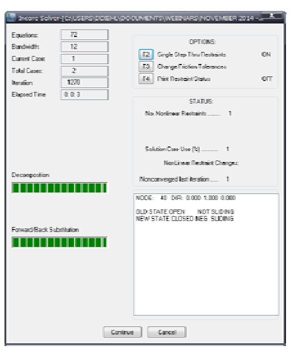
CAU2015

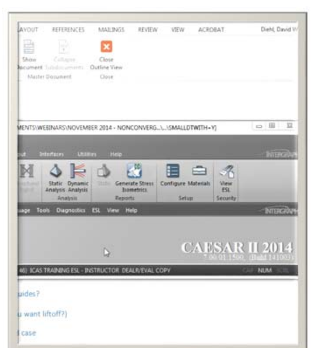
Failure to Launch

Resolving nonlinear convergence issues in CAESAR II

INTERGRAPH

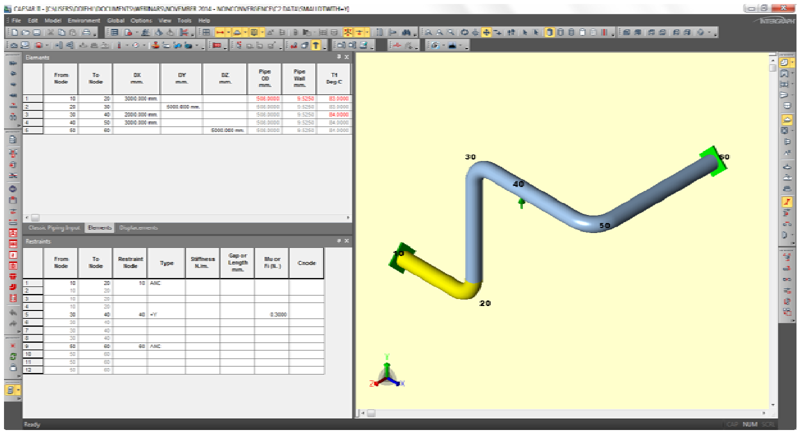
Do you ever see this screen in CAESAR II?





INTERGRAPH

Here’s the model that produced those “results”:



INTERGRAPH

BACKGROUND

Nonlinear boundary conditions and CAESAR II

- The piping codes say little about nonlinear support conditions in piping systems.
- B31.3 recently added two references – in appendices
  - ~~Appendix P~~ - ALTERNATIVE RULES FOR EVALUATING STRESS RANGE: "P300 GENERAL (a) This Appendix provides alternative rules for evaluating the stress range in piping systems. … The method is more comprehensive than that provided in Chapter II and is more suitable for computer analysis of piping systems, including nonlinear effects such as pipes lifting off supports."

    Removed from the 2014 Edition
  - Appendix S - PIPING SYSTEM STRESS ANALYSIS EXAMPLES:S302 Example 2: Anticipated Sustained Conditions Considering Pipe Lift-off
- CAESAR II has always accommodated nonlinear restraint conditions
  - Modeling (e.g., +Y, gaps, friction, rod swing)
  - Analysis
    - Codes: T1 (EXP)
    - CAESAR II: L1-L2 (EXP)

INTERGRAPH

What is a nonlinear restraint or condition in CAESAR II?

- CAESAR II uses F=KX to determine the position of every node in the piping system
  - F is the load vector under evaluation
  - K is the system and boundary condition stiffness
- K is constant

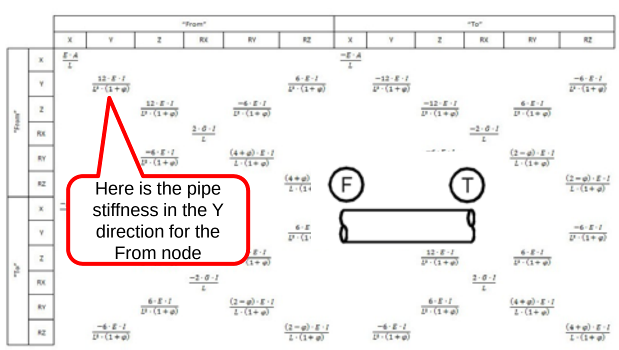
INTERGRAPH

What is a nonlinear restraint or condition in CAESAR II?

- Not all supports respond in a linear fashion. Examples include:
  - A resting support (+Y) that may lift off
  - A gap on a guide
  - Friction on a support is doubly nonlinear (magnitude and direction)
    - Friction magnitude may have the support stick or slip
    - With slip, the friction force is added to F and opposite the slip direction
- CAESAR II tests the solution to assure that all nonlinear conditions are satisfied for each load case
- CAESAR II will iterate until all nonlinear conditions are consistent using a linear K (& adjusted F)
  - Some programs just stop
  - CAESAR II has tolerances for some of these checks

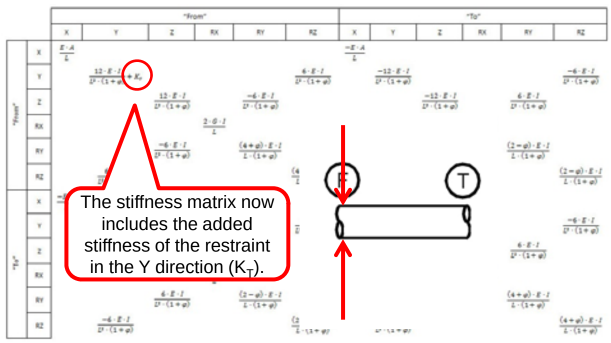
INTERGRAPH

Here's the stiffness matrix for a pipe element



INTERGRAPH

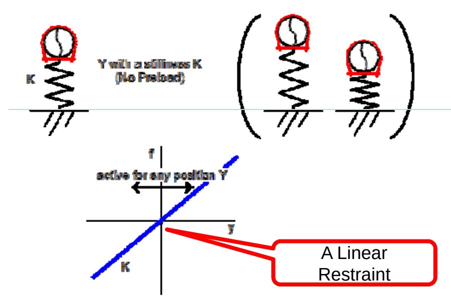
Now, include the Y restraint



INTERGRAPH

This added stiffness is constant…

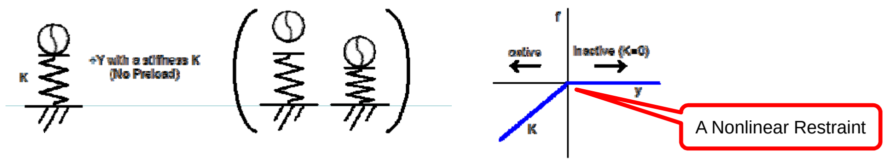
- The same restraint stiffness (KT) is effective for any Y position of this node.
- A constant value for stiffness K models a linear response.



INTERGRAPH

...But it need not be constant

- Here, the restraint stiffness (K) added to the system stiffness matrix is a function of the node's position.
- This changing stiffness is a nonlinear boundary condition.
- Nonlinear restraints require special treatment in the stiffness matrix.



INTERGRAPH

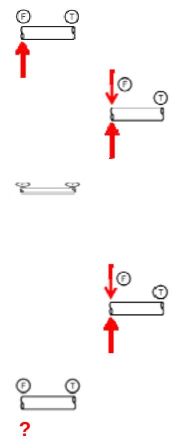
CAESAR II will adjust the stiffness matrix to suit the current load conditions

- Depending on the calculated, final position of the restraint node, the program will check the restraint stiffness defined at that position.
- If the stiffness used in the analysis matches the defined stiffness, the solution – for this restraint – is correct. If the defined stiffness for that position is incorrect, the program will update the system's global stiffness matrix with the proper restraint stiffness, and the process continues.
- CAESAR II will continue until all the nonlinear conditions applied in the global stiffness matrix are consistent with the calculated position.
- This process restarts for each basic load case in the analysis.

INTERGRAPH

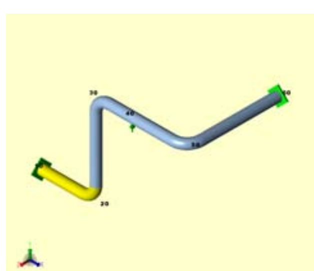
CAESAR II will adjust the stiffness matrix (assume and test)

- Here is a common "resting" support, a +Y restraint in CAESAR II terms:
- Initially, the restraint will be included in the model so the stiffness matrix will reflect this:
- If the solution shows a -Y load on that restraint, this support is modeled correctly – for this load case. If not, the program resets the stiffness matrix to appear as this:
- If this updated solution shows a +Y deflection, this support is modeled correctly – for this load case. If not, the program resets the stiffness matrix to appear as this:
- …and this iteration continues – for all the nonlinear restraints and for each load case analyzed – until the stiffness matrix is consistent with the "final" position.



INTERGRAPH

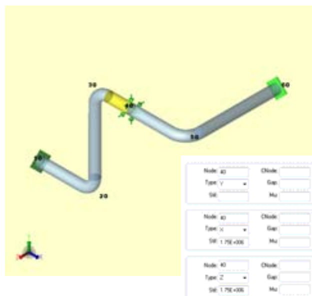
Nonlinear solution in CAESAR II – let's look at the model in the opening slide

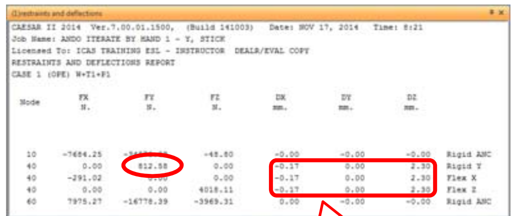


INTERGRAPH

Nonlinear solution in CAESAR II – let's look at the model in the opening slide

- Initial program "guess" – friction is sufficient to hold the pipe from horizontal movement – "stick" (include rigid restraint in Y and friction stiffness in X & Z at 40)





(1)restraints and deflections

CAESAR II 2014 Ver.7.00.01.1500, (Build 141003) Date: NOV 17, 2014 Time: 8:21
Job Name: ANDO ITERATE BY HAND 1 - Y, STICK
Licensed To: ICAS TRAINING ESL - INSTRUCTOR DEALR/EVAL COPY
RESTRAINTS AND DEFLECTIONS REPORT
CASE 1 (OPE) W+T1+P1

Node	FX N.	FY N.	FZ N.	DX mm.	DY mm.	DZ mm.	
10	-7684.25	[illegible]	-48.80	-0.00	-0.00	-0.00	Rigid ANC
40	0.00	812.58	0.00	-0.17	0.00	2.30	Rigid Y
40	-291.02	0.00	0.00	-0.17	0.00	2.30	Flex X
40	0.00	0.00	4018.11	-0.17	0.00	2.30	Flex Z
60	7975.27	-16778.39	-3969.31	0.00	-0.00	-0.00	Rigid ANC

(Although "stick", note deflection at 40)

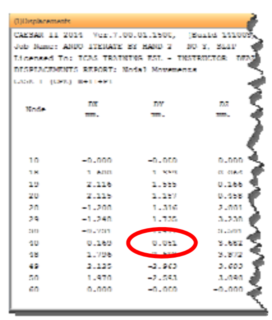
- +Y load @ 40 indicates liftoff
- Update the linear assumption(s) and try again

© Intergraph 2015

INTERGRAPH

Nonlinear solution in CAESAR II – let's look at the model in the opening slide

- Second try – liftoff will be modeled with no restraint (or friction) at 40



- -Y deflection indicates that the Y restraint should be included
- Back to square one! No convergence here.

© Intergraph 2015

INTERGRAPH

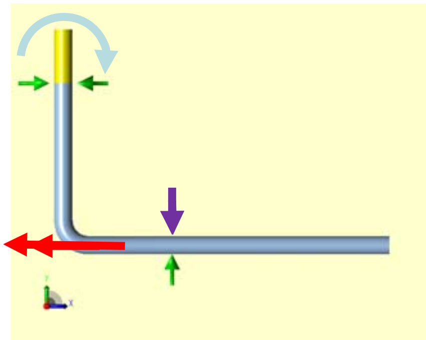
TOOLS TO RESOLVE THE ISSUE IN CAESAR II

It's your model

- Are your supports properly located? (Do you want liftoff?)
- If you are using friction, is your normal load dependable?
  - Do you always have deadweight in your load case?
  - Do you want friction on guides?
  - Is there full friction on gapped restraints?
- Do you need those “construction” gaps on guides?

Interaction of nonlinear conditions may cause looping

- Guide with a gap on a vertical run over a horizontal with a +Y and friction:
  - Thermal strain on horizontal run overcomes friction (slides in –X)
  - Closing gap puts a bending moment on horizontal which increases normal load on +Y
  - Increased friction reduces slide and gap does not close
  - Moment is eliminated and +Y normal load drops; original strain returns
  - … and the cycle continues



INTERGRAPH

A few approaches to resolving nonconvergence

- It's usually a "tipping point" – a small movement or a small load change may activate/deactivate the support model
- Take what you get – if load case 8 does not converge, you can reset load cases to generate output for the first 7 cases

INTERGRAPH

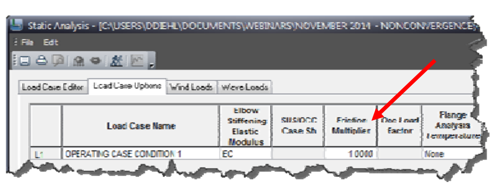
A few approaches to resolving nonconvergence

- …
- Focus on the load case that is giving the trouble
- Identify a non-converging, nonlinear condition and make it linear
  - Simple active/inactive supports are the best candidates (e.g., +Y)
  - Choose one and make one linear
  - If that converges, now you can check the "error" introduced by that change (remember, it is usually small)
  - If that does not converge linearize another nonlinear condition
- Document these model adjustments

INTERGRAPH

A few approaches to resolving nonconvergence

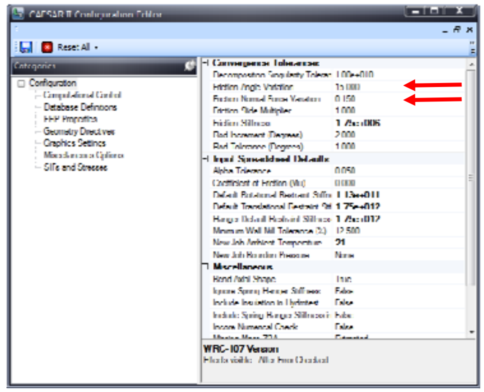
- …
- …
- When friction is a cause:
  - Remove or reduce friction for that load case (using the Load Case Editor) by changing the Friction Multiplier (*Friction Load = Friction Multiplier * Mu * Normal Load*)
  - You could also remove friction from a single restraint but that will not provide the same strain distribution.
  - Change friction tolerance – either in the program Configuration or "on the fly" in the analysis screen



INTERGRAPH

Convergence tolerance in the Configuration file

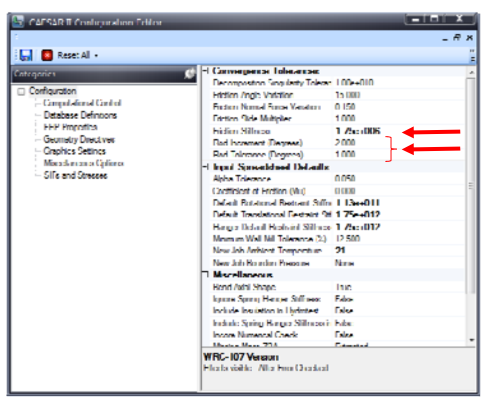
- Friction angle variation
  - If the intended movement is within 15 degrees of the existing friction load vector, the condition is “converged”
- Friction normal force variation
  - If the normal load does not change more than 15% between the current and previous iteration, the condition is “converged”



INTERGRAPH

Convergence tolerance in the Configuration file

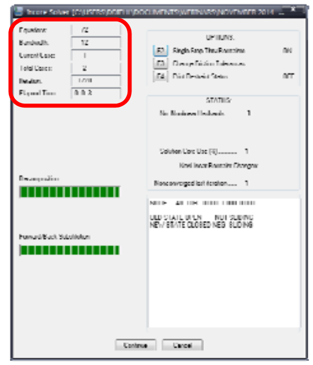
- …
- Friction stiffness
  - Sets the stiffness used for restraints modeling sticking friction
  - Not the same as rigid restraint stiffness
- Rod (e.g., +YROD) controls
  - Rod increment sets the maximum change in rod angle between iterations
  - If the rod angle changes less than rod tolerance between iterations, the condition is “converged”



INTERGRAPH

The non-convergence window – Solution Statistics

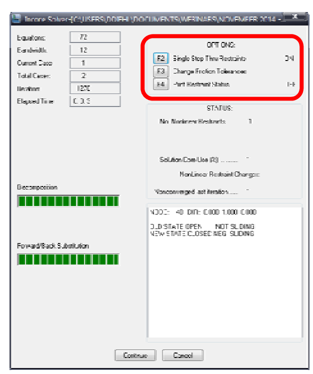
- This window appears for each CAESAR II static analysis but flashes so quickly, you would not notice.
- Non-convergence, however, has this screen remain.
- Equations and Bandwidth characterize the size of the stiffness matrix (number of nodes and closed loops)
- Keep an eye on the Current Case as that is your static load case that is causing the difficulties here
- Iteration shows how many times the program has tried to resolve all the nonlinear boundary conditions for this load case using a set of linear assumptions
  - Here, after 1270 attempts and no acceptable solution, this model will NOT converge



INTERGRAPH

The non-convergence window – Interactive Control

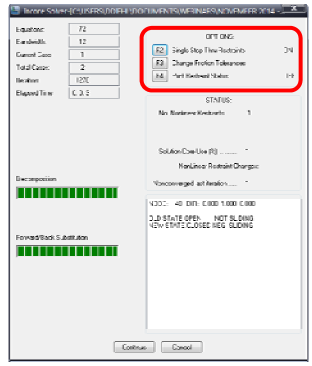
- Controls are available for the convergence process
- Press F2 or click on the "F2" button to Single Step Thru Restraints. This action will have the program display the restraints that remain incorrect and pause after each iteration. Use this to learn more about your model.
  - Click on Continue at the bottom of the window to move to the next iteration
  - Clicking "F2" again will toggle off the single step pause



INTERGRAPH

The non-convergence window – Interactive Control

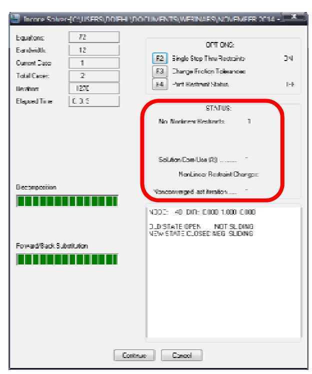
- Controls are available for the convergence process
- You can Change the current Friction Tolerances by pressing F3.
  - Adjust acceptable Normal Load Variation (default = 15%) and Friction Angle (Default = 15 degrees).
  - This may be useful in ending the current iteration
  - I discourage using this for a final analysis. Results may change based on what iteration you instigate the change. (Change the program configuration instead.)
- Print Restraint Status (F4) may consume a lot of paper. Refer to the window in the lower right instead.



INTERGRAPH

The non-convergence window – Status

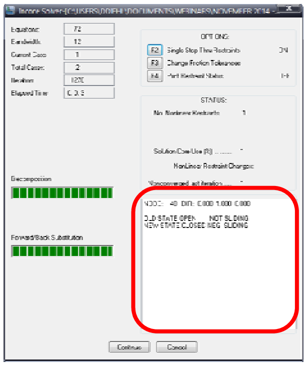
- The status area displays the “density” of the nonlinear conditions in this model
- No. of Nonlinear Restraints shows the total number of nonlinear conditions in the model
- Non-converged last iteration shows the conditions that remain unresolved
  - Here there is a single condition
  - Oftentimes you will see this cycle (e.g., 4, then 3, then 5, and repeat) where several conditions interact.



INTERGRAPH

The non-convergence window – NonLinear Restraint Changes

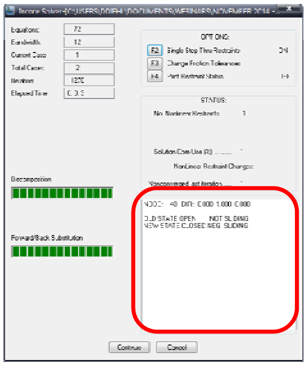
- This window displays all the unresolved nonlinear conditions
- There is much to learn from this window
- Node number of the uncertain nonlinear condition is listed
- Followed by the nonlinear direction (X,Y,Z)
  [here: DIR: 0.000 1.000 0.000 indicates a Y restraint]



INTERGRAPH

The non-convergence window – NonLinear Restraint Changes

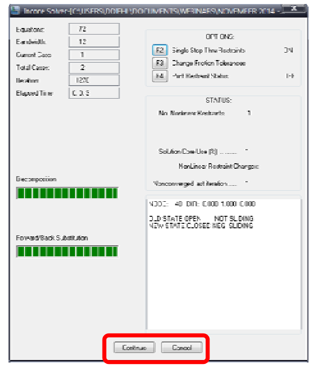
- There is much to learn from this window
- After the line identifying the restraint, two lines display the model change that requires another iteration:
  - OLD STATE / NEW STATE
  - OPEN / CLOSED – OPEN indicates the linear restraint is excluded from the iteration, CLOSED indicates that the linear restraint is included in the iteration
  - POS / NEG shows the direction of the current restraint action
  - SLIDING / NOT SLIDING determines the influence of friction
    - SLIDING can be viewed as applying a friction force opposite the direction of sliding
    - NOT SLIDING adds two restraints orthogonal to the calculated normal load
    - SLIDING may be followed by "ERR=", this indicates that the change in friction between iterations is excessive (15% on load, by default)



INTERGRAPH

The non-convergence window – Stop

- Use Continue to move on to the next iteration
  - Identify a pattern (e.g., Nonconverged = 5,6,3,5,6,3,5,6,3,…)
  - Identify the iteration with the smallest un-converged set (in this example: 3) that has a simple OPEN/CLOSE change
- Click Cancel to abort the analysis
  - No results will be displayed
  - (That is why we are talking about this – you want results.)
- Path forward:
  - You can remove the non-converging Load Case from analysis and review other results
  - Also, by reducing some nonlinearity in the input processor, this load case may produce results for review
- Note: Clicking F2 again will toggle off the single step.



INTERGRAPH

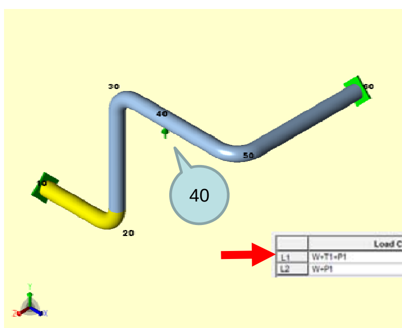
My approach

- Get results to evaluate the trouble
- Focus only on the Load Case that is presently not converging
- Find a simple Open/Close situation and make it linear – NOTE THE CHANGE
- Rerun
- Remove more of these simple Open/Close situations if necessary to get convergence
- Check the "error" in the results for this load case
- If still no solution and only friction remains, select a friction support and either 1) lock down the node or 2) remove mu at that node (again, simplify, make it more linear – all you want is a look into the results to decide)
- Note that if you have to remove several nonlinearities, you may be able to reinsert some of them when you finally pick the right one
- So, linearize and review the impact

INTERGRAPH

LET'S LOOK AT SOME EXAMPLES

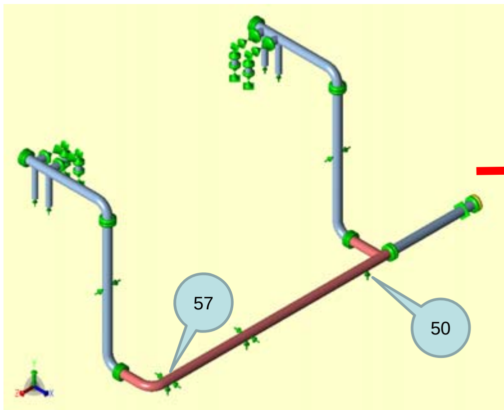
SmallDTwith+Y



	Load Cases	Stress Type
L1	W+T1+P1	OPE
L2	W+P1	SUS

INTERGRAPH

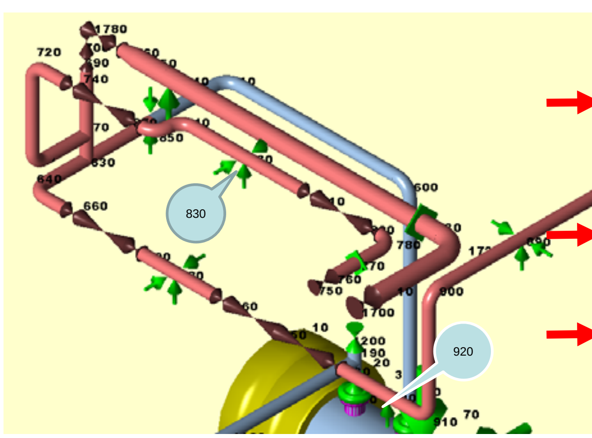
Multibranch



	Load Cases	Stress Type
L26	W+D1+T1+P1-U1-U2+U3+WIN1	OPE
L27	W+D1+T1+P1-U1-U2-U3+WIN1	OPE
L28	W+D1+T1+P1+U1+U2+U3+WIN2	OPE
L29	W+D1+T1+P1+U1+U2-U3+WIN2	OPE
L30	W+D1+T1+P1+U1-U2+U3+WIN2	OPE
L31	W+D1+T1+P1+U1-U2-U3+WIN2	OPE
L32	W+D1+T1+P1-U1+U2-U3+WIN2	OPE
L33	W+D1+T1+P1-U1+U2+U3+WIN2	OPE
L34	W+D1+T1+P1-U1-U2+U3+WIN2	OPE
L35	W+D1+T1+P1-U1-U2-U3+WIN2	OPE
L36	W+T1+P1+U1+U2+U3+WIN3	OPE
L37	W+T1+P1+U1+U2-U3+WIN3	OPE
L38	W+T1+P1+U1-U2+U3+WIN3	OPE
L39	W+T1+P1+U1-U2-U3+WIN3	OPE
L40	W+T1+P1-U1+U2-U3+WIN3	OPE
L41	W+T1+P1-U1+U2+U3+WIN3	OPE
L42	W+T1+P1-U1-U2+U3+WIN3	OPE
L43	W+T1+P1-U1-U2-U3+WIN3	OPE

INTERGRAPH

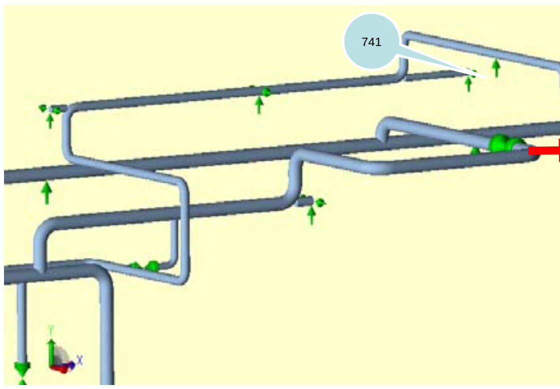
Load Case Blowout



	Load Cases	Stress Type
L99	W+P2-0.127U1-0.111U2+0.299U3+F1-WIN1	SUS
L100	W+P2 0.127U1+0.111U2 0.299U3+F1 WIN1	SUS
L101	W+P2-0.127U1-0.111U2-0.299U3+F1-WIN1	SUS
L102	W+T3+P2+0.273U1+0.385U3+F1+WN2	OPE
L103	W+T3+P2+0.273U1-0.385U3+F1+WN2	OPE
L104	W+T3+P2-0.273U1+0.385U3+F1+WN2	OPE
L105	W+T3+P2-0.273U1-0.385U3+F1+WN2	OPE
L106	W+T3+P2+0.273U1+0.385U3+F1-WN2	OPE
L120	W+T3+P2+0.191U1-0.156U2+0.385U3+F1+WN2	OPE
L121	W+T3+P2-0.191U1+0.156U2+0.385U3+F1+WN2	OPE
L122	W+T3+P2+0.191U1 0.156U2 0.385U3+F1+WN2	OPE
L285	W+P2 0.45U1 0.45U2 0.89U3+F1	OCC
L286	WNC+T5	OPE
L287	T5	EXP

INTERGRAPH

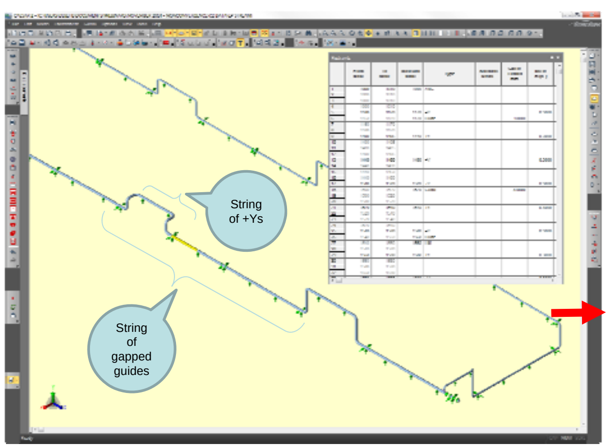
Dummy Leg



	Load Cases	Stress Type
L1	WW+HP	HYD
L2	W+T1+P1	OPE
L3	W+T2+P1	OPE
L4	W+T3+P1	OPE
L5	W+T4+P1	OPE
L6	W+P1	SUS
L7	W+T1+P1+U1	OPE
L8	W+T1+P1-U1	OPE
L9	W+T1+P1+U2	OPE
L10	W+T1+P1-U2	OPE
L11	W+T1+P1+WIN1	OPE
L12	W+T1+P1+WIN2	OPE
L13	W+T1+P1+WIN3	OPE
L14	W+T1+P1+WIN4	OPE

INTERGRAPH

LP Stream



	Load Cases	Stress Type
L1	WW+HP	HYD
L2	W+T1+P1	OPE
L3	W+P1	SUS

INTERGRAPH

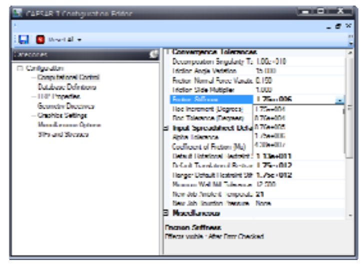
CHANGING FRICTION PARAMETERS

Friction parameters under your control

- Coefficient of friction (model input)
  - (Default) input value
  - Global adjustment as a Load Case Option
- Friction stiffness (Configuration file)
  - Sets the stiffness for restraints CAESAR II adds to the model to make the pipe stick (rather than slip).
  - Current default value is much less than "rigid"
  - Many years ago, we used 50,000 lbf/in (8.76E4 N/cm)
    - To speed up convergence (on much slower PCs)
    - These sliding supports (that were supposedly sticking) interacted with other changing nonlinear conditions to cause odd and unacceptable results
- Convergence tolerance (Configuration file and analysis interaction)
  - Change in normal load magnitude
  - Change in sliding direction

SmallDTwith+Y

- This is not a friction issue that can be addressed by:
  - Changing mu
    - Y load is too sensitive to any horizontal load
    - Operating mu = 0 in Load Case Options
  - Tolerance
    - Friction is never tested against a tolerance – it's a liftoff issue
  - Friction stiffness
    - Works here but with little effect
    - Using default of 1.75E6 N/cm causes liftoff
    - Using 8.76E4 N/cm is very similar to no friction

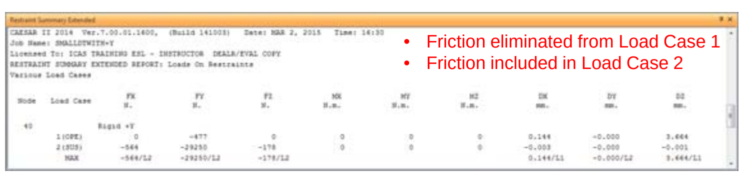


1.75E6 N/cm=1E6 lbf/in
8.76E4 N/cm=50,000 lbf/in

INTERGRAPH

SmallDTwith+Y

- No friction in Operating Case:



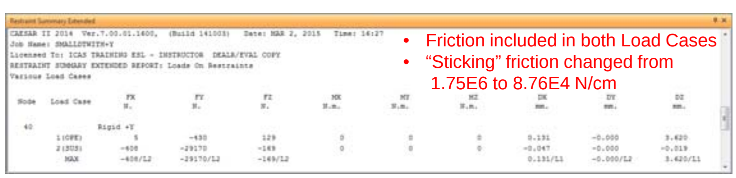
Restraint Summary Extended

CAESAR II 2014 Ver.7.00.01.1600, (Build 141003) Date: MAR 2, 2015 Time: 16:30
Job Name: SMALLDTWITH+Y
Licensed To: ICAS TRAINING ESL - INSTRUCTOR DEALR/EVAL COPY
RESTRAINT SUMMARY EXTENDED REPORT: Loads On Restraints
Various Load Cases

Node	Load Case	FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.	DX mm.	DY mm.	DZ mm.
40	Rigid +Y									
	1(OPE)	0	-477	0	0	0	0	0.144	-0.000	3.664
	2(SUS)	-564	-29250	-178	0	0	0	-0.003	-0.000	-0.001
	MAX	-564/L2	-29250/L2	-178/L2				0.144/L1	-0.000/L2	3.664/L1

- Friction eliminated from Load Case 1
- Friction included in Load Case 2

- Friction stiffness = 8.76E4 N/cm:



Restraint Summary Extended

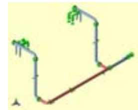
CAESAR II 2014 Ver.7.00.01.1600, (Build 141003) Date: MAR 2, 2015 Time: 16:27
Job Name: SMALLDTWITH+Y
Licensed To: ICAS TRAINING ESL - INSTRUCTOR DEALR/EVAL COPY
RESTRAINT SUMMARY EXTENDED REPORT: Loads On Restraints
Various Load Cases

Node	Load Case	FX N.	FY N.	FZ N.	MX N.m.	MY N.m.	MZ N.m.	DX mm.	DY mm.	DZ mm.
40	Rigid +Y									
	1(OPE)	5	-430	129	0	0	0	0.131	-0.000	3.620
	2(SUS)	-408	-29170	-169	0	0	0	-0.047	-0.000	-0.019
	MAX	-408/L2	-29170/L2	-169/L2				0.131/L1	-0.000/L2	3.620/L1

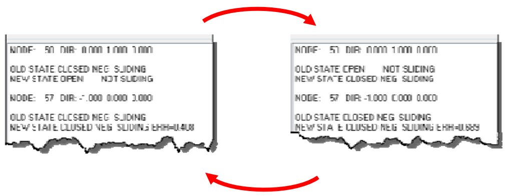
- Friction included in both Load Cases
- "Sticking" friction changed from 1.75E6 to 8.76E4 N/cm

INTERGRAPH

Multibranch



- Here, friction error cannot be resolved
  - CAESAR II requires that the vector representing friction does not vary more than 15% in magnitude and does not change orientation by more than 15 degrees between iterations (default)
- In the earlier look at this model, the +Y restraint at 50 was made linear. This time we will focus on the friction tolerance

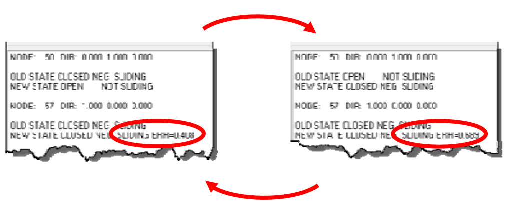


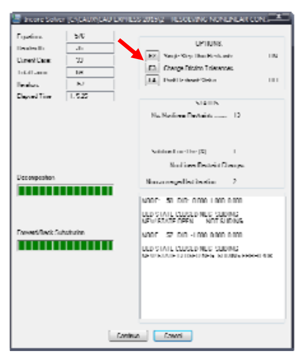
© Intergraph 2015

INTERGRAPH

Multibranch

- What tolerance is required to accept the friction vector applied at the guide at node 57?
- Note the minimum sliding error at 57 is 0.408
- Press (or click on) F3 to adjust the tolerance during the analysis

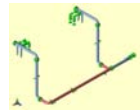




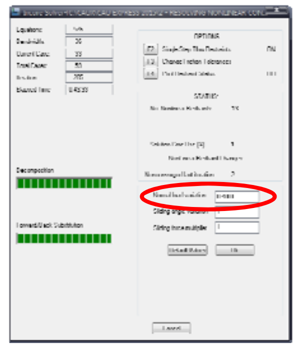
© Intergraph 2015

INTERGRAPH

Multibranch



- Reset the current tolerance
  - A new Normal Load Variation of even 0.4081 will accept this friction error
  - Sliding Angle Variation plays no role at the +Y & Guide support (a value of 1 works just like default 15)
- But the model does not converge due to the instability of the +Y restraint at node 50
- Returning the tolerance to 0.15 will return to the previous condition
- Note that the program will return to the default tolerance settings for the next load case



INTERGRAPH

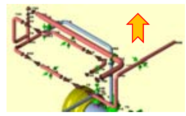
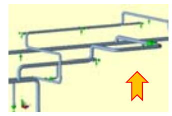
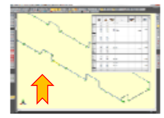
Multibranch

- But consider what we have done here
  - We are adjusting tolerance on Load Case 33
  - The previous load cases used default tolerance settings
  - Additional load cases may have a different tolerance as well
- Interactive changes to friction tolerance may produce results but…
- Tolerances may change between load cases and…
- These results may not be easily reproduced in the next analysis

INTERGRAPH

Remaining models introduced earlier

- Models
  - Load Case Blowout
  - Dummy Leg
  - LP Stream

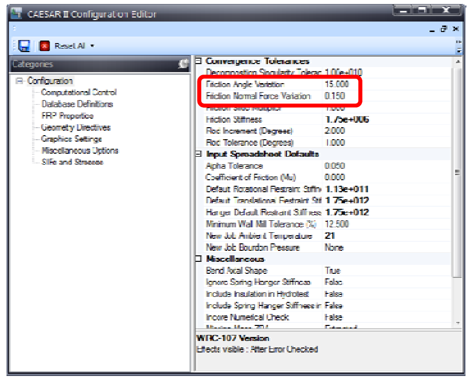




- Adjusting friction tolerance will "overlook" friction errors
- So, opening tolerance on friction will eliminate convergence issues with friction
- But these models still do not converge because of the underlying active/inactive restraint issues
- Note, too, that when these other unstable (active/inactive) restraints are addressed by model simplification, the friction troubles disappear.

CHANGING FRICTION TOLERANCE: PRESET VERSUS INTERACTIVE

Convergence tolerance in the Configuration file

- Friction angle variation
  - If the intended movement is within 15 degrees of the existing friction load vector, the condition is "converged"
- Friction normal force variation
  - If the normal load does not change more than 15% between the current and previous iteration, the condition is "converged"

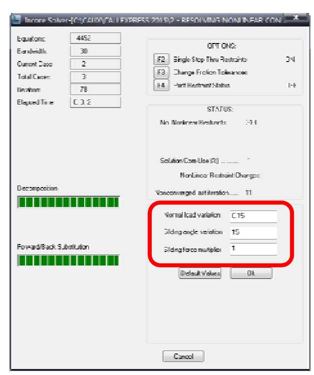


Preset

INTERGRAPH

The non-convergence window – Interactive Control

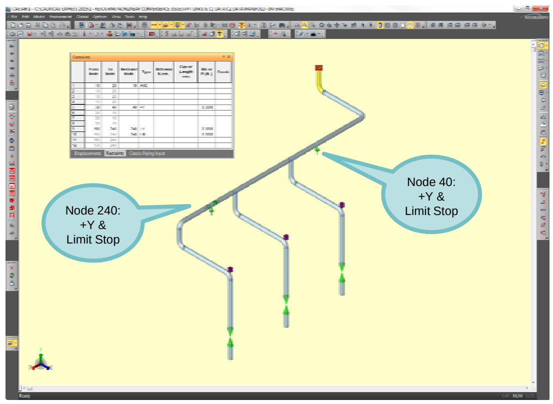
- You can Change the current Friction Tolerances by pressing F3.
  - Adjust acceptable Normal Load Variation (default = 15%) and Friction Angle (Default = 15 degrees).
  - This may be useful in ending the current iteration
  - I discourage using this for a final analysis. Results may change based on what iteration you instigate the change. (Change the program configuration instead.)



Interactive

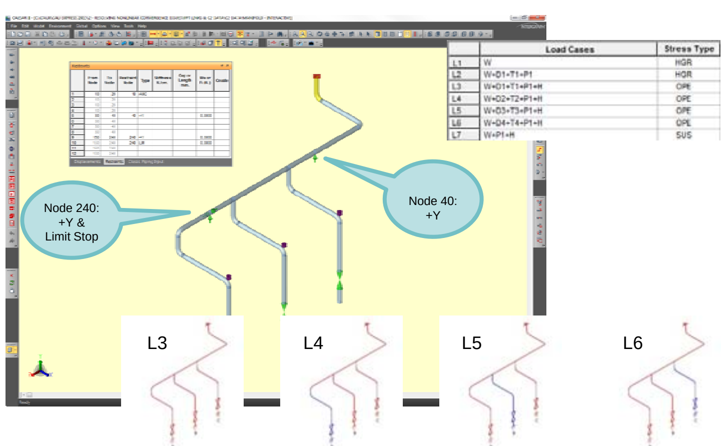
INTERGRAPH

One more model with friction - Manifold



INTERGRAPH

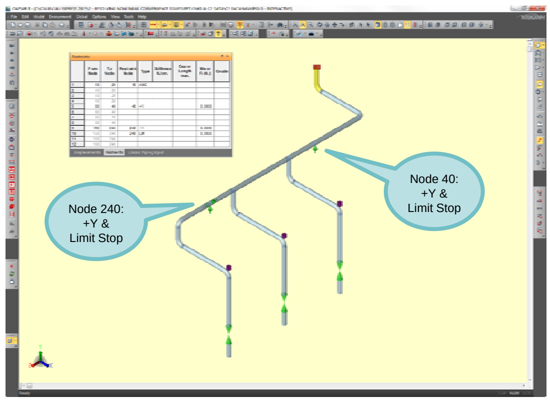
Four operating states: All hot and then each pump spared



	Load Cases	Stress Type
L1	W	HGR
L2	W+D1+T1+P1	HGR
L3	W+D1+T1+P1+H	OPE
L4	W+D2+T2+P1+H	OPE
L5	W+D3+T3+P1+H	OPE
L6	W+D4+T4+P1+H	OPE
L7	W+P1+H	SUS

INTERGRAPH

Comparing results from two approaches: Interactive & Preset



Excel

INTERGRAPH

Changing friction tolerances

- Changing tolerances during analysis:
  - May be helpful in getting results to review
  - Results are sensitive to when you make the change (which iteration?)
  - Tolerances return to default (configuration) values with the start of each new load case
- Changing default tolerances:
  - Provide a consistent solution (see second bullet above)
- Changing friction tolerances does not resolve those Active/Inactive support issues described earlier
- Individually resolving non-converging supports (Active/Inactive supports) should be settled before addressing friction tolerances

INTERGRAPH

IN CONCLUSION

When CAESAR II does not converge…

- Don't panic
- Make sure your load cases are sensible for nonlinear evaluation
  - Include deadweight
  - Do not evaluate occasional loads alone
- Reduce the nonlinearity to get results
- Evaluate the (possible) error introduced by the added linearity

Conclusions

- Manipulate the model to produce output
- Evaluate the impact of that change
  - In many cases, there is no adverse effect on the accuracy of the model
- Be transparent in your change
  - Single changes are easily controlled
  - Suggested order
    - Single direction to double acting
    - Gapped restraints to no gap or totally free (closed gaps require a displacement set)
    - Friction
      - At a single restraint
      - For a Load Case
      - Tolerance as a Configuration setting
      - Tolerance in the analysis processor
- Document your model adjustment

INTERGRAPH

FAILURE TO LAUNCH

Questions / Comments?


FAILURE TO LAUNCH
Thank You