Nonconvergence in Caesar II

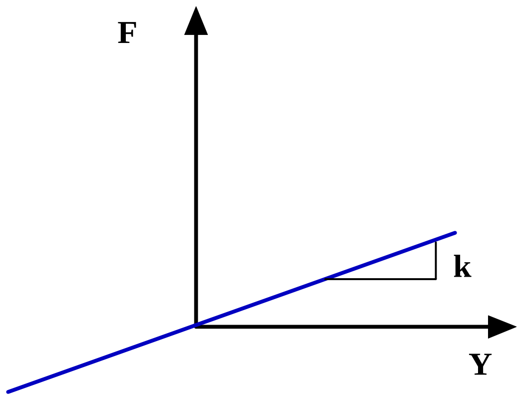
The equations in Caesar II (and all pipe flexibility analysis programs) are linear in nature and therefore are readily solved using matrix operations on computers. Difficulties with numerical methods occur when nonlinear boundary conditions exist. The computations solve a system of linear algebraic equations simultaneously until the solution does not change appreciably between successive iterations. But nonlinear boundary conditions may cause a change in the actual equations themselves between successive iterations and is the basis for all convergence problems in this type of numerical routine.

Once the iteration process begins the loads defined in the load case editor are applied to the piping and the pipe moves according to these loads, the stiffness of the pipe, and how it is restrained. When gaps are closed then the stiffness changes from zero to whatever stiffness is specified for the restraint in input. This changes the equations that Caesar II has to solve. On each successive iteration Caesar II must check the status of the piping at each nonlinear restraint location to determine whether the restraint is engaged or not engaged and then change the equations accordingly. Convergence is not achieved until every restraint in the system retains the same engaged/not-engaged state between several successive iterations (and of course the numerical solution also changes inappreciably at each node in the system as well).

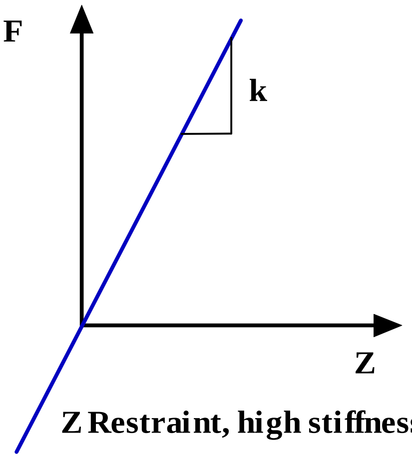
The various types of nonlinear boundary conditions, the convergence difficulties they may cause, and methods for obtaining convergence in Caesar II when encountering these difficulties are explained next.

Nonlinear Restraints

Nonlinear restraints are defined as restraints whose stiffness is not uniform along the line of action of the restraint. Uni-directional restraints (for example, pipe resting on a rack), restraints with a gap, and bi-linear restraints are all nonlinear. Whereas fully bi-directional restraints are linear. The following illustration shows force vs. displacement curves for several linear and nonlinear restraints in Caesar II. Note the slope of the curve dictates the stiffness of the restraint (rigid restraints have an almost vertical slope). All nonlinear restraints have more than one stiffness in their force vs. displacement curves while all linear restraints have a constant stiffness.

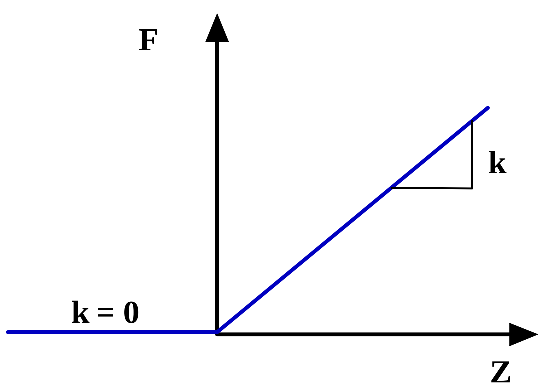


Y Restraint, low stiffness

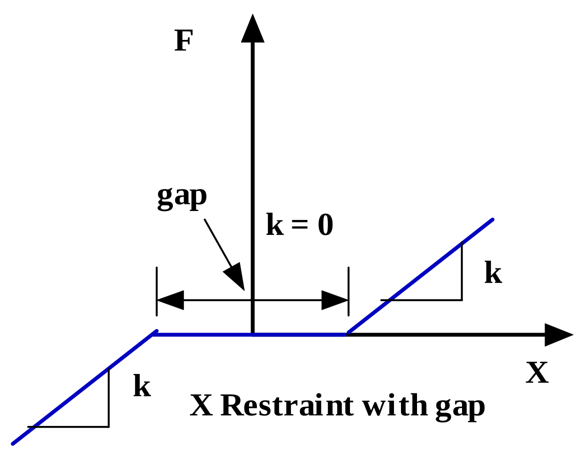


Z Restraint, high stiffness

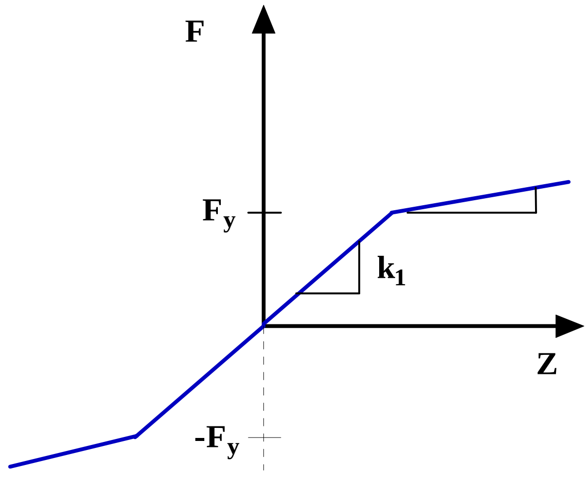
Nonlinear Restraints



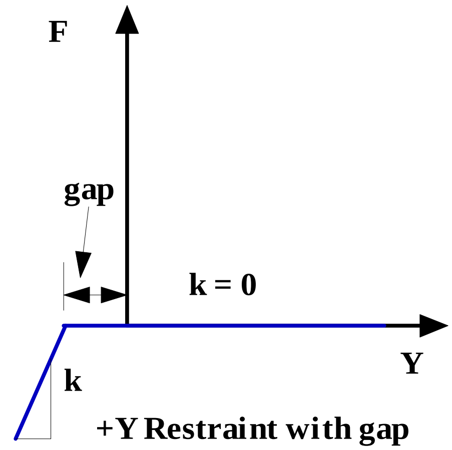
-Z Restraint



X Restraint with gap



Z2 (bi-linear) Restraint

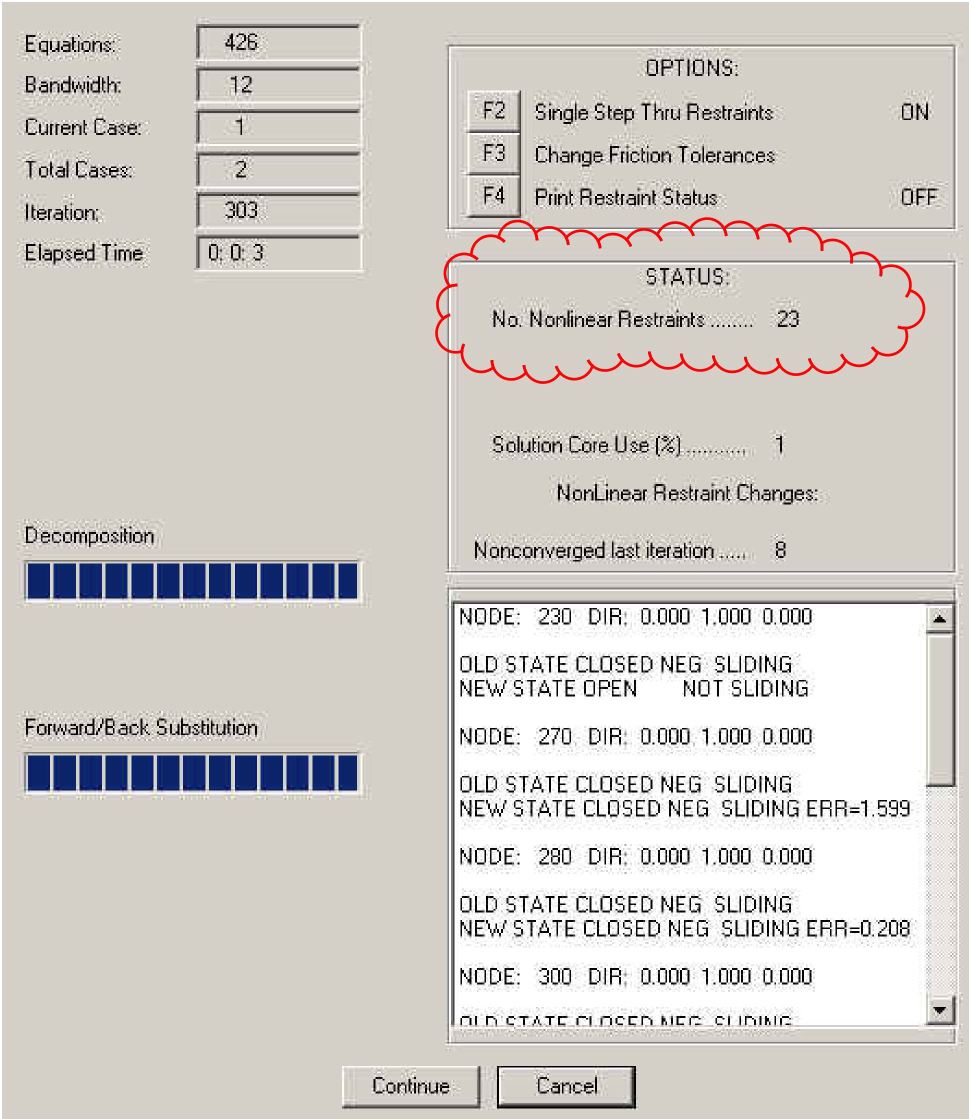


+Y Restraint with gap

Note that rotational restraints may also be linear or nonlinear in a similar manner. The stiffness in these cases relate a moment to rotation resulting in a stiffness in units of Ft-lbs/degree (length*force/angle).

Diagnosing Convergence Problems

Caesar II provides some good diagnostic tools for the user when the solution will not converge. When the solution does not converge a window called the Incore Solver will be available (it always comes up but in small converging systems this window may simply flash by on the screen) as in the illustration below.



Pressing the F2 function key on the keyboard will freeze the solution so the user can apply some diagnostics.

On the upper-left portion of the Incore Solver window is some general information about the solution. The most important of these is the Current Case and Total Cases. The current case is the load case that is being solved for and the total cases is the total number of native (non-combination) load cases in the load case editor. The solver performs the analysis on the load cases in the order they are entered in the load case editor.

The middle-right side of the window gives information on the total number of nonlinear restraints in the model (23 in the figure above) as well as the number of these restraints that did not converge on the previous iteration (8 in this case). On some models the number of nonconverged restraints may fluctuate and it is recommended to attempt to freeze the analysis when this number is lowest. Once frozen, pressing the Continue button will perform a single iteration of the analysis, so it is a simple matter to press

Continue several times to obtain a minimum number of non-converged restraints. Note that pressing the F2 function key (or the F2 button in the Incore Solver window) will continue the analysis normally.

In the white list box in the lower-right section of the Incore Solver is information on individual non-converged restraints. The node number of the restraint is given as is the direction cosine of the line of action of the restraint. This provides easy identification of which restraint is causing the problem, which is useful when there are several nonlinear restraints at the same node and only one or two of them are not converging. In the example above, of the four restraints that can be seen without scrolling the list box, only Y restraints are not converging (these are likely to be +Y restraints).

Below the direction cosine of each non-converged restraint is the state of the restraint. Open means that the restraint is not in contact with the pipe and the stiffness at this location is zero. Closed indicates that the restraint is in contact with the pipe and that its stiffness is being used in the analysis. Old State is the previous iteration and New State is the current iteration. A state of Sliding/Not Sliding indicates the friction state of the restraint, which is discussed further below.

Make a note of all the non-converged restraint node numbers and perhaps their direction cosines and then click the Cancel button to terminate this analysis (limited-run users are not charged a run when terminating the analysis in this manner).

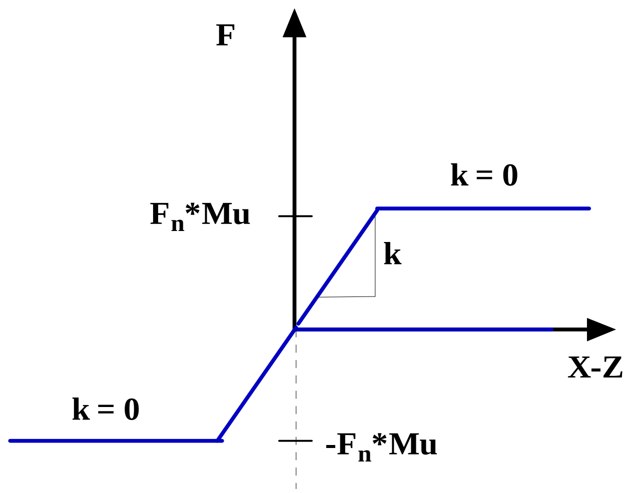
Next, return to input and review the non-converged restraints one-by-one. It is often useful to review the model graphically with the node numbers turned on. In this way it is quickly determined whether the non-converged restraints are clustered geometrically close to one another or if they are more evenly spread throughout the model. When there are large numbers of nonlinear restraints in a model it can be very difficult to determine the best way to proceed. Determining the best method for obtaining convergence is as much art as it is science since we cannot know exactly what is happening numerically in the solution of so many simultaneous equations.

Sometimes, especially when there are a small number of non-converged restraints, a simple change to the model may allow complete convergence. Small changes to operating temperature, for example increasing it by 5 F, may cause the system to converge (this is if it is the operating case that is not converging. For sustained case non-convergence the temperature will not affect the solution).

Convergence Problems due to Friction

Numerical modeling of friction is a difficult process at best and it is notorious for causing non-convergence in flexibility analysis programs. Friction always acts in a plane that is normal to the line of action of the restraint it is applied to. For example, on a +Y restraint the friction, if any, will act in the X-Z plane.

When pipe begins to move due to thermal expansion and other loads, static friction opposes this motion and the force between the pipe and the restraint increases (see figure below). As the load increases, usually due to heating of the pipe, it may eventually reach a point where the pipe will break free from friction and begin sliding. This breakaway force is equal to the normal force (the force along the line of action of the restraint) times the friction coefficient, Mu. In reality if the heating is gradual the pipe may slide a small amount and relieve a small amount of the friction force, but then increase again until it once again exceeds the breakaway force. This may occur many times with such ratcheting repeating until the pipe temperature reaches steady state and the system is in equilibrium.



+Y Restraint with friction

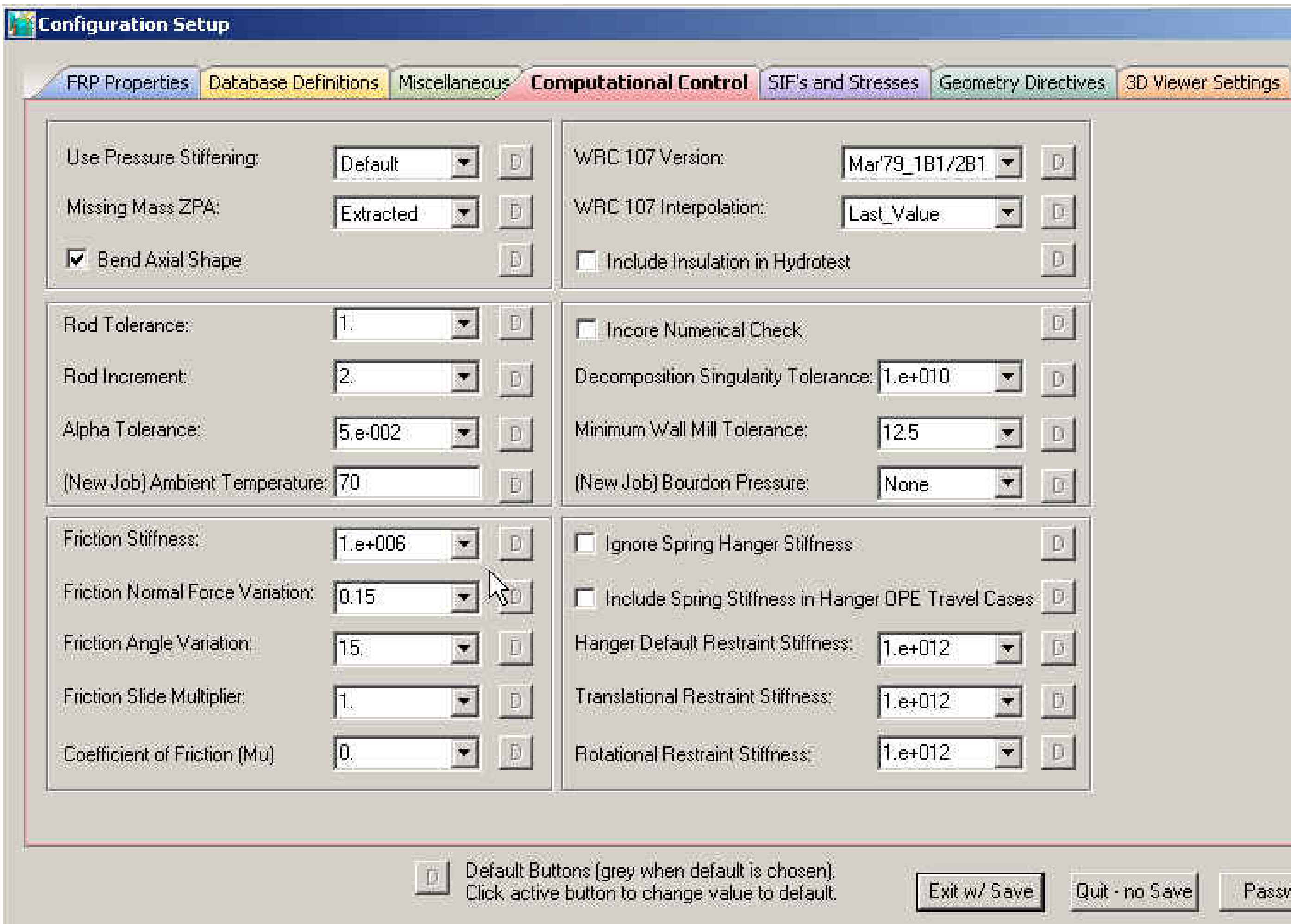
At this point, it is important to note that the friction force is still opposing further motion, even when in equilibrium. In reality when the pipe is sliding this force is equal to the dynamic friction coefficient times the normal force, which is slightly smaller than the static friction coefficient times the normal force. It is a common misconception that once the system reaches equilibrium that there is no further friction force on the system. This leads some to believe that friction is a transient phenomenon that only occurs during the start-up or shut-down cycle of a piping system. This is definitely not true! From the time the friction breakaway force was first exceeded until the system reaches equilibrium the force opposing motion is between the static and dynamic breakaway force, which is a fairly small range of force variability.

Caesar II must calculate three primary quantities to simulate friction at a given node where a friction coefficient is defined on a restraint. First, the direction of pipe motion must be calculated. This is easy to do from the resultant displacements on a given iteration. The angle that the resultant displacement vector in the friction plane makes with one of the axes in this plane is known as the friction angle. Then a friction restraint with a stiffness (given by the friction stiffness in Configure/Setup) is applied opposite this direction of motion. The normal force is multiplied by the user-defined friction coefficient, (Mu on the restraint definition) to determine the restraint breakaway force. The equation for the given node is then changed to reflect the direction and stiffness of the friction restraint for the next iteration.

For the node to be converged, the friction breakaway force, the state of the friction restraint, and the direction of the friction restraint must all be within certain tolerances between the current and last iteration. Sometimes, especially with many restraints with friction, the solution will "bounce" between a situation where the pipe is sliding (remove the friction restraint and replace it with the friction force opposing motion) and not sliding (put in a friction restraint opposing motion). This is a discrete change in the boundary condition, which can result in an infinite loop since the equations for the node simply change back and forth repeatedly.

Obtaining Convergence on Systems with Friction

Fortunately, there are several simple settings that can affect the way that friction is modeled in Caesar II. When a solution will not converge there are three items that can be changed that will determine whether or not the solution will be completed. All of these items may be accessed in the Caesar II Configuration File (shown below).



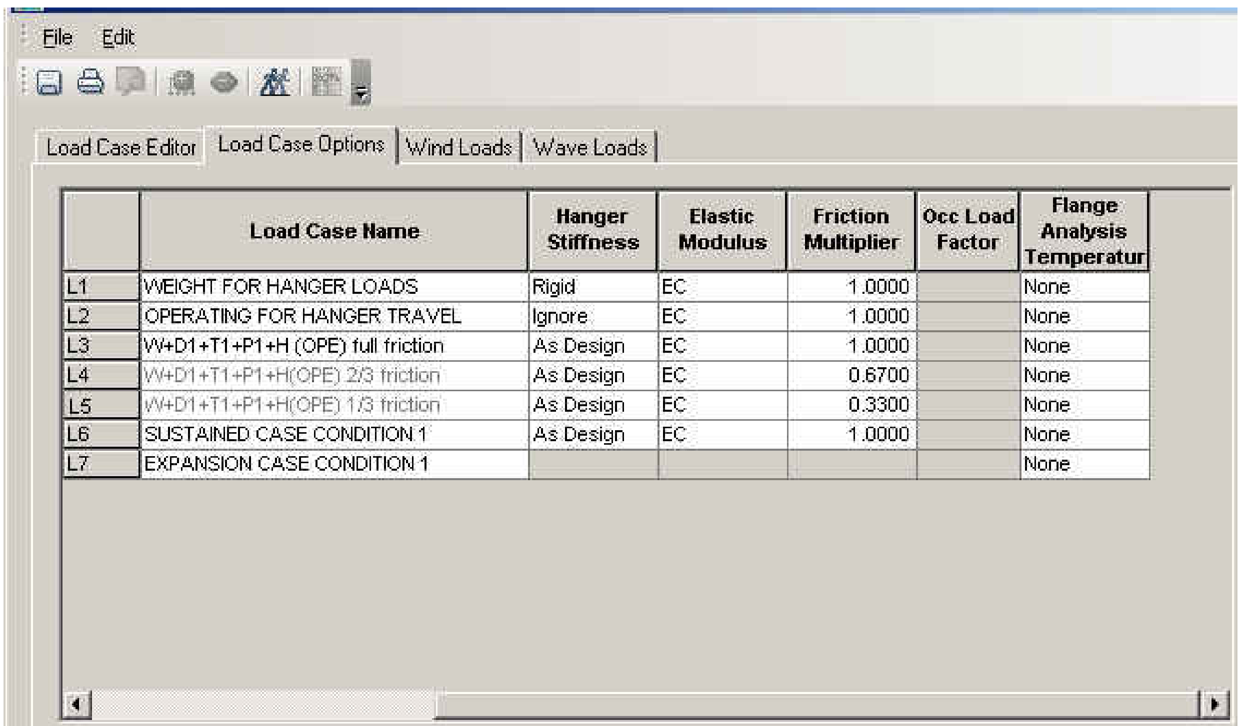
As previously discussed, the friction stiffness is set in the Configuration file and is seen just above and left of the arrow in the illustration above. The default is one million pounds per inch in the English system of units. Making this stiffness lower results in a larger displacement of the pipe against the friction restraint without exceeding the breakaway force. This effectively makes the friction effect "softer." This then reduces the variability of the friction force against the motion of the pipe between successive iterations and will have a large impact on convergence.

There are also two friction calculation tolerances available in the Configuration file. The first is the friction normal force variation. As implied, this allows successive values of the normal force, and hence the friction breakaway force to have some variability and therefore improve convergence. The second is the friction angle variation, which is the number of degrees of change between successive iterations that the friction vector (either restraint or force) can have. Increasing either of these values will allow more variability in the friction solution between iterations and giving a better chance of convergence.

At this juncture it is important to note that reducing friction stiffness, and increasing the normal force and angle variations will increase the error in the results. It should also be noted that the choice of friction coefficient on the restraint definition can be difficult to accurately determine. The friction coefficient is a function of the surface roughness of both the restraint and pipe, which can have large variations due to moisture, corrosion, pitting and other defects, and manufacturing processes.

Determination of the system's sensitivity to friction can be determined using Caesar II and will help the engineer determine, at least subjectively, the extent of error introduced by opening friction tolerances in the configuration file to obtain convergence.

The Caesar II Load Case Editor provides an item called the Friction Multiplier, which can be found under the Load Case Options tab (see figure below).



	Load Case Name	Hanger Stiffness	Elastic Modulus	Friction Multiplier	Occ Load Factor	Flange Analysis Temperatur
L1	WEIGHT FOR HANGER LOADS	Rigid	EC	1.0000		None
L2	OPERATING FOR HANGER TRAVEL	Ignore	EC	1.0000		None
L3	W+D1+T1+P1+H (OPE) full friction	As Design	EC	1.0000		None
L4	W+D1+T1+P1+H(OPE) 2/3 friction	As Design	EC	0.6700		None
L5	W+D1+T1+P1+H(OPE) 1/3 friction	As Design	EC	0.3300		None
L6	SUSTAINED CASE CONDITION 1	As Design	EC	1.0000		None
L7	EXPANSION CASE CONDITION 1					None

A good comparison can be made with three different values of the friction coefficient by creating two additional OPE load cases with identical loads to the primary operating load case. Then change the friction multiplier on these two new load cases to .67 and .33 respectively. The friction multiplier will be applied to all the Mu values in the model. Then a direct comparison between these three operating cases can be made in the Restraint Summary report in output. Any sensitive equipment connections or restraint loads can then be compared with one-third, two-thirds, and full friction applied. This comparison should give the engineer a good understanding of the effects of friction on the system. This then provides a basis to estimate the effects of changing the friction tolerances in the Configuration File to obtain convergence.

Convergence Problems due to Nonlinear Restraints

Initially, the piping system is said to be in a neutral state, which is simply the geometrical representation of how the system was input. This means that for bi-directional restraints with gaps the pipe is resting with an equal gap on each side and is essentially unrestrained. For uni-directional restraints the pipe is initially in contact with the restraint, but with zero load since no loads are being applied to the piping system. For a uni-directional restraint with a gap the pipe is resting where the restraint would be if it had no gap, but is not in contact with the restraint.

Occasionally, a restraint will be in a numerical situation where the solution will change back and forth due to the continual changes in the equations. At this point the solution is in an infinite loop and will never converge. The analysis must be terminated and the input somehow changed to avoid this numerical methods problem.

Linearization of Non-converged Restraints

A very effective method for obtaining convergence is the changing of one or more restraints from nonlinear to linear. Linear restraints do not have convergence issues.

If the restraint is bi-directional with a gap it is often useful to either remove the gap (linearizing the restraint) or change the gap dimension (remaining nonlinear, but changing the solution domain). Often, engineers model restraints with very small gaps to accommodate the diameter growth of pipe due to thermal expansion. Although you should not change the design of such gaps, often these small gaps are unneccessary when modeling a system for pipe flexibility analysis and the resultant error in the results should be tolerably small. In such cases removal of gaps, at least on non-converged restraints, will allow the analysis to converge.

If the restraint is uni-directional then it can be linearized by changing it to bi-directional. In this way a +Y restraint can be changed to a Y restraint, a –RZ restraint can be changed to an RZ restraint, and so on.

Bi-linear restraints are rarely non-convergent, but if they are, a small change in the breakaway force, Fy, or breakaway torque (on rotational bi-linears) may accomplish convergence. Altering the K1 and K2 values of stiffness may also help. Since bi-linear restraints are only used in unusual circumstances to model a very specific type of behavior (usually soil modeling or hinge joint modeling) it is not recommended in most circumstances to completely linearize these restraints.

If the non-converged restraints are clustered in a relatively small geometrical area of the system then linearization of one restraint near the middle of the cluster will sometimes allow convergence. If that doesn't work, linearization of two restraints near the opposite edges of the area of non-convergence will often do the trick. Experimentation with the location of linearization of restraints will sometimes be required before the solution can be obtained and in such systems there is no simpler method.

It should be mentioned at this point that if the model also contains friction at the locations of nonlinear restraints, then the friction problem should be dealt with first, prior to linearizing restraints (see discussion that follows).

It is important to realize that any time a nonlinear restraint is linearized to obtain convergence that error will be introduced into the results, unless of course the design is changed to accommodate the use of the new linear restraint. It is prudent therefore to assess, at least subjectively, the amount of error introduced. The simplest method for doing this is to review the restraint report in output once convergence is obtained. If the force on a formerly non-converged restraint is relatively small then so will the error be. But if the force on a formerly non-converged restraint is moderate to high then the error may propogate far beyond the node point of the restraint. In such cases a design change may be in order for such a location to actually use the same restraint that was modeled in Caesar II.

Convergence Difficulties due to Large Rotation Rod Restraints

Large rotation rods are primarily used in Caesar II when rotations of rod hangers exceed two or three degrees. This is to maintain the length of the restraint and force the pipe to move in an arc. Without use of the large rotation rod restraint the pipe is not forced to move in an arc because the degrees of freedom are de-coupled in pipe flexibility analysis.

There are two tolerances associated with the convergence criteria for large rotation rods. The first is the Rod Increment, which is defined as the maximum amount of change in the angle of the rod rotation between successive iterations. The default is 1.0 degree, but this number can be decreased in difficult to converge models. This forces the rod to not change its rotation by more than the rod increment between iterations, which can help convergence. The second tolerance is called the Rod Tolerance, which is defined as the maximum change in rod rotation between iterations for convergence to occur. This differs from the rod increment in that this is the actual tolerance for the angle change to allow convergence, while the increment is the maximum change allowed by the program. If the rod wants to rotate more than the rod increment then the increment is invoked instead.

Systems with long rods or many of them are difficult to converge because while some rods are converging this may cause others in the system to move away from convergence. It is generally a good idea to add large rotation rods only in a system that is truly experiencing large rotations. If small rotations are seen in the system then the use of large rotation rods is not warranted.

Both the rod tolerance and the rod increment are available in the Caesar II Configuration File.