

CAESAR II

Applications Guide



Version 2014 (7.0)

April 2014

DICAS-PE-200101E



Copyright

Copyright © 1985-2014 Intergraph CAS, Inc. All Rights Reserved. Intergraph is part of **Hexagon**.

Including software, file formats, and audiovisual displays; may be used pursuant to applicable software license agreement; contains confidential and proprietary information of Intergraph and/or third parties which is protected by copyright law, trade secret law, and international treaty, and may not be provided or otherwise made available without proper authorization from Intergraph Corporation.

Contains RealDWG™ by Autodesk, Inc. Copyright © 1998-2013 Autodesk, Inc. All rights reserved.

U.S. Government Restricted Rights Legend

Use, duplication, or disclosure by the government is subject to restrictions as set forth below. For civilian agencies: This was developed at private expense and is "restricted computer software" submitted with restricted rights in accordance with subparagraphs (a) through (d) of the Commercial Computer Software - Restricted Rights clause at 52.227-19 of the Federal Acquisition Regulations ("FAR") and its successors, and is unpublished and all rights are reserved under the copyright laws of the United States. For units of the Department of Defense ("DoD"): This is "commercial computer software" as defined at DFARS 252.227-7014 and the rights of the Government are as specified at DFARS 227.7202-3.

Unpublished - rights reserved under the copyright laws of the United States.

Intergraph Corporation
300 Intergraph Way
Huntsville, AL 35813

Documentation

Documentation shall mean, whether in electronic or printed form, User's Guides, Installation Guides, Reference Guides, Administrator's Guides, Customization Guides, Programmer's Guides, Configuration Guides and Help Guides delivered with a particular software product.

Other Documentation

Other Documentation shall mean, whether in electronic or printed form and delivered with software or on Intergraph Smart Support, SharePoint, or box.net, any documentation related to work processes, workflows, and best practices that is provided by Intergraph as guidance for using a software product.

Terms of Use

- a. Use of a software product and Documentation is subject to the End User License Agreement ("EULA") delivered with the software product unless the Licensee has a valid signed license for this software product with Intergraph Corporation. If the Licensee has a valid signed license for this software product with Intergraph Corporation, the valid signed license shall take precedence and govern the use of this software product and Documentation. Subject to the terms contained within the applicable license agreement, Intergraph Corporation gives Licensee permission to print a reasonable number of copies of the Documentation as defined in the applicable license agreement and delivered with the software product for Licensee's internal, non-commercial use. The Documentation may not be printed for resale or redistribution.
- b. For use of Documentation or Other Documentation where end user does not receive a EULA or does not have a valid license agreement with Intergraph, Intergraph grants the Licensee a non-exclusive license to use the Documentation or Other Documentation for Licensee's internal non-commercial use. Intergraph Corporation gives Licensee permission to print a reasonable number of copies of Other Documentation for Licensee's internal, non-commercial. The Other Documentation may not be printed for resale or redistribution. This license contained in this subsection b) may be terminated at any time and for any reason by Intergraph Corporation by giving written notice to Licensee.

Disclaimer of Warranties

Except for any express warranties as may be stated in the EULA or separate license or separate terms and conditions, Intergraph Corporation disclaims any and all express or implied warranties including, but not limited to the implied warranties of merchantability and fitness for a particular purpose and nothing stated in, or implied by, this document or its contents shall be considered or deemed a modification or amendment of such disclaimer. Intergraph believes the information in this publication is accurate as of its publication date.

The information and the software discussed in this document are subject to change without notice and are subject to applicable technical product descriptions. Intergraph Corporation is not responsible for any error that may appear in this document.

The software, Documentation and Other Documentation discussed in this document are furnished under a license and may be used or copied only in accordance with the terms of this license. THE USER OF THE SOFTWARE IS EXPECTED TO MAKE THE FINAL EVALUATION AS TO THE USEFULNESS OF THE SOFTWARE IN HIS OWN ENVIRONMENT.

Intergraph is not responsible for the accuracy of delivered data including, but not limited to, catalog, reference and symbol data. Users should verify for themselves that the data is accurate and suitable for their project work.

Limitation of Damages

IN NO EVENT WILL INTERGRAPH CORPORATION BE LIABLE FOR ANY DIRECT, INDIRECT, CONSEQUENTIAL INCIDENTAL, SPECIAL, OR PUNITIVE DAMAGES, INCLUDING BUT NOT LIMITED TO, LOSS OF USE OR PRODUCTION, LOSS OF REVENUE OR PROFIT, LOSS OF DATA, OR CLAIMS OF THIRD PARTIES, EVEN IF INTERGRAPH CORPORATION HAS BEEN ADVISED OF THE POSSIBILITY OF SUCH DAMAGES.

UNDER NO CIRCUMSTANCES SHALL INTERGRAPH CORPORATION'S LIABILITY EXCEED THE AMOUNT THAT INTERGRAPH CORPORATION HAS BEEN PAID BY LICENSEE UNDER THIS AGREEMENT AT THE TIME THE CLAIM IS MADE. EXCEPT WHERE PROHIBITED BY APPLICABLE LAW, NO CLAIM, REGARDLESS OF FORM, ARISING OUT OF OR IN CONNECTION WITH THE SUBJECT MATTER OF THIS DOCUMENT MAY BE BROUGHT BY LICENSEE MORE THAN TWO (2) YEARS AFTER THE EVENT GIVING RISE TO THE CAUSE OF ACTION HAS OCCURRED.

IF UNDER THE LAW RULED APPLICABLE ANY PART OF THIS SECTION IS INVALID, THEN INTERGRAPH LIMITS ITS LIABILITY TO THE MAXIMUM EXTENT ALLOWED BY SAID LAW.

Export Controls

Intergraph Corporation's software products and any third-party Software Products obtained from Intergraph Corporation, its subsidiaries, or distributors (including any Documentation, Other Documentation or technical data related to these products) are subject to the export control laws and regulations of the United States. Diversion contrary to U.S. law is prohibited. These Software Products, and the direct product thereof, must not be exported or re-exported, directly or indirectly (including via remote access) under the following circumstances:

- a. To Cuba, Iran, North Korea, Sudan, or Syria, or any national of these countries.
- b. To any person or entity listed on any U.S. government denial list, including but not limited to, the U.S. Department of Commerce Denied Persons, Entities, and Unverified Lists, <http://www.bis.doc.gov/complianceand enforcement/liststocheck.htm>, the U.S. Department of Treasury Specially Designated Nationals List, <http://www.treas.gov/offices/enforcement/ofac/>, and the U.S. Department of State Debarred List, <http://www.pmddtc.state.gov/compliance/debar.html>.
- c. To any entity when Licensee knows, or has reason to know, the end use of the Software Product is related to the design, development, production, or use of missiles, chemical, biological, or nuclear weapons, or other un-safeguarded or sensitive nuclear uses.
- d. To any entity when Licensee knows, or has reason to know, that an illegal reshipment will take place.

Any questions regarding export or re-export of these Software Products should be addressed to Intergraph Corporation's Export Compliance Department, Huntsville, Alabama 35894, USA.

Trademarks

Intergraph, the Intergraph logo, Intergraph Smart, PDS, SmartPlant, SmartMarine, FrameWorks, I-Sketch, IntelliShip, ISOGEN, SmartSketch, SPOOLGEN, SupportManager, and SupportModeler are trademarks or registered trademarks of Intergraph Corporation or its subsidiaries in the United States and other countries. Microsoft and Windows are registered trademarks of Microsoft Corporation. MicroStation is a registered trademark of Bentley Systems, Inc. Other brands and product names are trademarks of their respective owners.

Contents

Overview	9
Bends	11
Bend Definition.....	11
Single and Double Flanged Bends or Stiffened Bends.....	12
180° Return Fitting-to-Fitting 90° Bends.....	14
Mitered Bends.....	15
Closely-Spaced Mitered Bend.....	15
Widely-Spaced Mitered Bend.....	16
Elbows - Different Wall Thickness.....	19
Bend Flexibility Factor.....	20
Restraints	21
Anchors.....	22
Anchors with Displacements.....	23
Flexible Anchors.....	24
Flexible Anchors with Predefined Displacements.....	25
Flexible Nozzle - WRC Bulletin 297.....	26
Flexible Nozzle with Predefined Displacements.....	28
Flexible Nozzle with Complete Vessel Model.....	29
Double-Acting Restraints.....	32
Single-Directional Restraints.....	33
Guides.....	34
Limit Stops.....	36
Windows.....	38
Rotational Directional Restraints with Gaps.....	39
Single-Directional Restraint with Predefined Displacement.....	40
Single-Directional Restraint and Guide with Gap and Predefined Displacement.....	41
Restraint Settlement.....	42
Skewed Double-Acting Restraint with Gap.....	43
Skewed Single-Directional Restraint.....	44
Restraint between Two Pipes Using CNodes.....	45
Restraint between Vessel and Pipe Models.....	46
Restraints on a Bend at 45°.....	46
Restraints on a Bend at 30° and 60°.....	48
Vertical Dummy Leg on Bends.....	49
Vertical Leg Attachment Angle.....	52
Horizontal Dummy Leg on Bends.....	52
Large Rotation Rods - Basic Model.....	53
Large Rotation Rods - Chain Supports.....	55
Bi-Linear Restraints.....	56
Static Snubbers.....	58
Plastic Hinges.....	59
Sway Brace Assemblies.....	60

Hangers	63
General Information	64
Simple Hanger Design	65
Single Can Design	66
Constant Effort Support Design	67
Constant Effort Supports - No Design	68
Existing Springs - No Design	69
Multiple Can Design	70
Old Spring Redesign	71
Pipe and Hanger Supported From Vessel	72
Hanger Design with Support Thermal Movement	73
Hanger Between Two Pipes	73
Hanger Design with Anchors in the Vicinity	74
Hanger Design with User-Specified Operating Load	76
Simple Bottomed Out Spring	77
Lift Off Spring Can	78
Modeling Spring Cans with Friction	79
Expansion Joints	81
Universal Expansion Joints - Simple Model	81
Tied Bellows Expansion Joint - Simple Model	87
Universal Joint with Lateral Controls Stops - Comprehensive Tie Rod Model	88
Hinged Joint	90
Slotted Hinge Joint - Simple Model	92
Slotted Hinge Joint - Comprehensive Model	93
Tied Bellows - Simple vs. Complex Model	94
Slip Joint	95
Gimbal Joints	96
Tied Bellows Expansion Joint - Complex Model	97
Dual Gimbal	99
Simple Bellows with Pressure Thrust	101
Pressure-Balanced Tees and Elbows	104
Universal Joint - Comprehensive Tie Rod	105
Miscellaneous Models	107
Reducers	107
Ball Joints	108
Jacketed Pipe	110
Cold Spring	112
Connecting Equipment	112
Vertical Vessels	112
Horizontal Vessels	118
Tutorial A	123
System Overview	123
Preparing the Drawing	124
Generating CAESAR II Input	126
Input Review	140
Ending the Input Session	143
Performing the Static Analysis	143

Reviewing Static Results	146
Static Analysis Output Listing	151
Conclusions	159
Examples	161
Harmonic Analysis (TABLE)	161
Harmonic Analysis of this System	164
Relief Valve Loads (RELIEF)	166
Relief Valve Example Problem Setup	167
Relief Valve Loading Output	171
Dynamic Analysis of Water Hammer Loads (HAMMER)	175
Notes for Analyzing Water Hammer Loads	183
Water Hammer Loading Output	185
Solution Summary	187
Dynamic Analysis of Independent Support Earthquake Excitation (CRYNOS)	188
Results	200
Structural Analysis (FRAME)	201
Dynamic Analysis (NUREG9)	208
NRC Example NUREG 9	209
NRC Example Problem 2A	209
NRC Example Problem 2B	212
NRC Example Problem 2C	213
Omega Loop Modelling (OMEGA)	214
Jacketed Piping (JACKET)	219
Modeling Plan	220
Node Layout	221
Core Piping Input	222
Jacket Input (1st Half)	223
Jacket Input (2nd Half)	226
WRC 107	228
Convert Forces/Moments from CAESAR II Global Coordinates to WRC 107 Local Axes	229
NEMA SM23	237
Nozzle Results for PT69M	240
Nozzle Load Summation Report	241
Tutorial B	243
Evaluating Pump Discharge Loads	243
Creating Accurate Models	254
WRC 297 Calculations Completed at the End of Error Checking	257
Checking Nozzle Loads	263
System Redesign	265
Conclusion	275
Index	277

SECTION 1

Overview

The *CAESAR II Application Guide* shows the application of CAESAR II. Refer to this manual for examples of specific piping components, as well as examples of complete systems.

The following sections illustrate the techniques and methods used to model individual piping components, restraints, and attached equipment.

- *Bends* (on page 11)
- *Restraints* (on page 21)
- *Expansion Joints* (on page 81)
- *Miscellaneous Models* (on page 107)

Refer to these sections when you are modeling seldom-used components or unusual geometries. The component modeling examples are especially useful for modeling techniques and gaining a general understanding of the software.

Examples (on page 161) contains examples that illustrate the application of CAESAR II to various piping problems. These examples show modeling, problem solving, and software operation. This section provides engineering guidelines and indicates where assumptions must be made in attempting to solve real-world problems.

Tutorial A (on page 123) and *Tutorial B* (on page 243) contain tutorials that walk through the modeling and analysis of a complete system. Work through these sections if you have not previously attempted a particular analysis.

SECTION 2

Bends

This section provides guidelines and recommended best practices for the modeling various bend geometries in CAESAR II.

NOTE For detailed information about using the software to define bend geometry, see "Component Information" in the *CAESAR II User's Guide*.

Bend Definition

A bend is defined by the element entering the bend and the element leaving the bend. The bend curvature is always physically at the **To** end of the element entering the bend.

The input for the element leaving the bend must follow the element entering the bend. The bend angle is defined by these two elements. The default bend radius is 1-1/2 times the pipe nominal diameter (long radius), but it can be changed to any other value. When you specify a bend, two additional intermediate nodes are automatically generated--one at the 0° location and one at the bend midpoint (M).

For stress and displacement output, the **To** node of the element entering the bend is located geometrically at the far-point on the bend. The far-point is at the weld line of the bend, adjacent to the straight element leaving the bend. The 0° point on the bend is at the weld line of the bend, adjacent to the straight element entering the bend.

The **From** point on the element is located at the 0° point of the bend (and no 0° node point is generated) if the total length of the element as specified by **DX**, **DY**, and **DZ** is equal to:

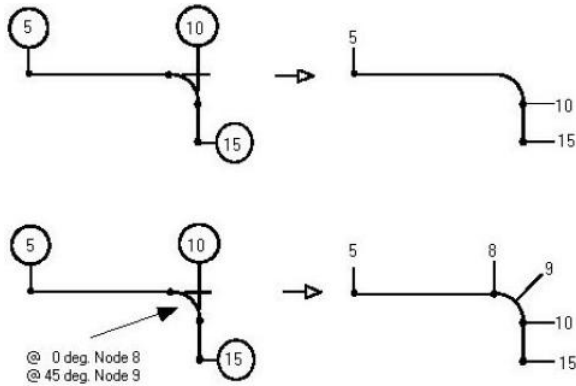
$$R \tan (b / 2)$$

Where **b** is the bend angle, and **R** is the bend radius of curvature to the bend centerline.

Nodes defined by the **Angle** and **Node** properties are placed at the given angle on the bend curvature. The angle starts with zero degrees at the near-point on the bend and goes to **b** degrees at the far-point of the bend. Angles are always entered in degrees. Entering the letter **M** as the angle designates the bend midpoint.

Nodes on the bend curvature cannot be placed closer together than the angle distance specified by **Minimum Angle to Adjacent Bend** in the **Geometry** category of the **Tools > Configure/Setup** command. This includes the spacing between the nodes on the bend curvature and the near- and far-points of the bend.

The minimum and maximum total bend angle is specified by the **Minimum Allowable Bend Angle** and **Maximum Allowable Bend Angle** properties, also in the **Geometry** category of the **Tools > Configure Setup** command.



Single and Double Flanged Bends or Stiffened Bends

Single- and double-flanged bend specifications affect only the stress intensification and flexibility of the bend. There is no automatic rigid element (or change in weight) generated for the end of the bend. Single- and double-flanged bends are indicated by typing a **1** or **2** to define the **Type** in the bend auxiliary input. Rigid elements defined before or after the bend do not alter the stiffness of the bend or its stress intensification factors.

When specifying single-flanged bends, it does not matter on which end of the bend you place the flange.

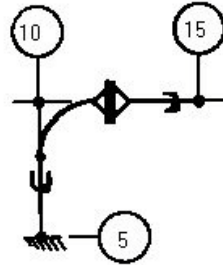
If you want to include the weight of the rigid flange at the bend ends, then put rigid elements (whose total length is the length of a flange pair) at the bend ends where the flange pairs exist.

As a guideline, British Standard 806 recommends stiffening the bends whenever a component that significantly stiffens the pipe cross section is found within two diameters of either bend end.

The flanges in the figures below are modeled only to the extent that they affect the stiffness and the stress intensification for the bends.

Flanges and Bends:

From: 5	<input type="checkbox"/> Name	<input checked="" type="checkbox"/> Bend
To: 10		<input type="checkbox"/> Rigid
		<input type="checkbox"/> Expansion Joint
DX: <input type="text"/>		<input type="checkbox"/> Restraints
DY: 10 ft. <input type="text"/>		<input type="checkbox"/> Hangers



Bends

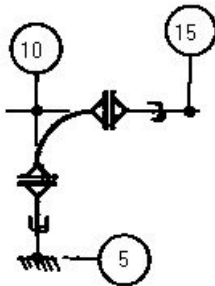
Radius: 9.000

Type: 1 - Single Flange

Angle 1: M Node 1: 9

Angle 2: 0.000 Node 2: 8

Angle 3: Node 3:



Bends

Radius: 9.000

Type: 2 - Double Flange

Angle 1: M Node 1: 9

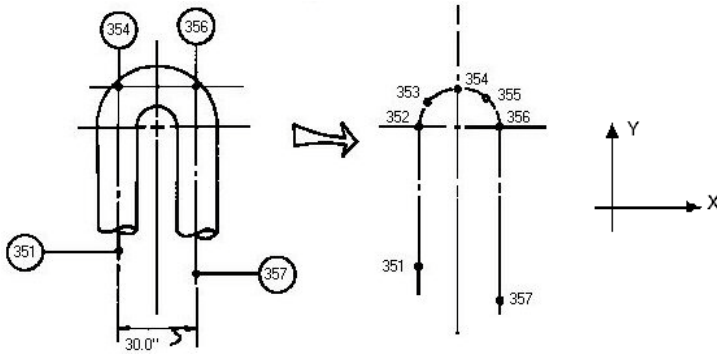
Angle 2: 0.000 Node 2: 8

Angle 3: Node 3:

180° Return Fitting-to-Fitting 90° Bends

Separate two 90° bends by twice the bend radius. The far-point of the first bend is the same as the near-point of the second, or following, bend. Intergraph CAS recommends that you place nodes at the midpoint of each bend that comprise the 180° return.

180° Bend:



From: 351	<input type="checkbox"/> Name	<input checked="" type="checkbox"/> Bend
To: 354		<input type="checkbox"/> Rigid
		<input type="checkbox"/> Expansion Joint
DX: <input type="text"/>		
DY: 5 ft. 6.000 in <input type="text"/>		
From: 354	<input type="checkbox"/> Name	<input checked="" type="checkbox"/> Bend
To: 356		<input type="checkbox"/> Rigid
		<input type="checkbox"/> Expansion Joint
DX: 2 ft. 6.000 in <input type="text"/>		
DY: <input type="text"/>		
<input type="checkbox"/> Restraints		
<input type="checkbox"/> Hangers		

Bends	
Radius:	15.000
Type:	<input type="text"/>
Angle 1:	M
Node 1:	353
Angle 2:	0.000
Node 2:	352
Angle 3:	<input type="text"/>
Node 3:	<input type="text"/>

DX = 2R

From: 356	<input type="checkbox"/> Name	<input checked="" type="checkbox"/> Bend
To: 357		<input type="checkbox"/> Rigid
		<input type="checkbox"/> Expansion Joint
DX: <input type="text"/>		
DY: -5 ft. 6.000 in <input type="text"/>		
<input type="checkbox"/> Restraints		
<input type="checkbox"/> Hangers		

Bends	
Radius:	15.000
Type:	<input type="text"/>
Angle 1:	M
Node 1:	355
Angle 2:	<input type="text"/>
Node 2:	<input type="text"/>
Angle 3:	<input type="text"/>
Node 3:	<input type="text"/>

Mitered Bends

Evenly spaced mitered bends, whether closely or widely spaced, are defined by two parameters:

- The number of cuts (changes in direction)
- The equivalent radius or miter spacing

For closely spaced miters, the equivalent radius is equal to the code defined as **R1** for B31.3 and **R** for B31.1. The equation for the equivalent radius to the spacing for evenly spaced miters is:

$$Req = S / [2 \tan(\theta)]$$

Where:

Req = equivalent miter bend radius

S = spacing of the miter cuts along the centerline

θ = code-defined half-angle between adjacent miter cuts: $\theta = \alpha / 2N$

Where:

α = total bend angle

N = number of cuts

When using B31.1, an additional parameter, **B** (length of miter segment at crotch), is examined for closely spaced miters. The following equation is used to compute **B** for evenly spaced miters:

$$B = S [1 - r_o / Req]$$

Where:

r_o = outside radius of pipe cross-section

Closely-Spaced Mitered Bend

Miter bends are closely spaced if:

$$S < r [1 + \tan (\theta)]$$

Where:

S = miter spacing

r = average pipe cross section radius: $(r_i+r_o)/2$

q = one-half the angle between adjacent miter cuts

B31.1 has additional requirements:

$$B > 6 t_n$$

$$\theta \leq 22.5 \text{ deg.}$$

B = length of the miter segment at the crotch.

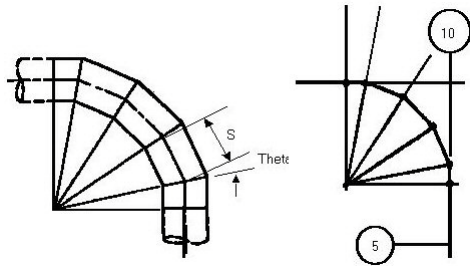
t_n = nominal wall thickness of pipe.

Bends

Closely spaced miters, regardless of the number of miter cuts, can be defined as a single bend. CAESAR II always calculates the spacing from the bend radius. If you have the miter spacing but not the bend radius, the radius must be calculated as shown in the following example. The mitered bend shown below has four cuts through 90° and a spacing of 15.913 inches.

$$\begin{aligned}R_{eq} &= S / [2 \tan (\theta)] \\ \theta &= \alpha / 2N \\ &= 90 / [2(4)] \\ &= 11.25^\circ \\ R_{eq} &= 15.913 / [2 \tan (11.25^\circ)] \\ &= 40\end{aligned}$$

Closely Spaced Miter Bend:



Bends

Radius:

Type:

Angle 1: Node 1:

Angle 2: Node 2:

Angle 3: Node 3:

Miter Points:

Widely-Spaced Mitered Bend

Mitered bends are widely spaced if:

$$S^3 r * [1 + \tan (\theta)]$$

Where:

S = spacing between miter points along the miter segment centerline

r = average cross section radius $(r_i+r_o)/2$

θ = one-half angle between adjacent miter cuts

B31.1 has the following additional requirement:

$$\theta \leq 22.5^\circ$$

In CAESAR II, you must enter widely spaced miters as individual, single-cut miters, each having a bend radius equal to:

$$R = r [1 + \cot(\theta)] / 2$$

Where:

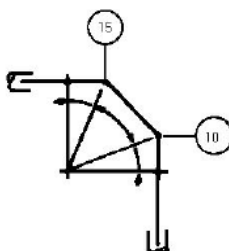
R = reduced bend radius for widely spaced miters.

During error checking, CAESAR II produces a warning message for each mitered component that does not pass the test for a closely spaced miter.

The following components should be re-entered as a group of single cut joints.

Widely Spaced Miter:

- Pipe O.D. = 10.375 in.
- Pipe Thickness = 0.500 in.
- Bend Angle = 90°
- Cuts = 2
- Req = 45 inches



Assuming closely spaced:

$$\theta = \alpha / 2N = 90 / (2 * 2) = 22.5 \text{ deg.}$$

$$S = R_{eq} 2 \tan(\theta) \quad r_2 = \frac{(D_o - t)}{2}$$

$$= 45 [2 \tan(22.5)] = \frac{(10.375 - 0.5)}{2}$$

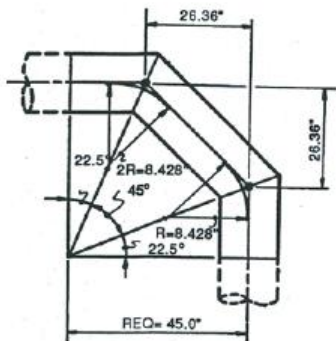
$$= 37.279 \text{ in.} \quad = 4.9375 \text{ in.}$$

Now check $s \geq r_2 [1 + \tan(\theta)]$

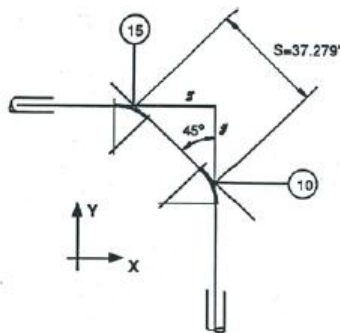
$$r_2 [1 + \tan(\theta)] = 4.9375 [1 + \tan(22.5)] = 6.9826 \text{ in.}$$

37.279 > 6.9826 The miter is widely spaced.

$$R = \frac{r [1 + \cot(\theta)]}{2} = 8.4288 \text{ in.}$$



$$g = 37.279 \sin 45 \text{ deg}$$



Calculate the Δ coordinates to get from the tangent intersection point of the single cut miter bend at node 10 to the single cut miter bend at node 15.

NOTE The straight pipe section coming into and going out of the bend must be $\geq R_{eq} \sin(\theta)$.

$$R = \frac{r}{2} [1 + \tan(\theta)]$$

$$= \left[\left(\frac{4.9375}{2} \right) [1 + \cot(22.5)] \right]$$

$$= 8.4288 \text{ in.}$$

$$DX = S \times \cos(45) = 37.279 \times \cos(45) = 26.360 \text{ in.}$$

$$DY = S \times \sin(45) = 37.279 \times \sin(45) = 26.360 \text{ in.}$$

Bends

Enter widely spaced miters as individual straight pipe elements, with bends specified, having one miter cut.

From: 5 Name
To: 10

Bend
 Rigid
 Expansion Joint

DX:
DY: 3 ft.

Restraints
 Hangers

Bends

Radius: 8.429
Type:

Angle 1: M Node 1: 9
Angle 2: 0.000 Node 2: 8
Angle 3: Node 3:

Miter Points: 1

Figure 1: Coming Up to the First Cut

From: 10 Name
To: 15

Bend
 Rigid
 Expansion Joint

DX: -2 ft. 2.360 in
DY: 2 ft. 2.360 in

Restraints
 Hangers

Bends

Radius: 8.429
Type:

Angle 1: M Node 1: 14
Angle 2: 0.000 Node 2: 13
Angle 3: Node 3:

Figure 2: Between the First and Second Cuts

From: 15 Name
To: 20

Bend
 Rigid
 Expansion Joint

DX: -3 ft.
DY:

Restraints
 Hangers

Figure 3: Coming Out of the Second Cut

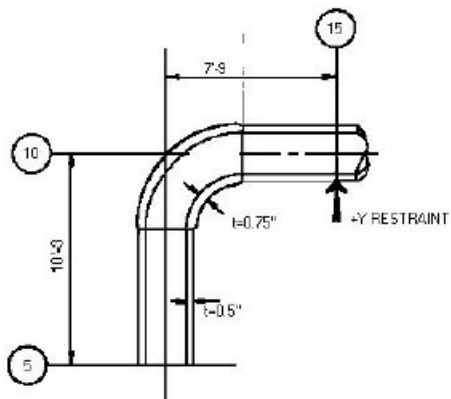
Elbows - Different Wall Thickness

When you define the fitting thickness for the bend, CAESAR II changes the thickness only of the curved portion of the bend element. The thickness of any preceding or following straight pipe is unaffected.

The specified fitting thickness applies only for the current elbow and is not persisted to any subsequent elbows in the job.

Stresses at the elbow are calculated based on the section modulus of the matching pipe as specified in the B31 codes. However, stress intensification factors and flexibility factors for the bend are based on the elbow wall thickness.

Thick Elbow:



NOTE The elbow at node 10 has a thickness larger than the matching pipe wall. The matching pipe has a thickness of 0.5.

From:	5	<input type="checkbox"/> Name
To:	10	
DX:		
DY:	10 ft. 3.000 in	...
DZ:		
<input type="checkbox"/> Offsets		
Diameter:	10.7500	
Wt/Sch:	0.5000	

Bends	
Radius:	15.000
Type:	
Angle 1:	M
Node 1:	9
Angle 2:	0.000
Node 2:	8
Angle 3:	
Node 3:	
Miter Points:	
Fitting Thk:	0.750
K-factor:	
<input type="checkbox"/> Seam/Welded	

Bend Flexibility Factor

Usually, bend flexibility factors are calculated according to code requirements. However, you can override the code calculation by typing a value for the **K-factor**. For example, if you type **1.500**, then the bend is 1.5 times as flexible as a straight pipe of the same length.

The image shows a software dialog box for configuring bend parameters. On the left is a vertical sidebar with tabs: 'Model Status', 'Allowable Stresses', 'Bends' (which is selected), 'Displacements', and 'Exp'. The main area contains the following fields:

- Radius: 18.000
- Type: [Dropdown menu]
- Angle 1: 45.000 Node 1: 59
- Angle 2: 0.000 Node 2: 58
- Angle 3: [Empty] Node 3: [Empty]
- Miter Points: [Empty]
- Fitting Thk: [Empty]
- K-factor: 1.500
- Seam Welded
- WI Factor: [Empty]

SECTION 3

Restraints

This chapter provides guidelines and recommended best practices for modeling various types of restraints in CAESAR II.

In This Section

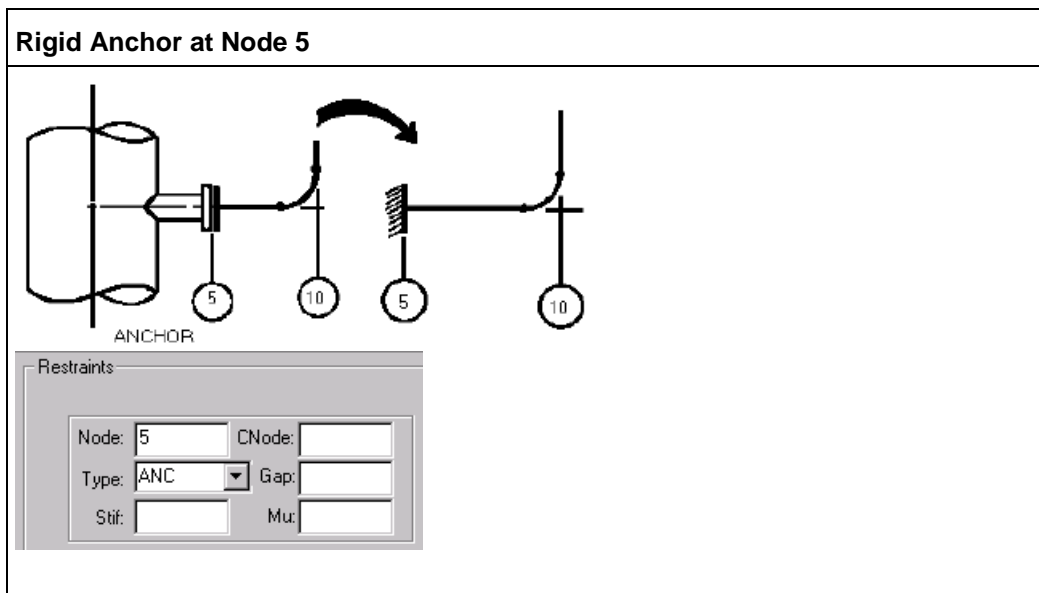
Anchors.....	22
Flexible Nozzle - WRC Bulletin 297.....	26
Double-Acting Restraints.....	32
Single-Directional Restraints.....	33
Guides.....	34
Limit Stops.....	36
Windows.....	37
Rotational Directional Restraints with Gaps.....	38
Single-Directional Restraint with Predefined Displacement.....	40
Single-Directional Restraint and Guide with Gap and Predefined Displacement.....	40
Restraint Settlement.....	41
Skewed Double-Acting Restraint with Gap.....	42
Skewed Single-Directional Restraint.....	44
Restraint between Two Pipes Using CNodes.....	45
Restraint between Vessel and Pipe Models.....	45
Restraints on a Bend at 45°.....	46
Restraints on a Bend at 30° and 60°.....	47
Vertical Dummy Leg on Bends.....	48
Vertical Leg Attachment Angle.....	52
Horizontal Dummy Leg on Bends.....	52
Large Rotation Rods - Basic Model.....	53
Large Rotation Rods - Chain Supports.....	54
Bi-Linear Restraints.....	56
Static Snubbers.....	58
Plastic Hinges.....	58
Sway Brace Assemblies.....	59

Anchors

Use the following guidelines when modeling anchors in CAESAR II:

- The anchor default stiffness for translational and rotational degrees-of-freedom is defined in the CAESAR II configuration file.
- You can use connecting nodes with anchors to rigidly fix one point in the piping system to any other point in the piping system.
- The value that you define for the **Stif** property applies to all six anchor degrees-of-freedom.
- Do not specify displacements at an anchor. If you know the displacements of a particular point, specify them without any additional restraints or anchors.
- Accurate definition of piping boundary conditions (restraints) is considered the single most important part of system modeling. It requires experience with piping fabrication and erection and with CAESAR II.

The example below shows a nozzle connection modeled as an anchor, along with the corresponding rigid anchor input on the **Restraints Auxiliary Data** tab in the **Classic Piping Input** dialog box.



Anchors with Displacements

Follow these guidelines to model anchors with displacements in CAESAR II:

- Enter only displacements for the node.
- Do not specify restraints or anchors at the node to be displaced.
- For anchors with displacements, ensure that all six degrees-of-freedom at the node are defined.

NOTE Degrees-of-freedom not defined (left blank) in any displacement vector are presumed to be free in all load cases.

You can define up to nine different displacement vectors.

Non-zero displacements are usually part of the thermal expansion effects. They should be added into any analysis case containing the corresponding thermal, such as W+P1+T1+D1. We recommend load cases that do this automatically.

The translations and/or rotations for any nodal degree of freedom that has displacements specified in any displacement vector are zero for the following load cases:

- Those that do not contain a vector as part of the load case identification
- Those where the specified non-zero value for load cases containing the vector is part of the load case identification

For example, defined displacements are used if the load case is W+P1+T1+D1 (OPE), and those displacements are held to zero if the load case is W+P1 (SUS).

After a degree-of-freedom is fixed in one displacement vector, it cannot be free in another displacement vector at the same node.

NOTE The default value for a blank displacement box is zero.

The figure below shows an anchor with predefined displacements.

Anchors with Predefined Displacements

The diagram shows a cross-section of an anchor with two nodes, 5 and 10. Node 5 is at the base of the anchor, and node 10 is at the end of the anchor. A coordinate system is shown with X and Y axes. Displacements are indicated as DX=0.005 and DY=0.131. The Displacements dialog box shows the following data:

	Vector 1	Vector 2	Vector 3
DX	0.0050		
DY	0.1300		
DZ	0.0000		
RX	0.0000		
RY	0.0000		
RZ	0.0000		

Flexible Anchors

Use the following guidelines to model flexible anchors in CAESAR II:

- Use six flexible restraints.
- Put four restraints on one element and the last two restraints on the next element in the restraints auxiliary data area (**Classic Piping Input** dialog box).
- Refer to the following flexible nozzle examples to improve modeling methods for intersections of this type.

Flexible Restraints for Nozzle and Shell

Restraints			
Node: 5	CNode:		
Type: X	Gap:		
Stif: 100000.00	Mu:		

Node: 5	CNode:		
Type: Y	Gap:		
Stif: 230000000.	Mu:		

Node: 5	CNode:		
Type: Z	Gap:		
Stif: 100000.00	Mu:		

Node: 5	CNode:		
Type: RX	Gap:		
Stif: 100000000	Mu:		

Restraints			
Node: 5	CNode:		
Type: RY	Gap:		
Stif: 100000000	Mu:		

Node: 5	CNode:		
Type: RZ	Gap:		
Stif: 100000000	Mu:		

Node:	CNode:		
Type:	Gap:		
Stif:	Mu:		

Node:	CNode:		
Type:	Gap:		
Stif:	Mu:		

NOTE For more information about defining restraints, see "Restraints" in the *CAESAR II User's Guide*.

Flexible Anchors with Predefined Displacements

To model flexible anchors with predefined displacements, use the following guidelines:

- Use six flexible restraints.
- Put four restraints on one element and the last two restraints on the next element.
- Define a unique connecting node (CNode) at each of the six restraints. All six restraints should have the same connecting node.
- Specify the displacements at the connecting node.

Flexible Anchors with Predefined Displacements

The CNode in the example below is 1005. CNode numbers must be unique.

NOZZLE & SHELL FLEX:
 KX=1E5 RKX=1E8
 KY=2.3E7 RKY=1E8
 KZ=1E5 RKZ=1E8

Displacements

Node 1:

	Vector 1	Vector 2	Vector 3
DX	0.0050		
DY	0.1300		
DZ	0.0000		
RX	0.0000		
RY	0.0000		
RZ	0.0000		

Restraints

Node:	<input type="text" value="5"/>	CNode:	<input type="text" value="1005"/>
Type:	<input type="text" value="X"/>	Gap:	<input type="text"/>
Stif:	<input type="text" value="100000.00"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text" value="5"/>	CNode:	<input type="text" value="1005"/>
Type:	<input type="text" value="Y"/>	Gap:	<input type="text"/>
Stif:	<input type="text" value="230000000"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text" value="5"/>	CNode:	<input type="text" value="1005"/>
Type:	<input type="text" value="Z"/>	Gap:	<input type="text"/>
Stif:	<input type="text" value="100000.00"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text" value="5"/>	CNode:	<input type="text" value="1005"/>
Type:	<input type="text" value="RX"/>	Gap:	<input type="text"/>
Stif:	<input type="text" value="100000000"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text" value="5"/>	CNode:	<input type="text" value="1005"/>
Type:	<input type="text" value="RY"/>	Gap:	<input type="text"/>
Stif:	<input type="text" value="100000000"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text" value="5"/>	CNode:	<input type="text" value="1005"/>
Type:	<input type="text" value="RZ"/>	Gap:	<input type="text"/>
Stif:	<input type="text" value="100000000"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text"/>	CNode:	<input type="text"/>
Type:	<input type="text"/>	Gap:	<input type="text"/>
Stif:	<input type="text"/>	Mu:	<input type="text"/>

Restraints

Node:	<input type="text"/>	CNode:	<input type="text"/>
Type:	<input type="text"/>	Gap:	<input type="text"/>
Stif:	<input type="text"/>	Mu:	<input type="text"/>

CAESAR II Applications Guide

25

Flexible Nozzle - WRC Bulletin 297

When modeling flexible nozzles, adhere to the following requirements:

- Frame only one pipe element into the nozzle node.
- Do not place restraints at the nozzle node.
- Do not place anchors at the nozzle node.
- Do not specify displacements for the nozzle node. Refer to the displacements at flexible nozzles below.

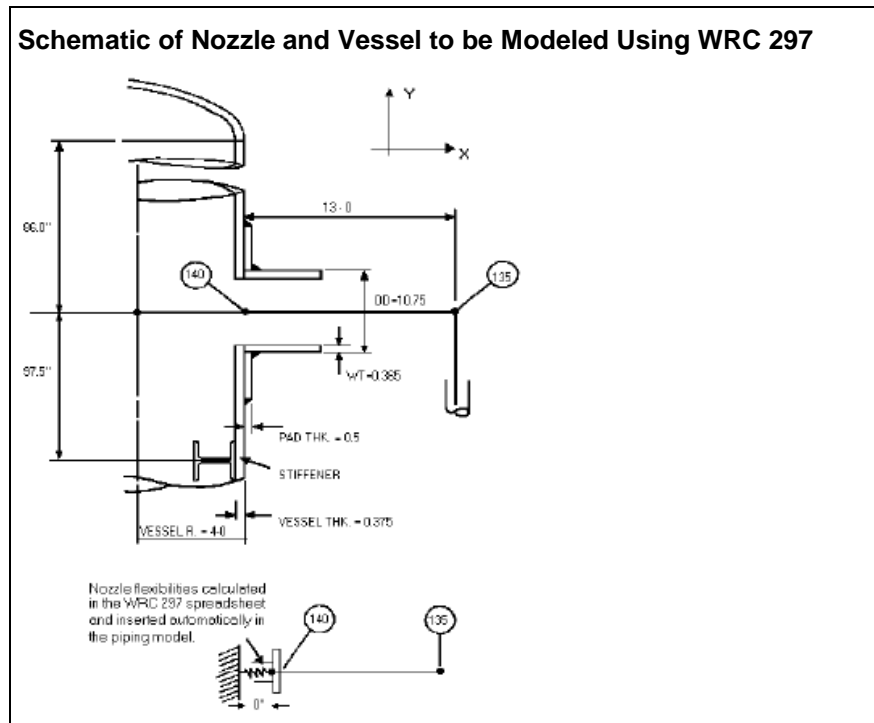
CAESAR II automatically performs the following functions:

- Calculates nozzle flexibilities for user-defined nozzle/vessel data.
- Calculates and inserts restraints to simulate nozzle flexibilities.
- Calculates flexibilities for the axial translations, circumferential, and longitudinal bending.

You must complete the error check process to view these calculated values.

CAESAR II uses the following criteria for its calculations:

- Shear and torsional stiffnesses are assumed rigid.
- Nozzle configurations outside of the WRC 297 curve limits are considered rigid. It is not unusual for one stiffness value to be rigid because of curve limits and for the others to be suitably flexible.
- You can use **Vessel Temperature** and **Material** in the WRC 297 auxiliary data area to optionally compute a reduced modulus of elasticity for the local stiffness calculations.



WRC 297 Input Example

Nozzles

Type

WRC 297

API 650

PD 5500

Nozzle Node: 140

Vessel Node (optional):

Nozzle Outer Diameter: 10.750

Nozzle Wall Thickness: 0.365

Vessel Outer Diameter: 96.000

Vessel Wall Thickness: 0.375

Vessel Pad Thickness: 0.500

Distance to Stiffener or Head: 86.000

Distance to Opposite Stiffener: 97.500

Vessel Centerline Direction Cosine VX:

Vessel Centerline Direction Cosine VY: 1.000

Vessel Centerline Direction Cosine VZ:

Vessel Temperature (optional):

Vessel Material (optional):

From: 135 To: 140 Name

DX: -1 ft. 4.000 in

DY: ...

DZ: ...

Offsets

Diameter: 10.7500

Wt/Sch: 0.3650

Bend Reducer

Rigid SIFs & Tees

Expansion Joint

Restraints Displacements

Hangers

Nozzles

Forces/Moments

Uniform Loads

Wind / Wave

Example WRC 297 output is shown below.

WRC 297 Nozzle Calculations

```

WRC NOZZLE NODE = 140
VESSEL Dmean(in.)= 95.125      VESSEL THK. (in.)= 0.875
NOZZLE O.D. (in.)= 10.750     NOZZLE THK. (in.)= 0.365
AXIAL TRANSLATIONAL STIFFNESS (lb./in.)= 555887.
LONGITUDINAL BENDING STIFFNESS (in.lb./deg)= 1149087.
CIRCUMFERENTIAL BENDING STIFFNESS (in.lb./deg)= 335126.
ANGLE BETWEEN NOZZLE & VESSEL CENTERLINES(deg)= 90.0000
LENGTH (L) (in.) = 182.959    THICKNESS RATIO = 2.397
CAPITAL LAMBDA = 20.054       SMALL LAMBDA = 1.178
    
```

Flexible Nozzle with Predefined Displacements

To model flexible nozzles with predefined displacements (WRC-297), use the following guidelines:

- Define a unique vessel node in the nozzle auxiliary data area (**Classic Piping Input** dialog box).
- Apply the predefined displacements to the vessel node.

NOTE These displacements can be specified on any element. The displacement node does not need to be on an element that defines it.

The CAESAR II-generated nozzle/vessel flexibilities are inserted in restraints that act between the nozzle node and the vessel node.

Flexible Nozzle With Predefined Displacements on Vessel Node

Nozzles

Type

WRC 297

API 650

PD 5500

Nozzle Node: 140

Vessel Node (optional): 240

Nozzle Outer Diameter: 10.750

Nozzle Wall Thickness: 0.365

Vessel Outer Diameter: 96.000

Vessel Wall Thickness: 0.375

Vessel Pad Thickness: 0.500

Distance to Stiffener or Head: 86.000

Distance to Opposite Stiffener: 97.500

Vessel Centerline Direction Cosine VX:

Vessel Centerline Direction Cosine VY: 1.000

Vessel Centerline Direction Cosine VZ:

Vessel Temperature (optional):

Vessel Material (optional):

Displacements

Node 1: 240

	Vector 1	Vector 2	Vector 3
DX	0.0000		
DY	2.5000		
DZ	0.0000		
RX	0.0000		
RY	0.0000		
RZ	0.0000		

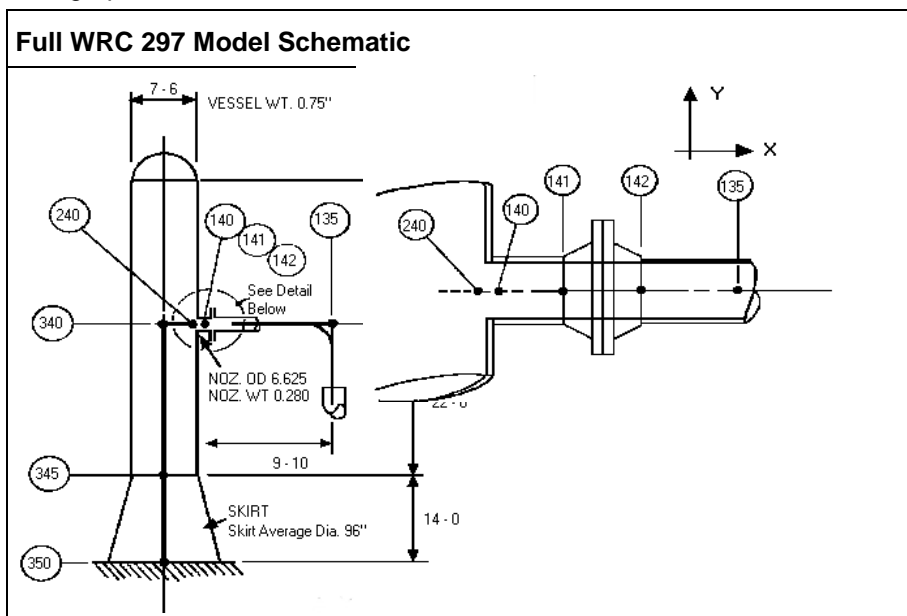
Flexible Nozzle with Complete Vessel Model

To model a flexible nozzle that includes a complete vessel, use the following guidelines:

- Define a unique vessel node on the nozzle auxiliary data area.
 - TIP** The nozzle auxiliary data area displays when you select **Nozzle Flex** on the **Classic Piping Input** dialog box. For more information, see "Nozzle Flex" in the *CAESAR II User's Guide*.
- Run a rigid element between the vessel node defined in the nozzle auxiliary data area and the centerline of the vessel. The outside diameter of the rigid element should be approximately equal to the outside diameter of the vessel. The weight of the rigid element should be zero.
- Model the actual vessel length using pipe elements. Model the vessel diameter and wall thicknesses as accurately as possible
- Use an anchor to model the vessel anchorage point.

The CAESAR II-generated nozzle/vessel flexibilities are inserted between the nozzle node and the vessel node.

The graphics below show a full WRC 297 model schematic.



Full WRC 297 and Vessel Model

From: 141 Name
 To: 140

DX: -2 ft. 3.000 in
 DY: ...
 DZ:

Offsets

Diameter: 6.6250
 Wt/Sch: 0.2800
 +Mill Tol %: 12.5000
 -Mill Tol %: 12.5000
 Seam Welded

Corrosion:
 Insul Thk:

Temp 1: 550.0000
 Temp 2:
 Temp 3: ...
 Pressure 1: 100.0000
 Pressure 2:
 Hydro Press:

Figure 4: Pipe Entering Nozzle

Type
 WRC 297
 API 650
 PD 5500

Nozzle Node: 140
 Vessel Node (optional): 240
 Nozzle Outer Diameter: 6.625
 Nozzle Wall Thickness: 0.280
 Vessel Outer Diameter: 90.000
 Vessel Wall Thickness: 0.750
 Vessel Pad Thickness: 0.500
 Distance to Stiffener or Head: 156.000
 Distance to Opposite Stiffener: 264.000
 Vessel Centerline Direction Cosine VX:
 Vessel Centerline Direction Cosine VY: 1.000
 Vessel Centerline Direction Cosine VZ:
 Vessel Temperature (optional):
 Vessel Material (optional):

Figure 5: WRC 297 Auxiliary Input

From: 240 Name
 To: 340

DX: -3 ft. 9.000 in
 DY: ...
 DZ:

Offsets

Diameter: 90.0000
 Wt/Sch: 0.7500

Temp 1: 550.0000

Figure 6: Vessel Radius Element

Bend Reducer
 Rigid SIFs & Tees
 Expansion Joint Structural

Rigids

Rigid Weight:

Figure 7: Rigid Weight is Blank (0,0)

From: 340 To: 345 Name

DX: DY: -22 ft. DZ:

Offsets

Diameter: 90.0000 Wt/Sch: 0.7500 +Mill Tol %: 12.5000 -Mill Tol %: 12.5000 Seam Welded

Corrosion: Insul Thk:

Temp 1: 550.0000

Figure 8: Vessel Element

From: 345 To: 350 Name

DX: DY: -14 ft. DZ:

Offsets

Diameter: 96.0000 Wt/Sch: 0.7500 Seam Welded

Wt Factor: -Mill Tol %: 12.5000

Corrosion: Insul Thk:

Temp 1: 70.0000

Figure 9: Vessel Skirt Element At Ambient Temperature

. Type: ANC, Gap: . Stif: Mu: ."/>

Restraints

Node: 350 CNode:

Type: ANC Gap:

Stif: Mu:

Nozzle Calculations

An example of the WRC 297 results found at the end of error checking is shown below.

```

WRC NOZZLE NODE = 140
VESSEL Dmean(in.)= 95.125      VESSEL THK. (in.)= 0.875
NOZZLE O.D. (in.)= 10.750     NOZZLE THK. (in.)= 0.365
AXIAL TRANSLATIONAL STIFFNESS ( lb./in. )= 555887.
LONGITUDINAL BENDING STIFFNESS (in.lb./deg)= 1149087.
CIRCUMFERENTIAL BENDING STIFFNESS (in.lb./deg)= 335126.
ANGLE BETWEEN NOZZLE & VESSEL CENTERLINES(deg)= 90.0000
LENGTH (L) (in.) = 182.959    THICKNESS RATIO = 2.397
CAPITAL LAMBDA = 20.054      SMALL LAMBDA = 1.178
    
```

Double-Acting Restraints

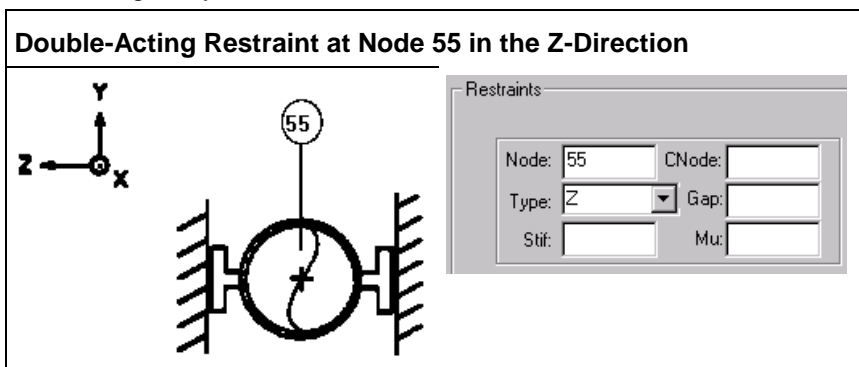
Double-acting restraints are those that act in both directions along the line of action. The most commonly used restraints are double-acting.

A CNode is the connecting node. If you do not enter a value in the **CNode** box, the restrained node is connected using the restraint stiffness to a rigid point in space. If you do enter a value for **CNode**, the restrained node is connected using the restraint stiffness to the connecting node.

If you specify a gap, it is the amount of free movement along the positive or negative line of action of the restraint before resistance to movement occurs. A gap is a length, so it is always positive.

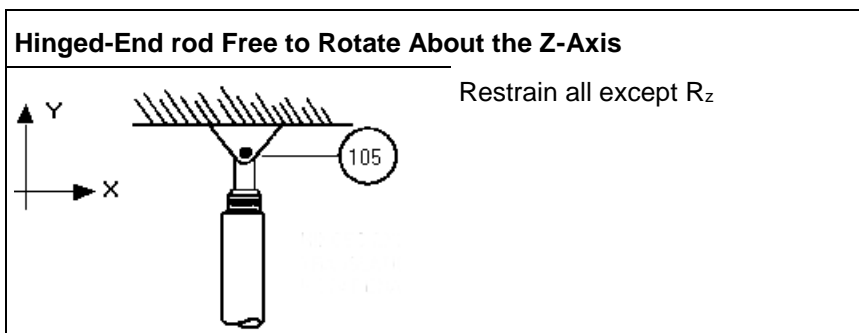
Transitional Restraints

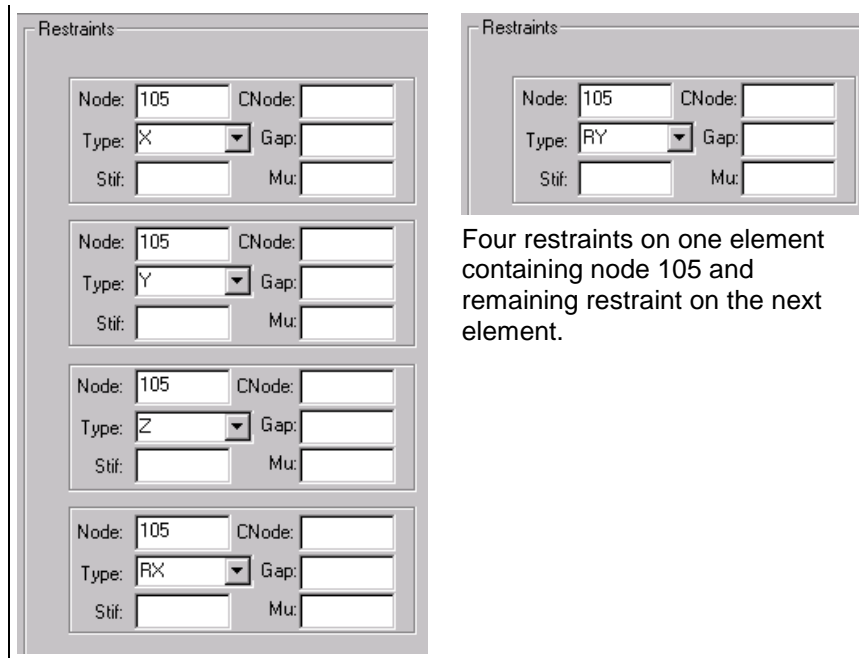
Restraint acts along both the positive and negative directions. Friction at double-acting restraints acts orthogonally to the line of action of the restraint.



Rotational Restraints

The behavior of rotation restraints is similar to double-acting translational restraints. Friction is not defined for rotational restraints.





Single-Directional Restraints

The following are some important facts pertaining to single-directional restraints:

- The plus or minus sign (+/-) on the single-directional restraint gives the direction of free movement. For example, a positive Y restraint (+Y) can move freely in the positive Y-direction and is restrained against movement in the negative Y-direction.
- Single-directional restraints can define restraint along positive, negative, or skewed axes.
- Any number of single-directional restraints can act along the same line of action. If more than one single-directional restraint acts along the same line of action, then there are usually two in opposite directions and are used to model unequal leg gaps.
- A CNode is the connecting node. If you do not define a value for **CNode**, then the restrained node is connected using the restraint stiffness to a rigid point in space. If you enter a value for **CNode**, then the restrained node is connected using the restraint stiffness to the connecting node.
- You can specify friction and gaps with single-directional restraints. The gap is in the opposite direction of the free travel.

Rigid Single - Directional Restraint in Y at Node 20

The plus or minus sign (+/-) on the restraint gives the direction of free movement. Because the stiffness is omitted, the restraint is rigid.

Guides

Use the following guidelines when modeling guides in CAESAR II:

- Guides are double-acting restraints with or without a specified gap.
- You can use connecting nodes (CNodes) with guides.
- You can define guides using the global system coordinates or using the applicable options in the restraints auxiliary data area.
- **TIP** The restraints auxiliary data area displays when you select **Restraints** on the **Classic Piping Input** dialog box.

A guided pipe in the horizontal or skewed direction has a single restraint, acting in the horizontal plane, orthogonal to the axis of the pipe.

A guided vertical pipe has X- and Z-direction supports.

CAESAR II computes direction cosines for guides. The software ignores user-defined guide direction cosines.

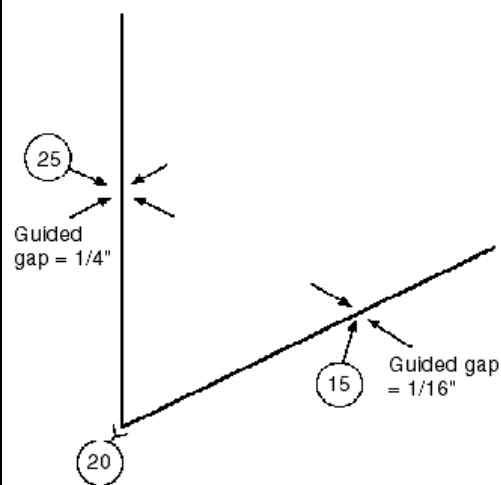
Guide on Horizontal Pipe with Single Directional Restraint

Node 25 is guided in Z with a gap of 2.5 inches. A single-directional restraint in the Y-direction also exists. Both restraints are rigid.

NOTE In this example, replacing the guide restraint with a Z restraint is equivalent.

Restraints			
Node:	25	CNode:	
Type:	Guide	Gap:	2.500
Stif:		Mu:	
Node:	25	CNode:	
Type:	+Y	Gap:	
Stif:		Mu:	

Guided Pipe in Both Horizontal and Vertical Directions



From:	15	<input type="checkbox"/> Name
To:	20	
DX:		
DY:		...
DZ:	8 ft.	
From:	20	<input type="checkbox"/> Name
To:	25	
DX:		
DY:	5 ft.	...
DZ:		

Restraints			
Node:	15	CNode:	
Type:	Guide	Gap:	0.063
Stif:		Mu:	
Node:	15	CNode:	
Type:	+Y	Gap:	
Stif:		Mu:	
Restraints			
Node:	25	CNode:	
Type:	Guide	Gap:	0.250
Stif:		Mu:	

Limit Stops

Use the following guidelines when working with limit stops in CAESAR II:

- Limit stops are single- or double-acting restraints whose line of action is along the axis of the pipe.
- The plus or minus sign (+/-) on the single-directional restraint gives the direction of unlimited free movement.
- A limit stop and a single-directional restraint can have gaps. The gap is the distance of permitted free movement along the restraining line of action.
- A gap is a length and is always positive. The plus or minus sign (+/-) on the restraint determines the orientation of the gap along the line of action. The gap direction is opposite to the direction of free travel.
- You can use connecting nodes (CNode) with any limit stop model.
- A limit stop can be defined using the restraint type **LIM**.
- Limit stops provide double- or single-acting support parallel to the pipe axis. Limit stops can have gaps and friction. The positive line of action of the limit stop is defined by the **From** and **To** node on the element.
- CAESAR II computes direction cosines for orthogonal or skewed limit stops. The software ignores user-defined limit stop direction cosines.

Directional Limit Stop with a Gap

Direction of data input
Orientation of (positive) limit stop

20

15

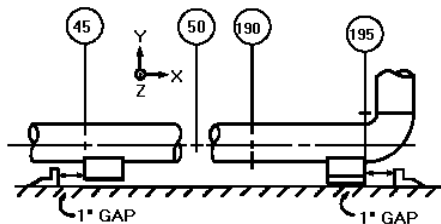
Gap = 1.5

From:	15	<input type="checkbox"/> Name
To:	20	
DX:	5 ft.	
DY:		...
DZ:		

orcesMoments Ha	Node:	20	CNode:	
	Type:	+LIM	Gap:	1.500
	Stif:		Mu:	

Two Limit Stops Acting in Opposite Directions

The stop at node 45 permits unlimited free movement in the positive X-direction and 1.0 inches of free movement in the negative X-direction before the limit stop becomes active.



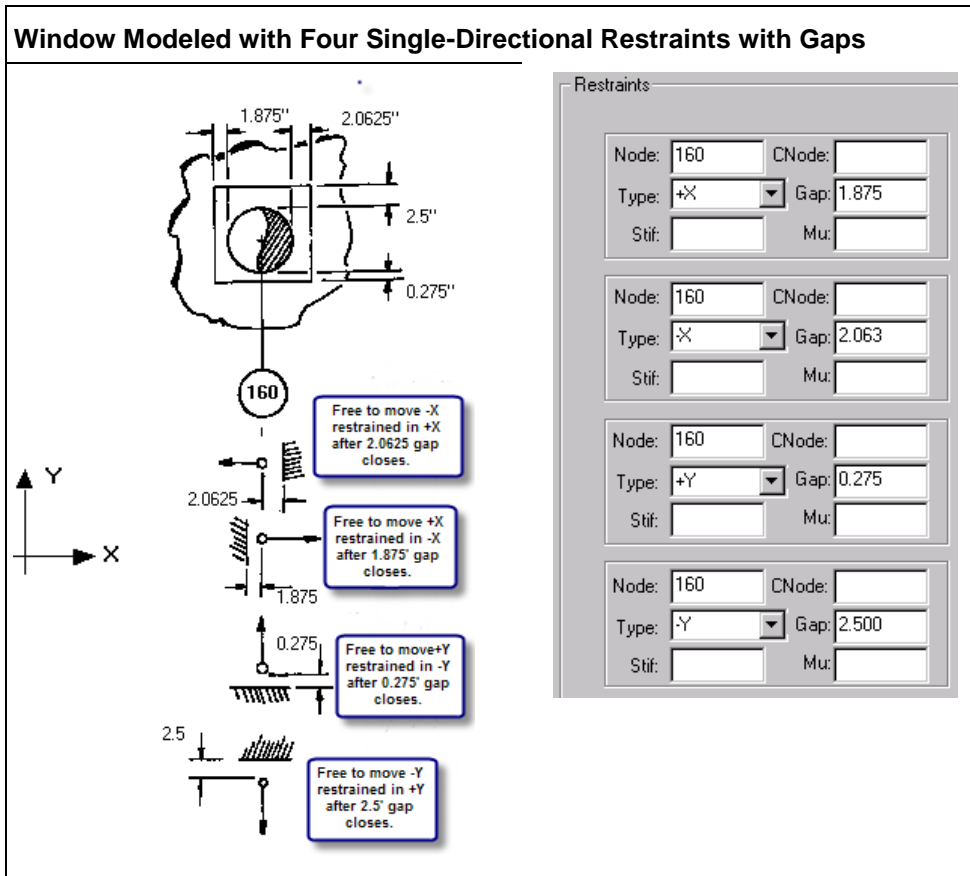
The stop at node 195 permits unlimited free movement in the negative X-direction and 1.0 inches of free movement in the positive X-direction.

From: 45 To: 50 <input type="checkbox"/> Name	From: 190 To: 195 <input type="checkbox"/> Name
DX: 5 ft. DY: <input type="text"/> DZ: <input type="text"/>	DX: 8 ft. DY: <input type="text"/> DZ: <input type="text"/>
orcesMoments Node: 45 CNode: <input type="text"/> Type: +LIM Gap: 1.000 Stif: <input type="text"/> Mu: <input type="text"/>	orcesMoments Node: 195 CNode: <input type="text"/> Type: -LIM Gap: 1.000 Stif: <input type="text"/> Mu: <input type="text"/>

Windows

Use the following guidelines when modeling windows in CAESAR II.

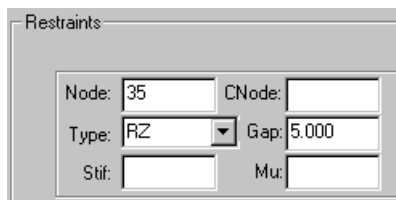
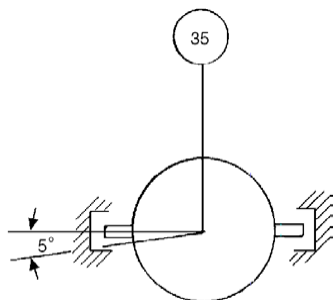
- Equal leg windows are modeled using two double-acting restraints with gaps orthogonal to the pipe axis.
- Unequal leg windows are modeled using four single-acting restraints with gaps orthogonal to the pipe axis, as shown in the example below.
- The gap is always positive. The plus or minus sign (+/-) on the restraint determines the direction of movement before the gap closes. If there is no plus or minus sign (+/-) on the restraint, then the restraint is double-acting, and the gap exists on both sides of the line of action of the restraint. If there is a plus or minus sign (+/-) on the restraint, then the gap exists on the restrained line of action of the restraint. For example, a positive Y (+Y) restraint is restrained against movement in the negative Y-direction. Any gap associated with a positive Y restraint is the free movement in the negative Y-direction before the restraint begins acting.



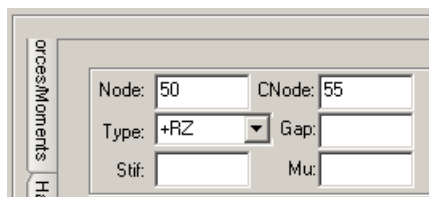
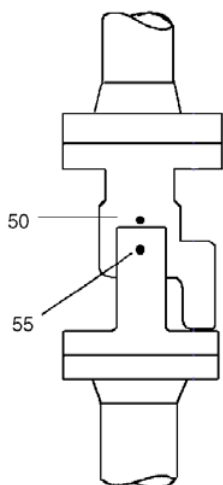
Rotational Directional Restraints with Gaps

Rotation directional restraints with gaps are considered specialty items. Usually, they used only in sophisticated expansion joint or hinge models.

Rotational Directional Restraints



The figure on the left shows a bi-directional rotational restraint with gap. Allowable rotation of 5° in either direction about the Z-axis before resistance to rotation is encountered.

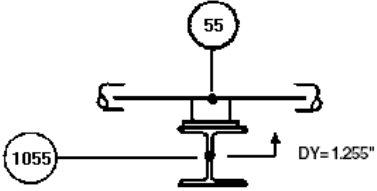


The figure on the left shows a hinge assembly with directional rotational restraint. Hinge assembly at node 50 can rotate relative to assembly at node 55 only in the positive direction about the Z-axis.

Single-Directional Restraint with Predefined Displacement

Define the one-directional restraint as usual, and enter a unique node number in the **CNode** box. Specify the predefined displacements for the CNode.

Single-Directional Restraint with Predefined Displacement



Piping at node 55 rests on top of the restraint that is displaced in the Y-direction, simulated by node 1055.

Restraints

Node: 55 CNode: 1055

Type: +Y Gap:

Stif: Mu:

Displacements

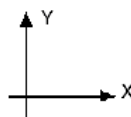
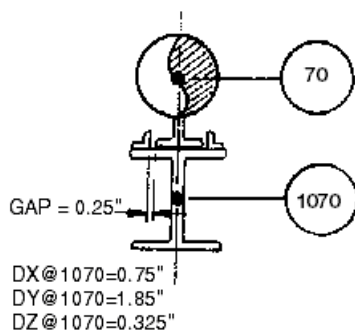
Node 1: 1055

	Vector 1	Vector 2	Vector 3
DX	0.0000		
DY	1.2550		
DZ	0.0000		
RX	0.0000		
RY	0.0000		
RZ	0.0000		

Single-Directional Restraint and Guide with Gap and Predefined Displacement

Define the single-directional restraint and guide as usual. Enter a unique node number in the **CNode** boxes for the single-directional restraint and the guide. Use the same node number for both. Specify the predefined displacements for the CNode.

Guide Plus Single-Directional Restraint with Predefined Displacement:



Guided piping at node 70 rests on a structural member node 1070. The structure undergoes a predefined displacement.

Restraints

Node: 70	CNode: 1070
Type: +Y	Gap:
Stif:	Mu:

Node: 70	CNode: 1070
Type: X	Gap: 0.250
Stif:	Mu:

Displacements

Node 1: 1070

	Vector 1	Vector 2	Vector 3
DX	0.7500		
DY	1.8500		
DZ	0.3250		
RX	0.0000		
RY	0.0000		
RZ	0.0000		

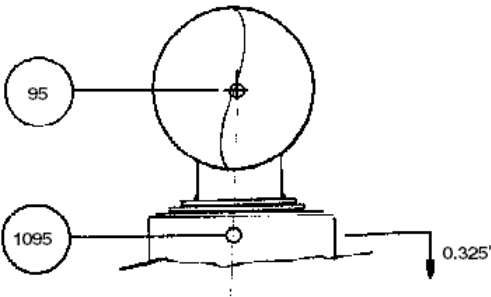
NOTE In the example above, ignoring other loads that may be acting on the pipe, the 0.250-inch gap will be closed in the negative X-direction, and the pipe will have a displacement of 0.50-inches in the positive X-direction.

Restraint Settlement

Use the following guidelines when modeling restraint settlement in CAESAR II:

- Model restraint settlement using a single-directional restraint with predefined displacements. The magnitude of the predefined displacement is the amount of anticipated settlement in the negative Y-direction.
- The operating load case is used to include the effect of settlement.
- The settlement displacements are prescribed for the connecting node at the single directional restraint. For more information, see *Single-Directional Restraint with Predefined Displacement* (on page 40).
- Settlement is considered to be an EXP load with a half-cycle.

Restraint Settlement



The weight of this pipe at node 95 exerts a sufficient load on the foundation node 1095 to cause a calculated 0.325-inch settlement.

Restraints

Node: 95 CNode: 1095

Type: +Y Gap:

Stif: Mu:

Displacements

Node 1: 1095

	Vector 1	Vector 2	Vector 3
DX	0.0000		
DY	-0.3250		
DZ	0.0000		
RX	0.0000		
RY	0.0000		
RZ	0.0000		

Skewed Double-Acting Restraint with Gap

Use the following guidelines when modeling skewed restraints in CAESAR II:

- You can use direction vectors or direction cosines to define the line of action of the restraint. If direction vectors are used, CAESAR II converts them to direction cosines.
- Direction cosines can be quickly checked in the graphics processor.
- Any translational axis can be used in the restraint description. The redefinition of the axis does not affect any other restraint description for the element.
- Use caution when entering skewed direction input data. A common mistake is to specify an axial instead of transverse restraint when modeling a skewed guide. Plotted section views of the restrained nodes are a useful check of the skewed direction specification.
- The sense of the direction or cosine unit vector is unimportant. In the definition of double-acting restraints, the direction vector and cosines are used only to define the restraint line of action and are not concerned with a direction along that line.
- You can use a simple rule for finding perpendicular, skewed, or direction vectors. The restraint is to be perpendicular to the pipe. If the pipe has skewed delta dimensions DX and DZ, the perpendicular restraint directions vector are (DX, 0, -DZ).

Skewed Double-Acting Restraint with Gap

SECTION A-A

ZERO GAP

Figure 10: Double Acting Restraint Y, Guided @ 45 Deg.

From: 35 To: 40 Name

DX: 5 ft. DY: ... DZ: -5 ft.

Node: 40 CNode: Type: Y Gap: Stif: Mu:

Node: 40	CNode:
Type: X (1, 0, 1)	Gap:
Stif:	Mu:

Figure 11: Input Using Unit Direction Vectors

Node: 40	CNode:
Type: X (.707, 0, .7)	Gap:
Stif:	Mu:

Figure 12: Input Using Direction Cosines

Node: 40	CNode:
Type: X (5, 0, 5)	Gap:
Stif:	Mu:

Figure 13: Input Using Perpendicular Vector

Node: 40	CNode:
Type: Guide	Gap:
Stif:	Mu:

Figure 14: Input Using Guide Restraint Type

Skewed Single-Directional Restraint

Use the following when modeling skewed single-directional restraints in CAESAR II:

- Skewed restraints can be nonlinear.
- Direction vectors or direction cosines can be used to define the line of action of the restraint. If direction vectors are used, CAESAR II converts them to direction cosines.
- The direction of the cosines or the direction vector is along the positive line of action of the positive (+) restraint. Refer to the example below
- Direction cosines can be quickly checked in the graphics processor.
- You can use connecting nodes (CNode) with any skewed single-directional restraint.

Skewed Single-Directional Restraint

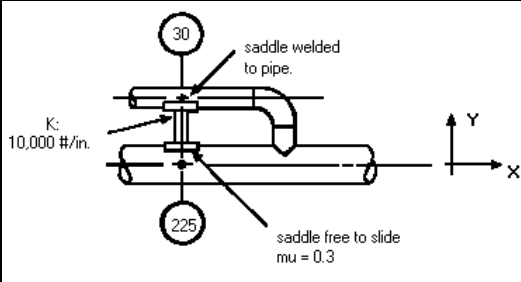
Restraints	
Node: 40	CNode:
Type: +X(1,0,1)	Gap:
Stif:	Mu:

Restraint between Two Pipes Using CNodes

NOTE For the following two examples, make sure **Connect Geometry Through CNodes** is not selected to avoid plotting and geometry errors. For more information, see "Geometry Definitions" in the *CAESAR II User's Guide*.

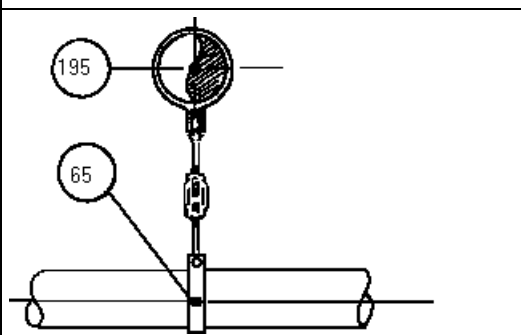
Nonlinear or linear restraints can act between two different pipe nodes. The **CNode** property effectively represents to what the other end of the restraint is attached.

Nonlinear Restraint Between Two Pipes



orces/Moments Ha	Node:	30	CNode:	225
	Type:	+Y	Gap:	
	Stif:	10000.000	Mu:	0.300

Rigid Rod Hanger Between Two Pipes

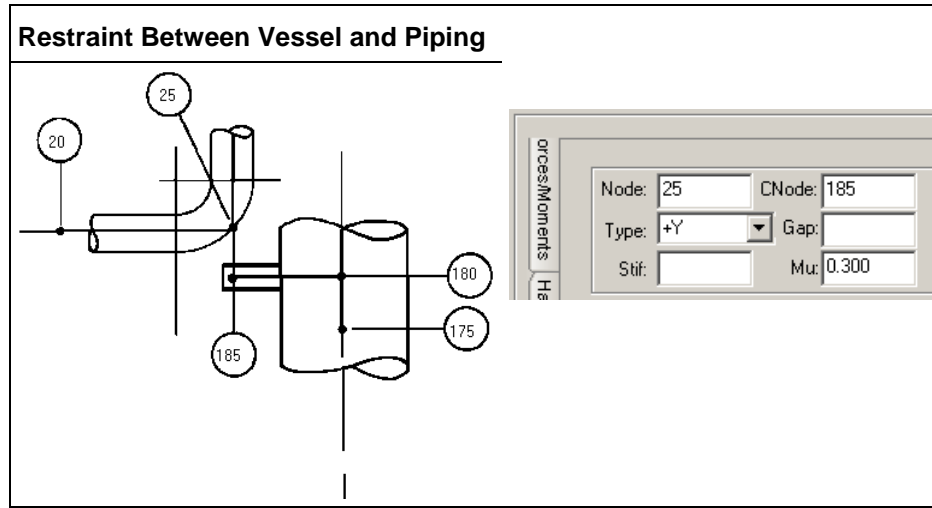


orces/Moments Ha	Node:	195	CNode:	65
	Type:	Y	Gap:	
	Stif:		Mu:	

Restraint between Vessel and Pipe Models

Use the following guidelines when modeling restraints that between vessel and pipe in CAESAR II:

- Use a restraint with a connecting node to link the pipe to the rigid element that extends from the vessel shell.
- You can specify any number of restraints between the restrained node and the connecting node.
- Restraints can be linear or nonlinear with gaps and/or friction.



Restraints on a Bend at 45°

Linear and/or non-linear restraints can act at any point on the bend curvature. Points on the bend curvature are like any other point in the piping system.

The figure below shows a bend supported vertically by a rigid rod. The rod is allowed to take tensile loads only, so it is modeled as a single-directional restraint that can move freely in the positive Y-direction.

NOTE If the positions of nodes 19 and 20 are unclear, see *Bends* (on page 11).

The line of action of the rod is shifted away from node 19. A downward force at node 15 produces a positive Z-moment about node 20 in the system as modeled, but a negative Z-moment about node 20 in real-time.

The magnitude of this moment is a function of the load and the moment arm (the amount of the shift). If this is considered significant, then a rigid element with zero weight can be placed between node 19 and the actual point of rod attachment. The restraint is then placed at the actual point of rod attachment.

90° Bend Restrained at Midpoint

Bends

Radius:

Type:

Angle 1: Node 1:

Angle 2: Node 2:

Angle 3: Node 3:

Restraints

Node: CNode:

Type: Gap:

Stif: Mu:

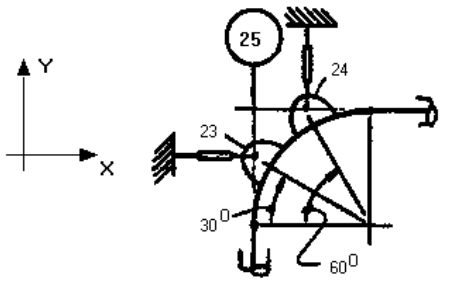
Restraints on a Bend at 30° and 60°

You can define up to three nodes at any angle on the bend curvature, presuming the points are more than 5° apart. Restraints can be modeled on any of these nodes. If necessary, one of the points can be at the 0° point on the bend, which is the bend near point.

The **To** node of the bend is placed at the tangent intersection point for geometric construction, but it is placed at the bend far point for analysis purposes. Consequently, specifying a node at the bend far-weld point generates an error.

Nodes and angles on the bend curvature can be specified in any order.

Restraints on Intermediate Points Along a Bend



Bends

Radius:

Type:

Angle 1: Node 1:

Angle 2: Node 2:

Angle 3: Node 3:

From: Name

To:

DX:

DY: ...

DZ:

Bend

Rigid

Expansion Joint

Restraints

Hangers

Nozzles

Restraints

Node: CNode:

Type: Gap:

Stif: Mu:

Node: CNode:

Type: Gap:

Stif: Mu:

Vertical Dummy Leg on Bends

You can model dummy legs on bends in several ways. The three most common methods used to model dummy legs are outlined below.

Near/Far Point Method

- Easy input
- Dummy leg acts along centerline of vertical run
- Dummy leg does not act at the proper place on the bend curvature

On Curvature Method

- Easy input
- Dummy leg acts at the proper place on the bend curvature
- Dummy leg does not act along the centerline of the vertical run

Offset Element Method

- Difficult input
- Dummy leg acts at the proper place on the bend curvature
- Dummy leg acts along centerline of vertical run

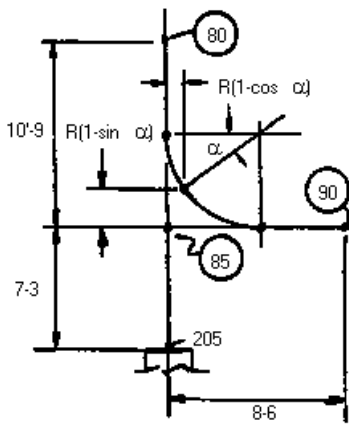
NOTE The element immediately after the bend must define the downstream side of the bend. Do not define dummy legs on the element immediately following a bend.

Dummy legs and/or any other elements attached to the bend curvature should be coded to the bend tangent intersection point. The length of the dummy leg is taken directly from the **DX**, **DY**, and **DZ** properties defined for the dummy leg of the pipe. There is no automatic alteration of the dummy leg length due to the difference between the bend tangent intersection point and the actual point on the bend curvature where the dummy leg acts. Enter the true length of the dummy leg in the **DX**, **DY**, and **DZ** boxes in the **Classic Piping Input** dialog box.

Input and output plots of the dummy leg always show it going to the bend tangent intersection point.

During error checking, the software generates a warning message for each dummy leg/bend model. Verify that the warning message description of the bend is accurate.

Vertical Dummy Leg on Bend



BEND GEOMETRY

The bend shown on the left enters from the top left corner of the control station nodes 80 to 85 and exits horizontally to the right (nodes 85 to 90). The dummy leg is attached near the 45° point on the bend. The centerline of the dummy leg should line up with the centerline of the vertical run of pipe.

From: Name
 To:
 DX:
 DY: ...
 DZ:

Bends
 Radius:
 Type:
 Angle 1: Node 1:
 Angle 2: Node 2:
 Angle 3: Node 3:

From: Name
 To:
 DX:
 DY: ...
 DZ:

NOTE Coding the bend is the same for all three methods except alpha is not required for the Near/Far Point Method.

Figure 15: Near Point Method

Dummy leg length is equal to the distance from its base to the bend tangent intersection point plus the bend radius.

Figure 16: On Curvature Method

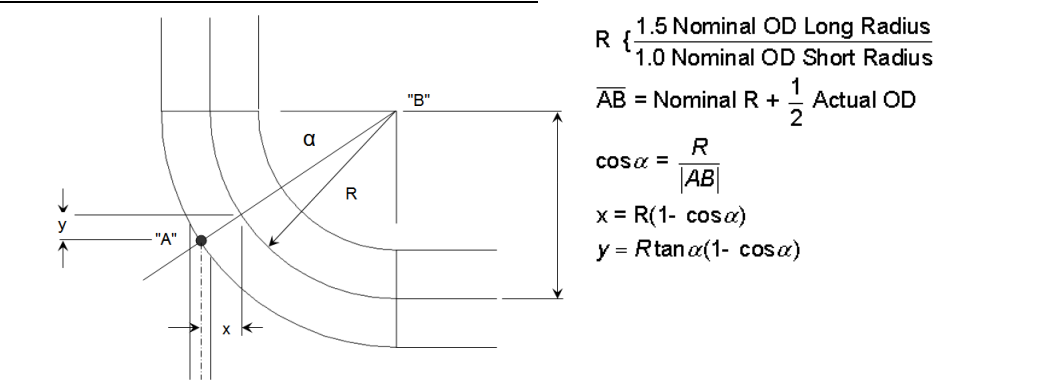
- Calculate α from the relation shown on the following page.
- Enter α on the bend specification instead of the midpoint.
- Calculate the offset distance $y = R(1 - \sin\alpha)$.
- Add the distance y to the dummy leg length.

Figure 17: Offset Element Method

- Calculate α and the offset distance x and y from the figure on the following page.
- Add x and y to the dummy leg length.
- Activate the offsets to subtract them from the dummy leg.

Vertical Leg Attachment Angle

Dummy Leg Attachment Angle Calculation



Horizontal Dummy Leg on Bends

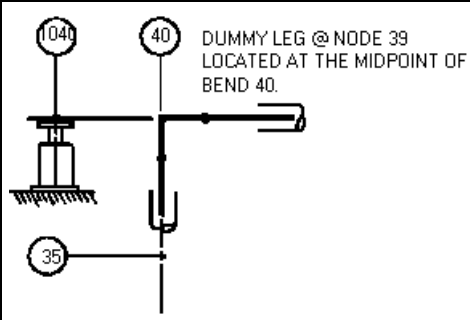
The element leaving the bend must define the downstream side of the bend. Do not define dummy legs on the element immediately following a bend.

Enter the true length of the dummy leg in the **DX**, **DY**, and **DZ** boxes in the **Classic Piping Input** dialog box.

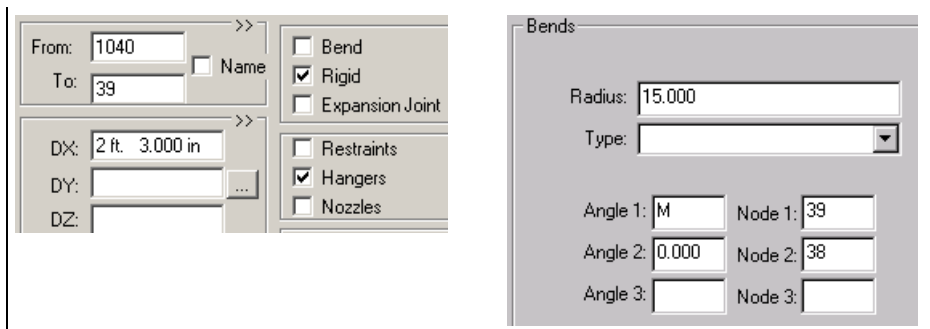
Input and output plots of the dummy leg always show the dummy leg going to the bend tangent intersection point.

During error checking, the software generates a warning message for each dummy leg/bend model. Verify that the warning message description is accurate.

Horizontal Dummy Leg on Midpoint of Bend



The dummy leg is defined as a zero weight rigid supported on one end by a spring can.



Large Rotation Rods - Basic Model

Large rotation rods are used to model relatively short rods where large orthogonal movement of the pipe causes shortening of the restraint along the original line of action.

Large rotation rods can be entered in any direction. Use the **Type** list on the **Classic Piping Input** dialog box, to select the rotation rod: **XROD**, **YROD**, or **ZROD**. After the rotation rod is selected, the restraint options change as follows:

- **Gap** changes to **Len**, which is the length of large rotation swing.
- **Mu** changes to **Fi**, which is the initial load on the restraint if used to model variable support spring hanger. (Imagine the large rotation rod as providing a bowl in which the pipe node is free to move.)

Enter large rotation rods only where they are needed. Repeatedly using large rotation rods when they are not necessary can cause the system to become unstable during the nonlinear iteration. First, analyze the system without the large rotation rods, and then add large rotation rods where horizontal movement at support points is greatest. Usually, you should add only one rod in an area at a time.

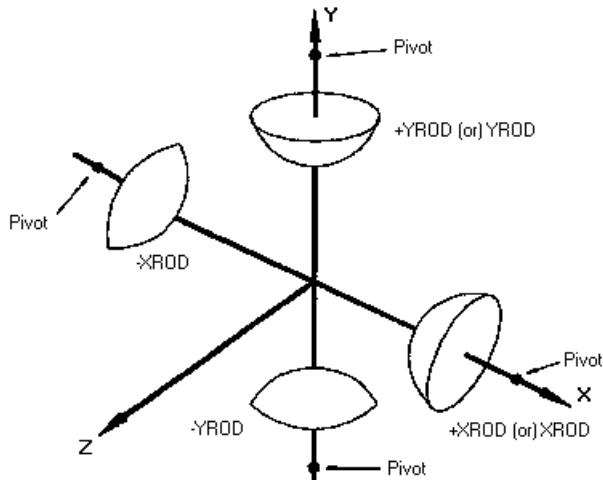
The rod angle tolerance is currently set at 1°.

Large rotation is generally considered to become significant when the angle of swing becomes greater than 5° degrees.

As for any other support, connecting nodes can be used for large rotation rods. Graphically, the connecting nodes and the restraint node do not have to be at the same point in space. There is no plot connectivity forced between large rotation rod nodes and connecting nodes.

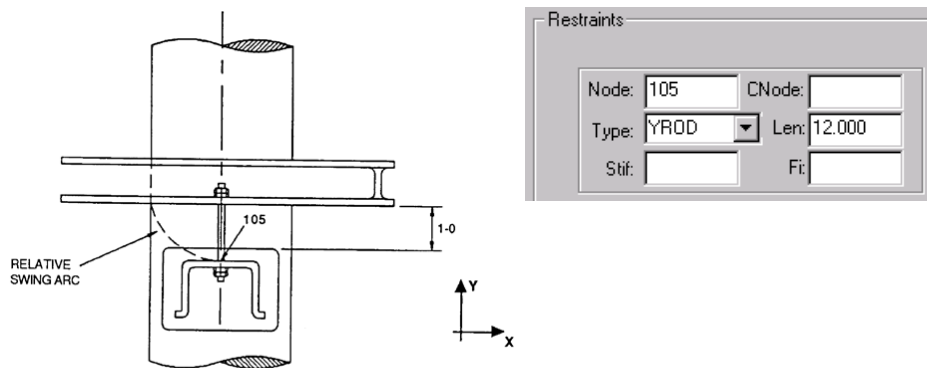
Restraints

The plus or minus signs (+/-) on the large rotation rod determine the orientation of the swing axis. A positive YROD (+YROD) is equivalent to an YROD and indicates that the concave side of the curvature is in the positive Y-direction.



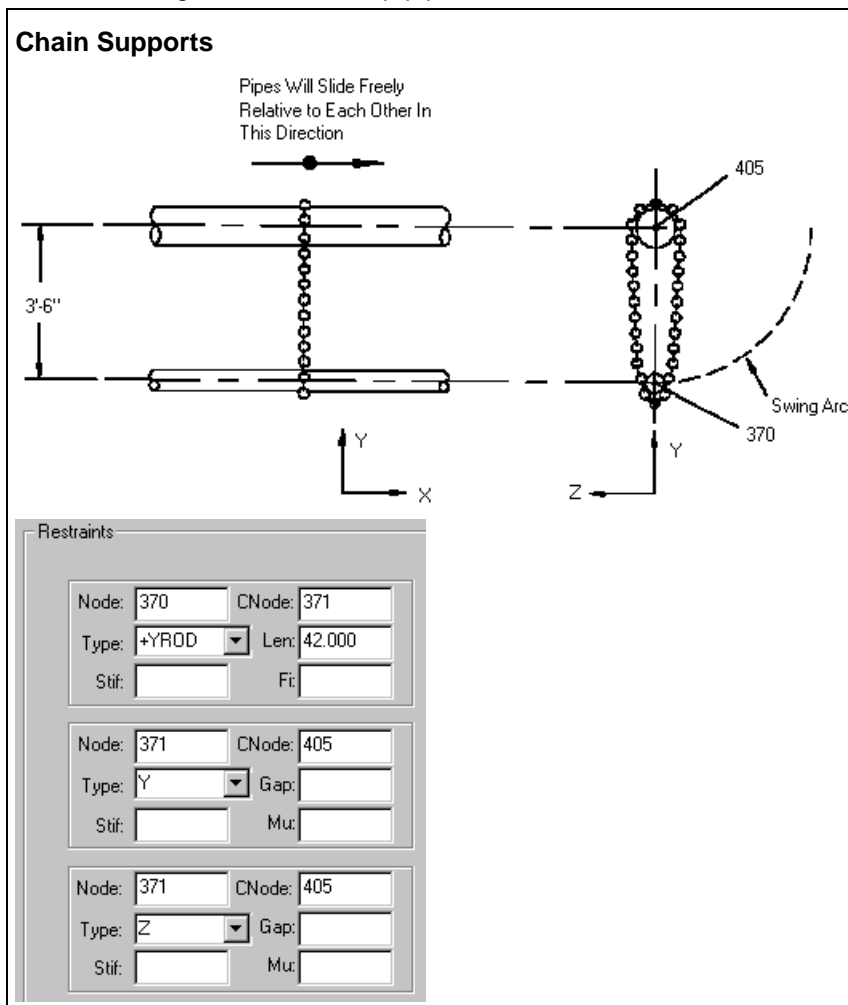
In the example below, the rod pivots about the structural steel support. There is a very short swing arm, so even a small amount of horizontal movement produces a relatively large swing. The output report for this restraint shows X- and Y-direction loads.

Large Rotation Rods:



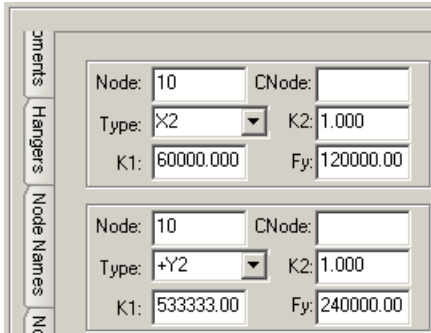
Large Rotation Rods - Chain Supports

In the model below, imagine that you want the large rotation to swing only in the plane of the chain support (the Y-Z plane). The two pipes should move freely relative to each other in the axial direction (the X-Y plane). To do this, use three restraints with connecting nodes. The first restraint is the large rotation rod with its connecting nodes. In turn, the first restraint is connected to the second and third linear restraints, allowing only Y-Z interaction between the large rotation rod connecting node and the top pipe node.



Bi-Linear Restraints

Bi-linear restraints are used most often to model soil support where some soil ultimate load bearing capacity can be calculated. Bi-linear restraints are designated by typing a **2** immediately after the direction in the restraint **Type** box.

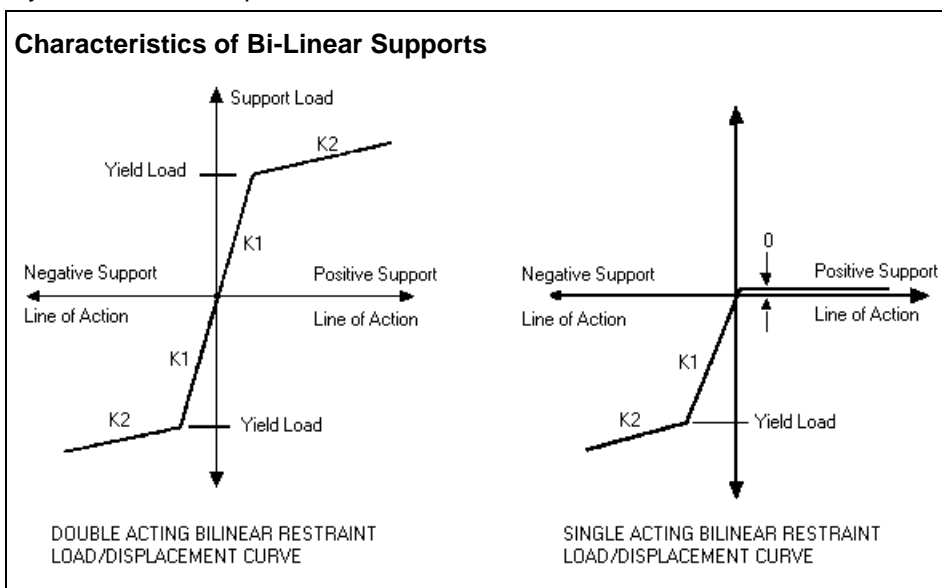


After a bi-linear spring is entered, the restraint options change as follows:

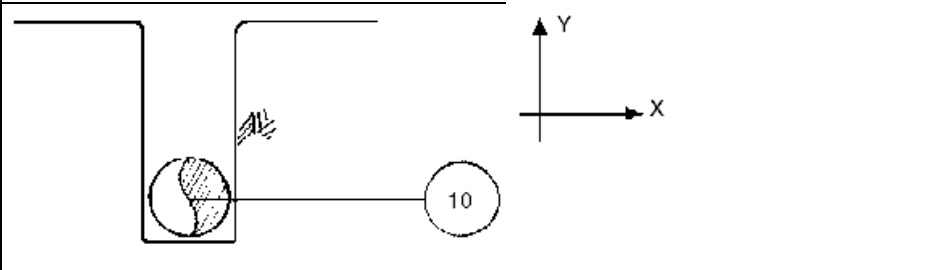
- **Stif** changes to **K1**, which is the initial stiffness.
- **Gap** changes to **K2**, which is the yield stiffness.
- **Mu** changes to **Fy**, which is the yield load.

Both the yield stiffness (**K2**) and the yield load (**Fy**) are required. If necessary, the yield stiffness can be defined as a negative value. The initial stiffness (**K1**) is optional. If it is left blank, the software presumes a rigid initial stiffness. Some sub-sea pipeline resistance tests have shown that load carrying capacity drops after the ultimate load is reached, and displacement continues.

For more information about the use of the spring types used to model underground piping systems, "Buried Pipe Modeler" in the *CAESAR II User Guide*.



Pipe in a Trench Bi-Linear Restraint Modeling



Estimated Stiffness Models

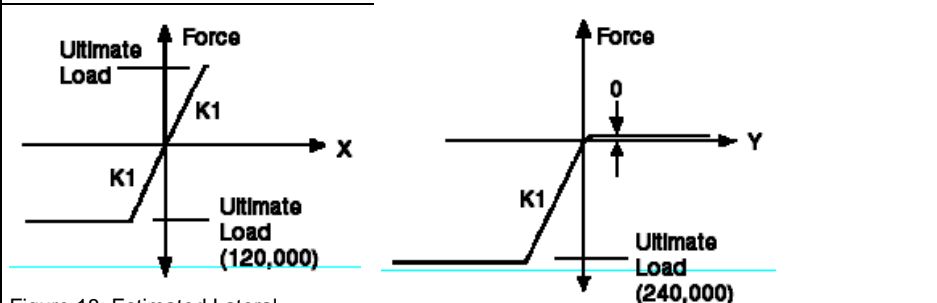


Figure 18: Estimated Lateral Stiffness: 60,000 lb./in.

Figure 19: Estimated Vertical Stiffness: 533,333 lb./in.

Comments	Node: 10	CNode:
	Type: X2	K2: 1.000
Hangers	K1: 60000.000	Fy: 120000.00
	Node: 10	CNode:
Node Names	Type: +Y2	K2: 1.000
	K1: 533333.00	Fy: 240000.00
NC		

Static Snubbers

Static snubbers are translational restraints designed to provide resistance to dynamic loads such as seismic and wind. Snubbers are activated by checking the **Snubbers Active?** Check box for the appropriate load case (the one that contains the occasional load) on the **Load Case Options** tab in the **Load Case Editor**. The figure below shows that snubbers will be active in load case L2 to restrain the pipe against the uniform load.

	Load Case Name	Output Status	Output Type	Comb Method	Snubbers Active?
L1	OPERATING CASE CONDITION 1	Keep	Disp/Force/Stress		<input type="checkbox"/>
L2	W+D1+T1+P1+U1(OPE)	Keep	Disp/Force/Stress		<input checked="" type="checkbox"/>
L3	SUSTAINED CASE CONDITION 1	Keep	Disp/Force/Stress		<input type="checkbox"/>
L4	EXPANSION CASE CONDITION 1	Keep	Disp/Force/Stress	Algebraic	<input type="checkbox"/>
L5	L2-L1(OCC)	Keep	Disp/Force/Stress	Algebraic	<input type="checkbox"/>
L6	L3+L5(OCC)	Keep	Disp/Force/Stress	Scalar	<input type="checkbox"/>

Figure 20: Load Case Options Tab

Static snubbers, or static analysis snubbers, have **SNB** following the translational direction that displays in the restraint **Type** box (located on the **Classic Piping Input** dialog box). After you enter a snubber, **Gap** and **Mu** are no longer available.

Static snubbers can be directional. That is, they can be preceded by a plus (+) or minus (-) sign. To model static snubbers, follow the steps below.

1. Run the operating case without defining a snubber.
2. Note the displacements, in all six degrees of freedom, at each snubber location.
3. In the **Classic Piping Input** dialog box, add each snubber with a distinct CNode.
4. Place the displacements from Step 1 on all snubber CNodes.
5. Modify the load cases by including **D1** everywhere that **T1** displays.
6. Activate snubbers in the **Load Case Editor** for all load cases containing occasional loads.

Plastic Hinges

The steps in setting up a plastic hinge are illustrated below. The leg from A to B is over-heated, causing bending of the B-D support leg. This example models the plastic deformation at cross-section E-E. The plastic hinge is formed between nodes 10 and 15. The expansion joint is used to provide translational and torsional rigidity at the plastic hinge junction. Two bi-linear supports are used to model rigid resistance to bending until a breakaway force (yield force) is exceeded, at which point bending is essentially free.

Plastic Hinge in a Support Leg¹

Model Status

Allowable Stresses

Bends

Dist

Axial Stif: 100000000

Trans Stif: 100000000

Bending Stif: 1.000

Torsion Stif: 100000000

Effective ID: 0.000

The yield force is computed using the following equation:

$$F_y = S_y Z (SF)$$

Where

S_y is the yield stress

Z is the section modulus

SF is the safety factor

Restraints

Node: 10	CNode: 15
Type: RX2	K2: 1.000
K1:	Fy: 5100.000

Node: 10	CNode: 15
Type: RZ2	K2: 1.000
K1:	Fy: 5100.000

¹The plastic hinge is modeled as a zero length expansion joint with rotational bi-linear restraints.

Sway Brace Assemblies

The sway brace is commonly used to allow unrestrained thermal movements while tuning the system dynamically to eliminate vibration. A sway brace resembles a spring in that it can be pre-loaded in the cold (installed) position. After pipe thermal growth, it reaches the neutral position and the load on the system in the operating condition is zero or negligible.

The sway brace is composed of a single compression spring enclosed between two movable plates. The spring is precompressed a full inch, which provides an initial force that instantaneously opposes vibration. Any movement from the sway brace neutral position is opposed by a load equal to the pre-load plus travel from the neutral position multiplied by the sway brace spring constant. After maximum allowed travel (usually 3-inches in either direction) is reached, the sway brace locks, preventing additional movement.

Manufacturers typically recommend a specific size sway brace for a given pipe nominal diameter.

A more specific sway brace selection is possible when the exact restraining force required to control the piping vibration is known. The energy necessary to control the piping is proportional to the mass, amplitude of movement, and the force causing the vibration. From this relation, the exact restraining force required to control the piping vibration can be calculated and an appropriate sway brace size selected.

After it is selected, the sway brace can be modeled in CAESAR II using a combination of a bi-linear restraint and a translational restraint:

In the event that the sway brace is to be installed in the operating condition (or the neutral position is to be adjusted in the operating position), the modeling in CAESAR II is a little more complex. In this case, before modeling the sway brace, you must analyze the piping system without the sway brace to obtain displacements from the cold to neutral operating position:

Run an analysis on the system without the sway brace to obtain the displacements from cold to operating condition. In the example illustrated below, the assumed CAESAR II-calculated displacement from cold to operating position is 0.5 inches

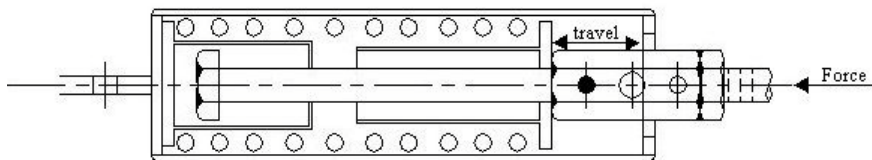
In the SUS case, the displacement D2 (vector 2) represents the pre-load in cold position. Under shutdown conditions, the pipe returns to its cold position and the brace exerts a force as previously described.

$$\text{Sustained case restraint loads on sway brace} = \text{Pre-Load} + \text{Hot Deflection} * \text{Spring Rate}$$

In OPE, the displacement allows thermal expansion, and the sway assumes neutral position exerting zero or negligible load on the pipe.

Operating case restraint loads on sway brace ≈ 0.0 (does not restrain thermal expansion)

Sway Brace Installed in Operating Condition



Sway brace opposing compression force (movement occurs after pre-load is overcome).

Restraints

Node: 10 CNode: 101
 Type: X2 K2: 150.000
 K1: Fy: 150.000

Node: 10 CNode: 101
 Type: X Gap: 3.000
 Stif: Mu:

- Spring Rate: 150 lb./in.
- Initial Loading: 150 lb.
- Allowed Movement: 3 in.
- Calculated Displacement: .5 in.

Displacements

Node 1: 101

	Vector 1	Vector 2	Vector 3
DX		0.5000	
DY			
DZ			
RX			
RY			
RZ			

NOTE Be sure to include D2 in the sustained and operating cases.