Dynamics" and is based on the north-south component of the May 18, 1940 El Centro California earthquake. As stated in this document, the recorded maximum quantities were 0.33g, 13.7 in/sec, and 8.3 in. This is intended to apply to elastic systems having between 5 and 10 % critical damping. For the El Centro input given below, the three straight lines are defined by.

(1) $u_{max} = (y_{so})_{max} = 8.3 \text{ in.}$ small f

(2) $u_{max} = \dfrac{1.5(\dot{y}_{so})_{max}}{2\pi f} = \dfrac{3.3}{f} \text{ in.}$ intermediate f

(3) $u_{max} = \dfrac{2(\ddot{y}_{so})_{max}}{(2\pi f)^2} = \dfrac{6.6}{f^2} \text{ in.}$ large f

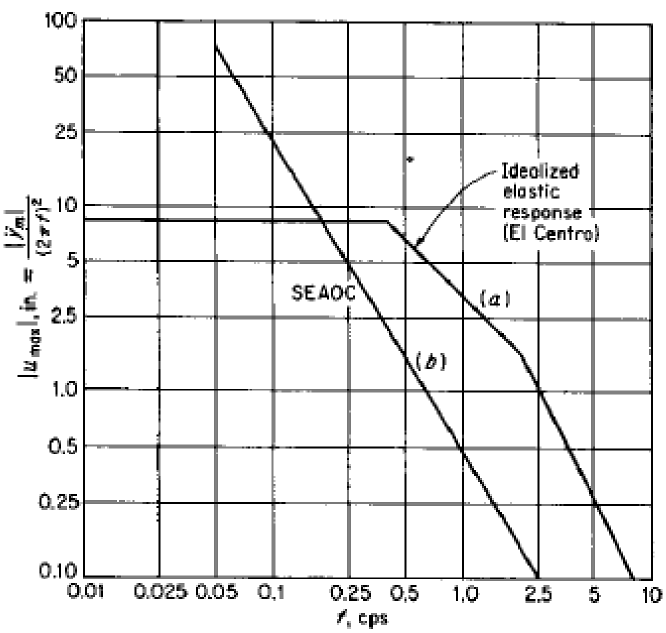


FIGURE 6.12 Idealized response spectrum for El Centro earthquake, May 18, 1940, N-S component and SEAOC recommendation.

Uniform Building Code (1991 Edition)

This spectrum data can be accessed by selecting the file “UBC_1991.spe” available in the folder “SpectrumLibrary” through Layout window >Misc> Spectrums > File > Library. This library has predefined spectrums for three (3) types of soils as shown in the figure below (taken from Uniform Building Code 1991 Edition).

Rock and Stiff Soils (Soil Type 1) [Library Name in CAEPIPE: RASS (S1)]

Deep and Cohesionless or Stiff Clay Soils (Soil Type 2) [Library Name in CAEPIPE: DCSCS (S2)]

Soft to Medium Clays and Sands [Library Name in CAEPIPE: SMCS (S3)]

These spectrums must be scaled by the Zero Period Accelerations (ZPA), which is the product of Z and I, where Z is the seismic zone coefficient and I is the earthquake importance factor as given in UBC Tables 23-I and 23-L. They are reproduced from UBC code for easy reference. This product can be applied in CAEPIPE as a scale factor through Layout window > Loads > Spectrum > Factor (s).

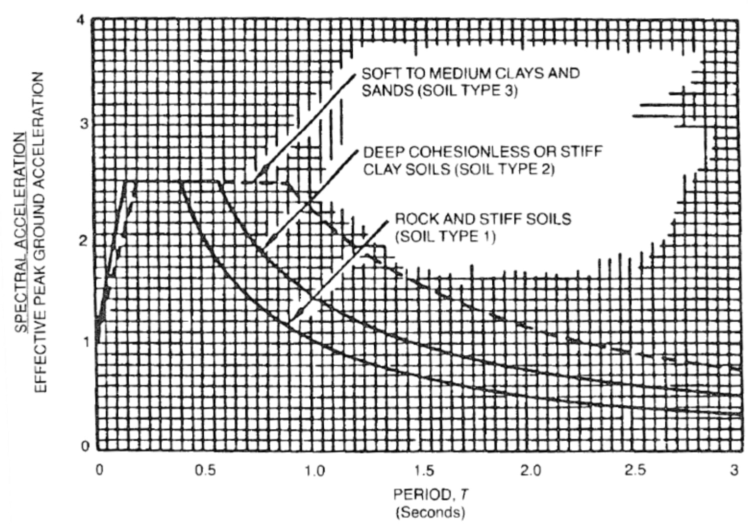


FIGURE NO. 23-3—NORMALIZED RESPONSE SPECTRA SHAPES

TABLE NO. 23-I
SEISMIC ZONE FACTOR *Z*

ZONE	1	2A	2B	3	4
Z	0.075	0.15	0.20	0.30	0.40

The zone shall be determined from the seismic zone map in Figure No. 23-2.

TABLE NO. 23-L—OCCUPANCY REQUIREMENTS

OCCUPANCY CATEGORY[1]	IMPORTANCE FACTOR I	
	Earthquake[2]	Wind
I. Essential facilities	1.25	1.15
II. Hazardous facilities	1.25	1.15
III. Special occupancy structures	1.00	1.00
IV. Standard occupancy structures	1.00	1.00

[1]Occupancy types or functions of structures within each category are listed in Table No. 23-K and structural observation requirements are given in Sections 305, 306 and 307.
[2]For life-safety-related equipment, see Section 2336 (a).

Nuclear Regulatory Commission (NRC) Guide 1.60

This spectrum data can be accessed by selecting the file "NRC_G1.6_2014.spe" available in the folder "SpectrumLibrary" through Layout window >Misc> Spectrums > File > Library. This library have ten (10) predefined spectrums are based upon Safe Shutdown Earthquake (SSE) for Horizontal and Vertical Design Response Spectra corresponding to 0.5, 2.0, 5.0, 7.0 and 10% Damping factors as shown in the figure below (taken from NRC Guide 1.60, July 2014, Revision 2)

These predefined spectrums are named in the CAPIPE spectrum library as follows.

- NRC-HDS-D0.5 (Horizontal Design Spectra with 0.5% Damping Factor)
- NRC-HDS-D2 (Horizontal Design Spectra with 2% Damping Factor)
- NRC-HDS-D5 (Horizontal Design Spectra with 5% Damping Factor)
- NRC-HDS-D7 (Horizontal Design Spectra with 7% Damping Factor)
- NRC-HDS-D10 (Horizontal Design Spectra with 10% Damping Factor)
- NRC-VDS-D0.5 (Vertical Design Spectra with 0.5% Damping Factor)
- NRC-VDS-D2 (Vertical Design Spectra with 2% Damping Factor)
- NRC-VDS-D5 (Vertical Design Spectra with 5% Damping Factor)
- NRC-VDS-D7 (Vertical Design Spectra with 7% Damping Factor)
- NRC-VDS-D10 (Vertical Design Spectra with 10% Damping Factor)

The horizontal and vertical component Design Response Spectra in Figures 1 and 2 correspond to a maximum horizontal ground acceleration of 1.0g. For sites with different acceleration values specified for the design earthquake, the Design Response Spectra should be linearly scaled from Figures 1 and 2 in proportion to the specified maximum horizontal ground acceleration. This proportion factor calculated can be applied in CAEPIPE as a scale factor through Layout window > Loads > Spectrum > Factor (s).

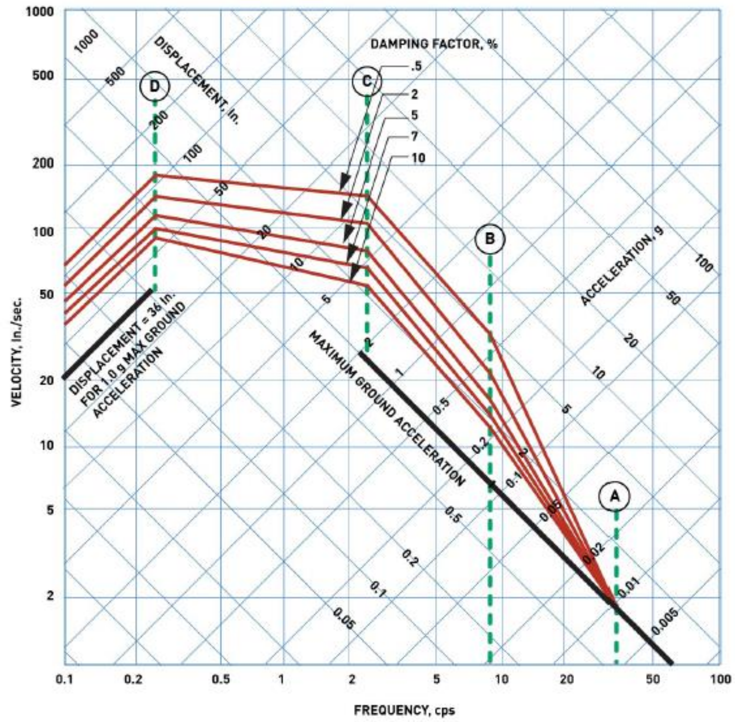


Figure 1. Horizontal Design Response Spectra Scaled to 1 g Horizontal Ground Acceleration

Horizontal Component:

The numerical values of design displacements, velocities, and accelerations for the horizontal component Design Response Spectra are obtained by multiplying the corresponding values of the maximum ground displacement and acceleration by the factors given in Table 1 given below. In Figure 1, the base diagramconsists of three parts: the bottom line on the left part represents the maximum grounddisplacement, the bottom line on the right part represents the maximum acceleration, and themiddle part depends on the maximum velocity. The horizontal component Design ResponseSpectra in Figure 1 of this guide correspond to a maximum horizontal ground acceleration of 1.0g. The maximum ground displacement is taken proportional to the maximum groundacceleration, and is set at 36 inches for a ground acceleration of 1.0 g. The displacement regionlines of the Design Response Spectra are parallel to the maximum ground displacement line andare shown on the left of Figure 1. The velocity region lines slope downward from a frequency of0.25 cycles per second (cps) or Hertz (Hz) (control point D) to a frequency of 2.5 cps (controlpoint C) and are shown at the top. The remaining two sets of lines between the frequencies of 2.5cps and 33 cps (control point A), with a break at a frequency of 9 cps (control point B), constitutethe acceleration region of the horizontal Design Response Spectra. For frequencies higher than33 cps, the maximum ground acceleration line represents the Design Response Spectra.

Table 1. Horizontal Design Response Spectra

Relative Values of Spectrum Amplification Factors for Control Points

Percent of Critical Damping	Amplification Factors for Control Points			
	Acceleration[a,b]			Displacement[a,b]
	A (33 cps)	B (9 cps)	C (2.5 cps)	D (0.25 cps)
0.5	1.0	4.96	5.95	3.20
2.0	1.0	3.54	4.25	2.50
5.0	1.0	2.61	3.13	2.05
7.0	1.0	2.27	2.72	1.88
10.0	1.0	1.90	2.28	1.70

a. Maximum ground displacement is taken proportional to maximum ground acceleration, and is 36 in. for ground acceleration of 1.0 gravity.

b. Acceleration and displacement amplification factor are taken from recommendations given in Newmark, N. M., John A. Blume, and Kanwar K. Kapur, "Design Response Spectra for Nuclear Power Plants," American Society of Civil Engineers (ASCE) Structural Engineering Meeting, San Francisco, April 1973, (ADAMS Accession No. ML13207A044).

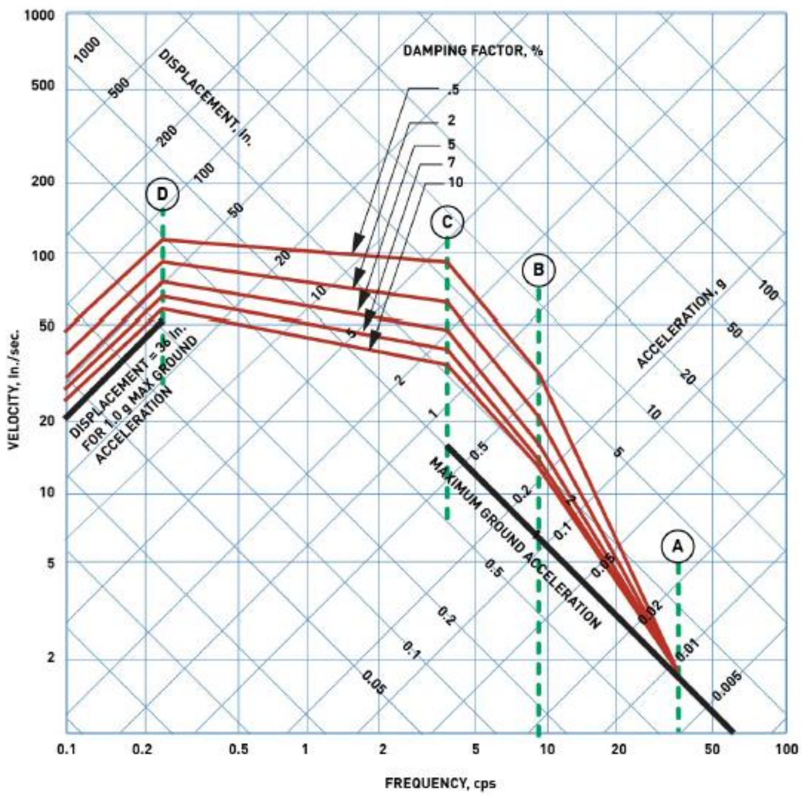


Figure 2. Vertical Design Response Spectra scaled to 1 g Horizontal Ground Acceleration

Table 2. Vertical Design Response Spectra

Relative Values of Spectrum Amplification Factors for Control Points

Percent of Critical Damping	Amplification Factors for Control Points			
	Acceleration[a,b]			Displacement[a,b]
	A (33 cps)	B (9 cps)	C (3.5 cps)	D (0.25 cps)
0.5	1.0	4.96	5.67[c]	2.13
2.0	1.0	3.54	4.05	1.67
5.0	1.0	2.61	2.98	1.37
7.0	1.0	2.27	2.59	1.25
10.0	1.0	1.90	2.17	1.13

a. Maximum ground displacement is taken proportional to maximum ground acceleration and is 36 in. for ground acceleration of 1.0 gravity.

b. Acceleration amplification factors for the vertical design response spectra are equal to those for horizontal design response spectra at a given frequency, whereas displacement amplification factors are 2/3 those for horizontal design response spectra. These ratios between the amplification factors for the two design response spectra are in agreement with those recommended in Newmark, N. M., John A. Blume, and Kanwar K. Kapur, "Design Response Spectra for Nuclear Power Plants," American Society of Civil Engineers (ASCE) Structural Engineering Meeting, San Francisco, April 1973, (ADAMS Accession No. ML13207A044).

The Vertical Component

The numerical values of design displacements, velocities, and accelerations in these spectra are obtained by multiplying the corresponding values of the maximum horizontal ground motion (acceleration = 1.0 g and displacement = 36 in.) by the factors given in Table 2.

The vertical component Design Response Spectra corresponding to the maximum horizontal ground acceleration of 1.0 g are shown in Figure 2. The displacement region lines of the Design Response Spectra are parallel to the maximum ground displacement line and are shown on the left of Figure 2. The velocity region lines slope downward from a frequency of 0.25 cps (control point D) to a frequency of 3.5 cps (control point C) and are shown at the top. The remaining two sets of lines between the frequencies of 3.5 cps and 33 cps (control point A), with a break at the frequency of 9 cps (control point B), constitute the acceleration region of the vertical Design Response Spectra. It should be noted that the vertical Design Response Spectra values are 2/3 those of the horizontal Design Response Spectra for frequencies less than 0.25; for frequencies higher than 3.5, they are the same, while the ratio varies between 2/3 and 1 for frequencies between 0.25 and 3.5. For frequencies higher than 33 cps, the Design Response Spectra follow the maximum ground acceleration line.

Note:
Since the acceleration values below 2.5 cps and 3.5 cps for Horizontal Design Response Spectra and Vertical Design Response Spectra respectively are NOT given in the Figures 1 and 2 above, the acceleration values corresponding to 2.5 cps (for Horizontal Design Response Spectra) and 3.5 cps (for Vertical Design Response Spectra) are entered for 0.1 cps in the respective directions.

Similarly, the accelerations values above 33 cps and up to 60 cps are entered as 1.0 g in the respective directions.

Index

A

B

C

D

E

F

O

P

Q

R

S

T

U

V

W

Z