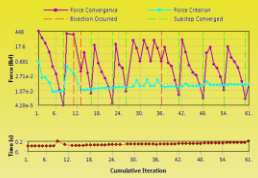
Twin Cities
ANSYS® User Meeting
November 2010

Nonlinear Convergence

Content by Epsilon FEA, LLC

Nonlinear Convergence



Types of Nonlinearities

1. High Deformation (NLGEOM)
2. Contact
3. Material

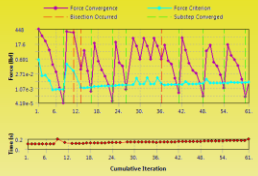
Types of Non-Convergence

1. Element shape distortion
2. Contact Status
3. Residuals Exceeds Allowable

How to Fix it

1. Pinpoint the source of instability
2. Twelve Case Studies
3. Leverage Experience

Types of Nonlinearities

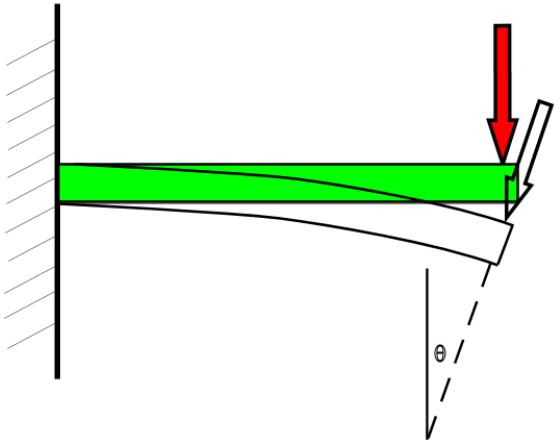


1. High Deformation (NLGEOM)
2. Contact
3. Material

- Stiffness Matrix is built using material moduli and ***geometry***

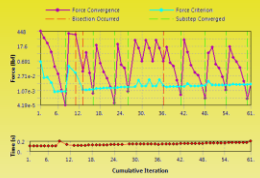
$$F = KX$$

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}$$



- As the geometry (coordinates) changes, so does the K Matrix
- For each iteration, a new stiffness matrix is made based on the previous iteration's deflections.

Types of Nonlinearities

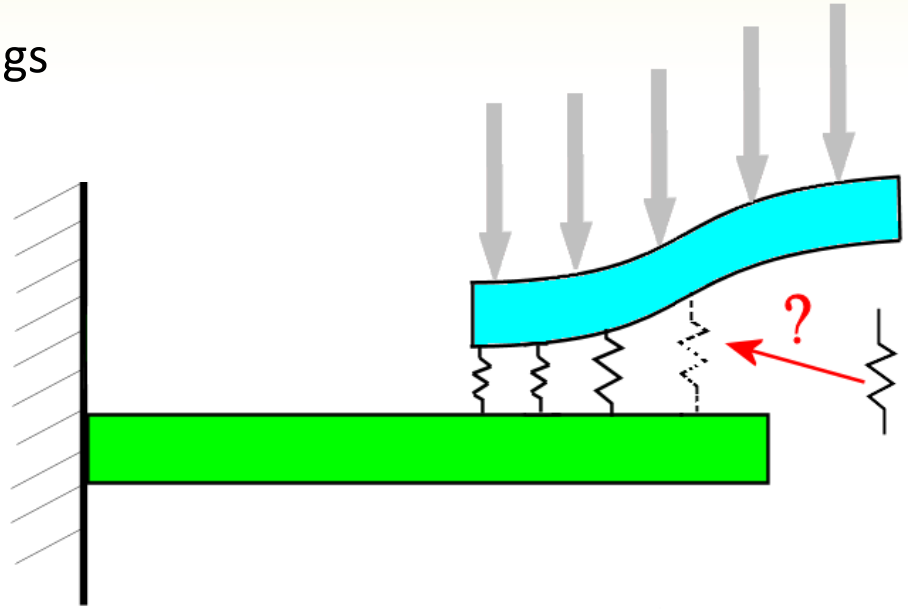


1. High Deformation (NLGEOM) Contact
2. Contact
3. Material

- Contact is enforced with springs

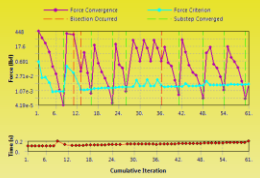
$$F = KX$$

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}$$



- After each iteration, a new guess is made for activation/location of springs and the corresponding stiffness (penalty method)

Types of Nonlinearities

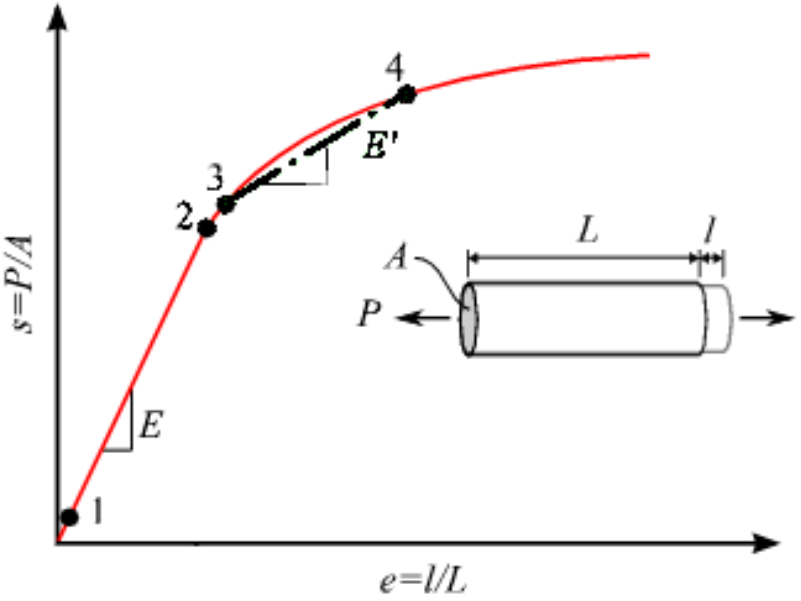


1. High Deformation (NLGEOM)
2. Contact
3. Material

- Stiffness Matrix is built using material ***moduli*** and geometry

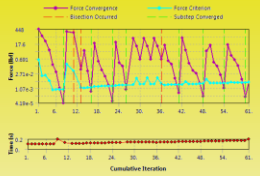
$$F = KX$$

$$\begin{bmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{bmatrix} = \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} & k_{15} & k_{16} \\ k_{21} & k_{22} & k_{23} & k_{24} & k_{25} & k_{26} \\ k_{31} & k_{32} & k_{33} & k_{34} & k_{35} & k_{36} \\ k_{41} & k_{42} & k_{43} & k_{44} & k_{45} & k_{46} \\ k_{51} & k_{52} & k_{53} & k_{54} & k_{55} & k_{56} \\ k_{61} & k_{62} & k_{63} & k_{64} & k_{65} & k_{66} \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}$$



- After each iteration, a new material stiffness is computed

Convergence Failures

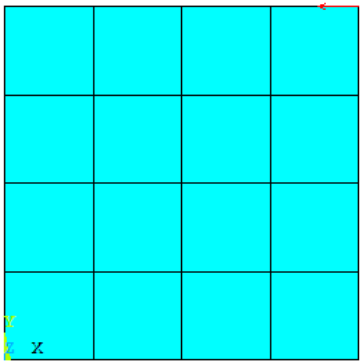


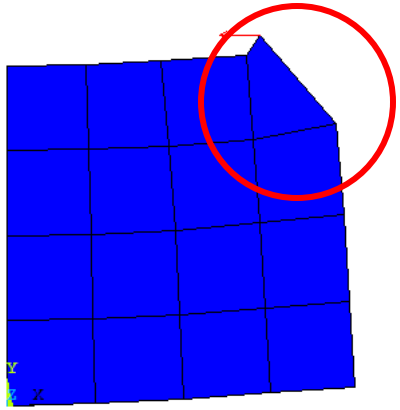
1. Element Distortion
2. Contact
3. Material

- With high deformation (NLGEOM), node locations are updated with deflections
- Element shapes changes accordingly
- New element shape no longer is acceptable

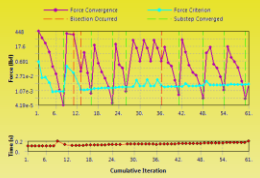
R E S T A R T I N F O R M A T I O N

REASON FOR TERMINATION. . . . . .ERROR IN ELEMENT FORMULATION



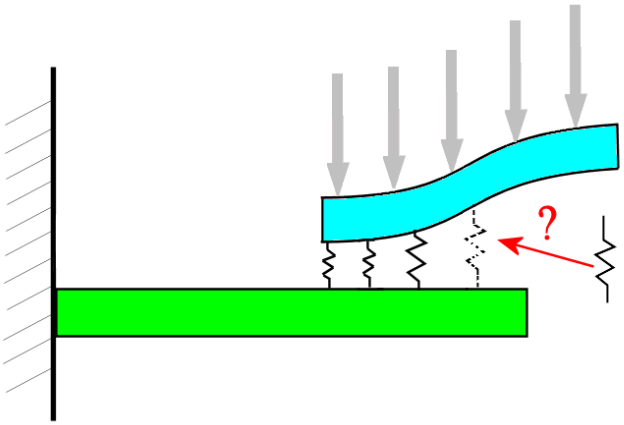


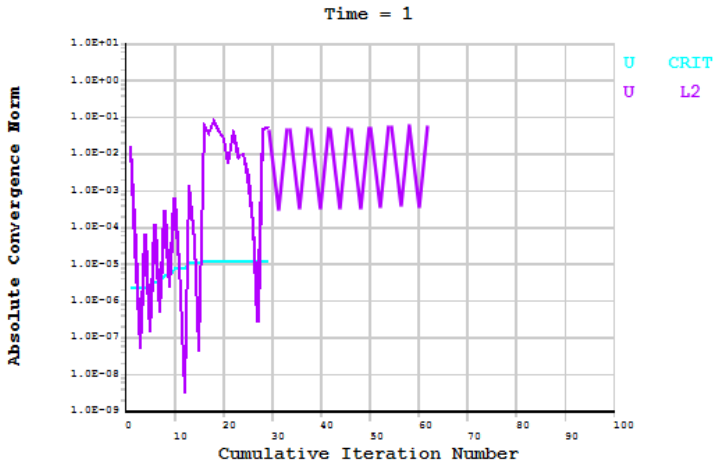
Convergence Failures



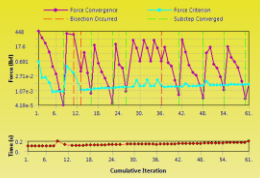
1. High Deformation (NLGEOM) Contact
2. Contact
3. Material

- Contact status changes endlessly (aka ‘chattering’)
  - Gets in activation/deactivation loop on set of springs
    - Closed > Open > Closed > Open > …
    - Often just 2 – 20 elements are the offenders
  - Internal contact algorithm doesn’t recognize the repetition
    - ANSYS can just keep grinding away at it





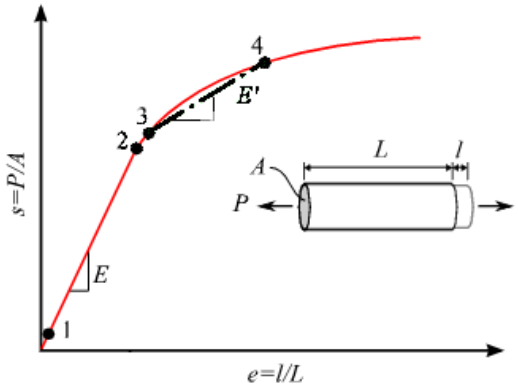
Convergence Failures

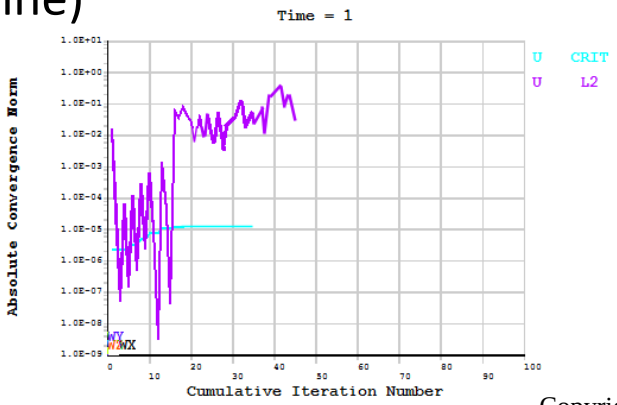


1. High Deformation (NLGEOM)
2. Contact
3. Residuals Exceed allowable

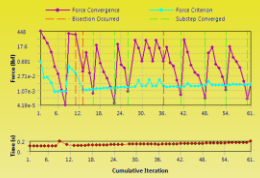
$$F = KX$$

- Estimate is made for materials and geometry giving $[K_1]$
  - Assumes some contact status
  - Assumes some point on the material curve
- A resulting $[K_1]^{-1}[F]$ gives a resulting $[X_1]$
- A resulting $[K_1][X_1]$ gives a resulting $[F_1]$
- Error Norm compares $[F_1]$ to boundary condition $[F]$
  - Norm results in a single scalar (purple line)
  - Norm compared to criteria (blue line)



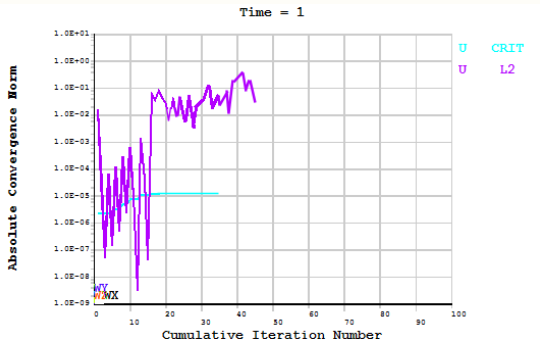


Convergence Failures

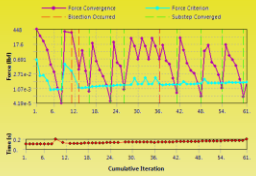


1. High Deformation (NLGEOM)
2. Contact
3. Residuals Exceed allowable, cont'd
   - Norm compare $[F_1]$ to boundary condition $[F]$
   - New estimate is made for $[K_2]$
     - Updated contact
     - Updated Geometry
     - Updated moduli
       - Default is "Newton Raphson" method
       - Interpolates/extrapolates linearly based on previous iteration
   - Cycle repeats to $[K_n]$ until the error norm is less than criteria
   - **Convergence depends on the quality of each new estimate $[K_n]$!!**

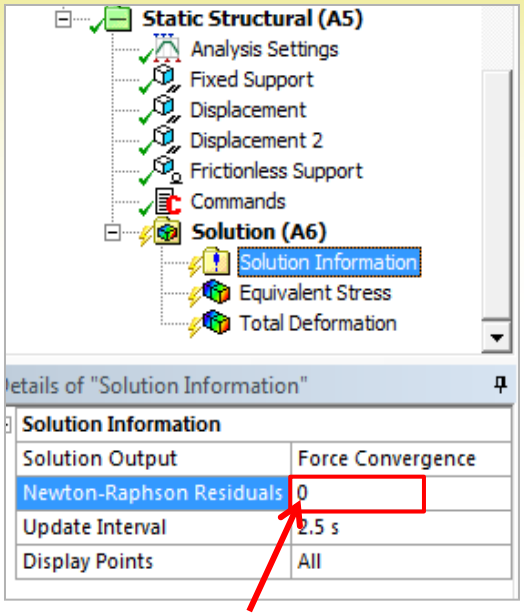
$$F = KX$$



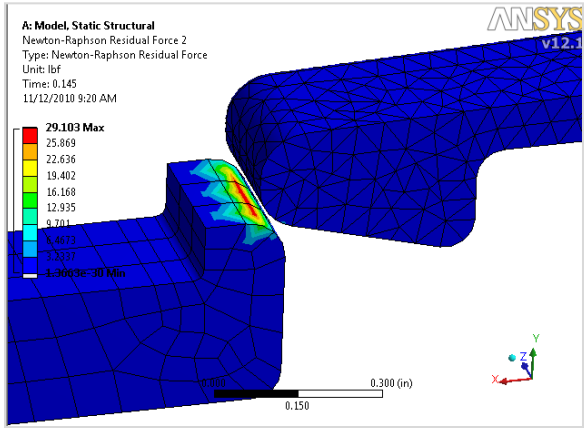
How to Fix it



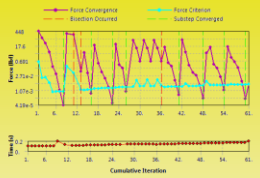
- Pinpoint Location of instability
  - **Debug on coarse/simplified model**
  - Interpret output from solver
  - View the unconverged solution
    - In classic, read in set 999999
  - Leverage diagnostics to find offending elements
    - Show distorted elements
    - Store Newton Raphson Residuals
      - Has to be set *before* solving
      - `NLDIAG,NRRES,ON`
      - A lone hotspot will usually dominate the norm.



Set to non-zero #

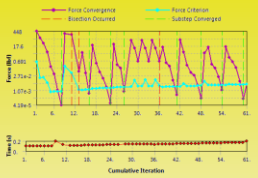


How to Fix it



- 12 Case Studies
  1. DOF Exceeded at 1st Substep
  2. Identical Bisections
  3. Small Time Step Fails
  4. Changing Contact Status
  5. Contact Penetration
  6. Large Plastic Strain Increments
  7. Element Distortion
  8. DOF exceeded at nth Substep (contact)
  9. DOF exceeded at nth Substep (material)
  10. Residuals too High (contact)
  11. Residuals too High (material)
  12. Arc-Length Method (material)

Case Study 1



DOF Exceeded at 1st Substep

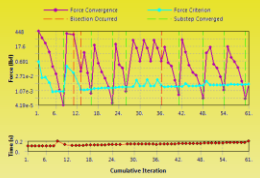
- Not really a “convergence failure”
- Numerical Instability
  - Caused by singular matrix
  - Unconstrained problem
- Problem setup is flawed – you forgot something!
  - Check BC’s, Loads, Materials

```
*** WARNING ***                          CP =      25.366   TIME= 15:26:06
There are 6 small equation solver pivot terms.

*** ERROR ***                            CP =      25.381   TIME= 15:26:06
The value of UX at node 29 is 854093480.  It is greater than the
current limit of 1000000.  This generally indicates rigid body motion
as a result of an unconstrained model.  Verify that your model is
properly constrained.
```

```
*** ERROR ***                            CP =      24.789   TIME= 15:24:17
A small negative equation solver pivot term has been encountered at the
UZ degree of freedom of node 20.  Check for an insufficiently
constrained model.
```

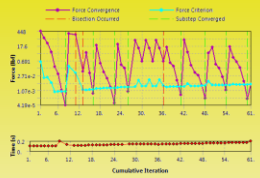
Case Study 1, Cont'd



DOF Exceeded at 1st Substep

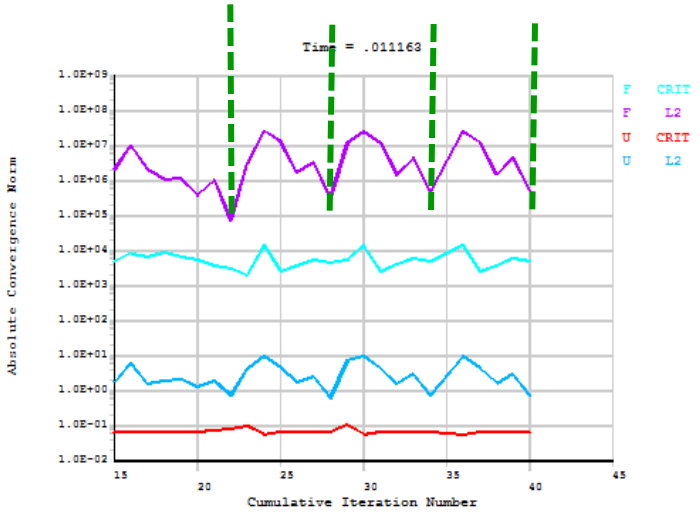
- **Run on simplified model – must find the source**
  - Switch to linear materials
  - Switch to bonded contact
  - Unselect whole components
- If simplification isn't easy, there's a few old tricks:
  1. Add a few random DOF constraints to view a converged substep
  2. Add weak springs
     - Occasionally this all that's needed
     - Be careful! (Kinda bad form, but it's not uncommon)
  3. Run thermal expansion case – view deflections
  4. Run modal extraction – view deflections

Case Study 2

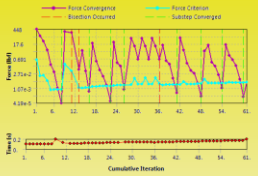


Identical Bisections

- Newton Raphson estimates for $[K_n]$ are not changing!
- Often is caused by ‘stepped’ loading (rather than ‘ramped’)
  - A bisection in time is a bisection in ‘ramped’ loads
  - If load is ‘step’ applied, bisection has no benefit
  - Ramped vs. Stepped controlled by KBC
    - Default is ramped
  - Constraint Equations (CE’s) enforced deflection are stepped!
  - Contact interference defaults to ramped
    - Maybe it’s set to ‘stepped’ via KEYOPTION 9?
  - Might Try Arc-Length Method



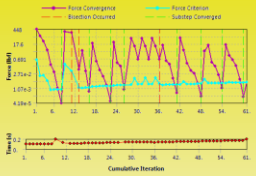
Case Study 3



Small Time Step Fails

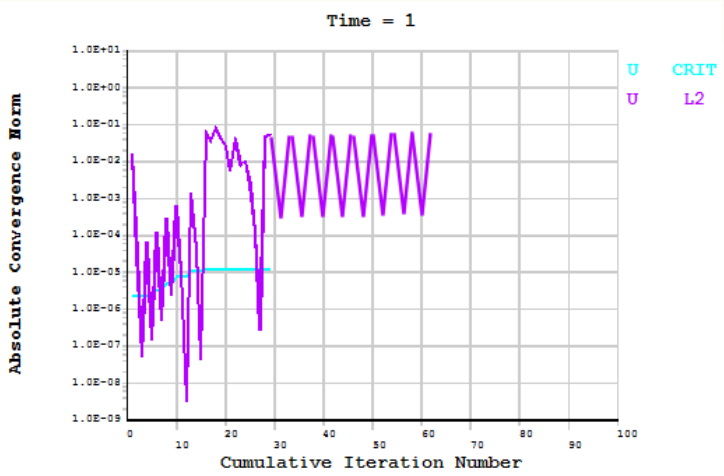
- Could be gross over-application of loads
  - Unit problem – orders of magnitude greater loading
- If it doesn't converge at time = 0.05 – it probably won't
  - If it doesn't converge at time =0.01 – it almost definitely won't
- New estimate is made for **$[K_n]$**
  - Updated contact
  - Updated Geometry
  - Updated moduli
    - Default is "Newton Raphson" method
    - Interpolates/extrapolates linearly based on previous iteration
- Cycle repeats to until the error norm is less than criteria

Case Study 4

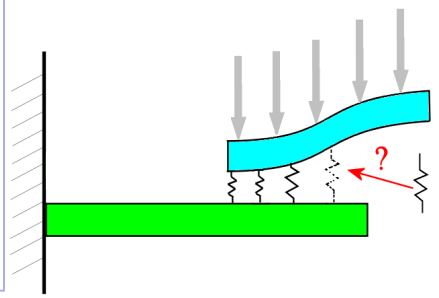


Changing Contact Status

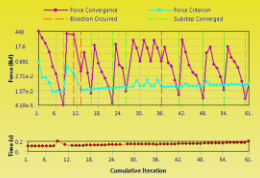
- AKA ‘Chattering’
  - Gets in activation/deactivation loop on set of springs
    - Closed > Open > Closed > Open > …
    - Often just 2 – 20 elements are the offenders
- Rare in recent versions of ANSYS
- Subtle changes in mesh can perturb solution off instability point
- Alter FKN by 5-10X lower.
  - Verify acceptable penetration




```
6 CONTACT POINTS HAVE CHANGE OF CONTACT STATUS
     FORCE CONVERGENCE VALUE   =  10.64       CRITERION= 0.2110E-01
    EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.3872E-02
     LINE SEARCH PARAMETER =  0.5447     SCALED MAX DOF INC = -0.2109E-02
          6 CONTACT POINTS HAVE CHANGE OF CONTACT STATUS
     FORCE CONVERGENCE VALUE   =  7.734       CRITERION= 0.2109E-01
    EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.6622E-03
     LINE SEARCH PARAMETER =   1.000     SCALED MAX DOF INC = -0.6622E-03
     FORCE CONVERGENCE VALUE  = 0.8278       CRITERION= 0.2098E-01
```

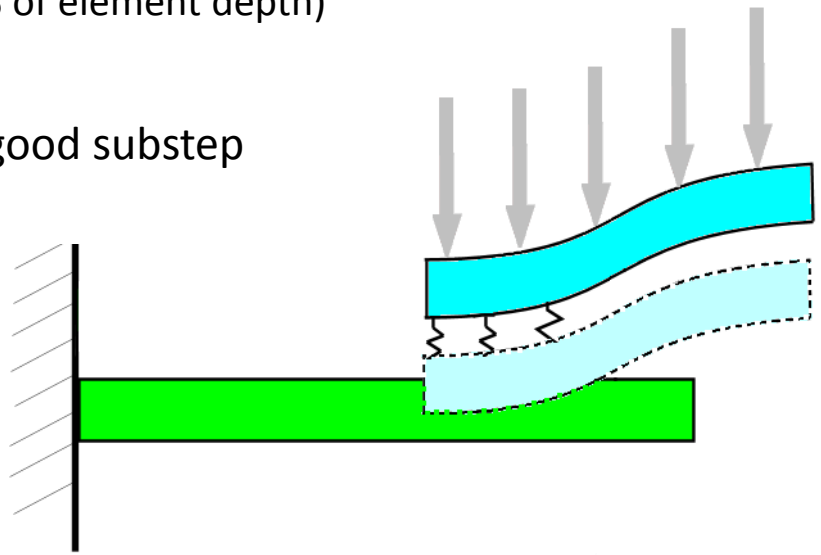


Case Study 5

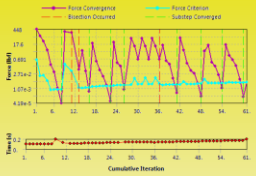


Contact Penetration

- Not really a “convergence failure”
- Rare failure in recent ANSYS versions
- Look for unrealistically constrained over penetration
  - Like a 2” press fit on a 3” dia. Pipe
  - Look for unrealistically soft FKN (like < 0.0001)
    - Default is 1.0
  - Look for manually set FTOLN to small value (like 0.01)
    - Default is 0.2 (meaning roughly 20% of element depth)
- Remember to leverage restarts
  - Make changes and restart from last good substep



Case Study 6

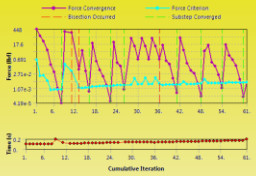


Large Plastic Strain Increments

- Not really a “convergence failure”
- Max plastic strain increment defaults to 15%
  - That’s pretty large for path-dependent problem
- Rare failure
  - Large strain increment usually results in element shape error first
- Look for unrealistic material model (unit problem?)
- Take smaller steps while applying load
- Hard to imagine increasing CUTCONTROL beyond 15%...

```
 *** NOTE ***                                CP =     163.193   TIME= 10:35:56
 The incremental plastic strain computed in this iteration is larger than the
criterion of 15% leading to bisection.  You may try incrementing the load
more slowly by increasing the number of substeps or use the CUTCONTROL
command to re-specify this criterion.
```

Case Study 7



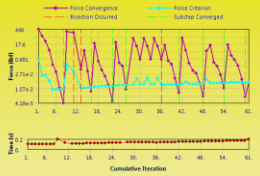
Element Distortion

- Common Failure
- Smaller time steps often fixes this
- Look for unrealistic loads (unit problems)
- Pinpoint offending elements
  - Use NLDIAG to locate offending elements
  - Future ANSYS version will specify offending elements
  - Simplify problem – Unselect components/regions
  - Do element shape checking on previous iteration or unconverged solution
- **Mixed U-P Formulation (Keyoption 6 for SOLID186)**
  - **Magic Bullet!**
  - Longer solutions (25% or so?)
  - Adds pressure DOF to each element to prevent collapsing

```
        R E S T A R T   I N F O R M A T I O N

REASON FOR TERMINATION. . . . . .ERROR IN ELEMENT FORMULATION
```

Case Study 7



Element Distortion, cont'd

- Switch to Explicit Analysis (Autodyne/LS-Dyna)
  - Short duration or stead state only
- Change the original mesh
  - Improve element quality
    - In Workbench switch shape checking to 'aggressive mechanical'
  - Skew original mesh to allow for known directional deformation
- Mesh Morphing
  - Emerging Technology. (Proceed with skepticism in 3D).
  - Terrific Article in PADT's Focus (Eric Miller) – images below

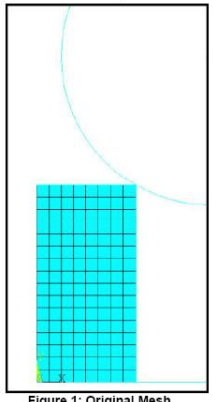


Figure 1: Original Mesh

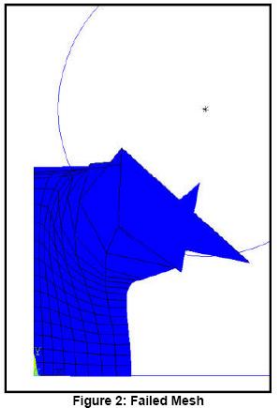


Figure 2: Failed Mesh

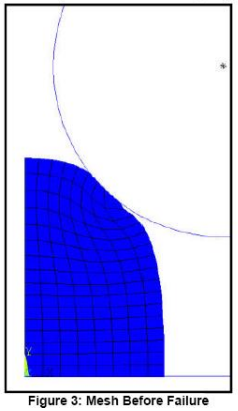


Figure 3: Mesh Before Failure

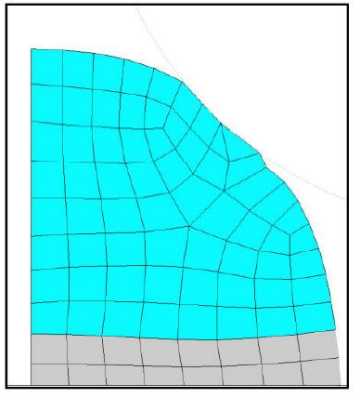


Figure 4: New Mesh

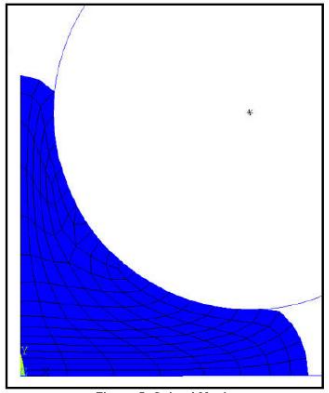
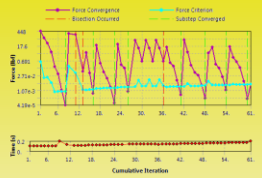


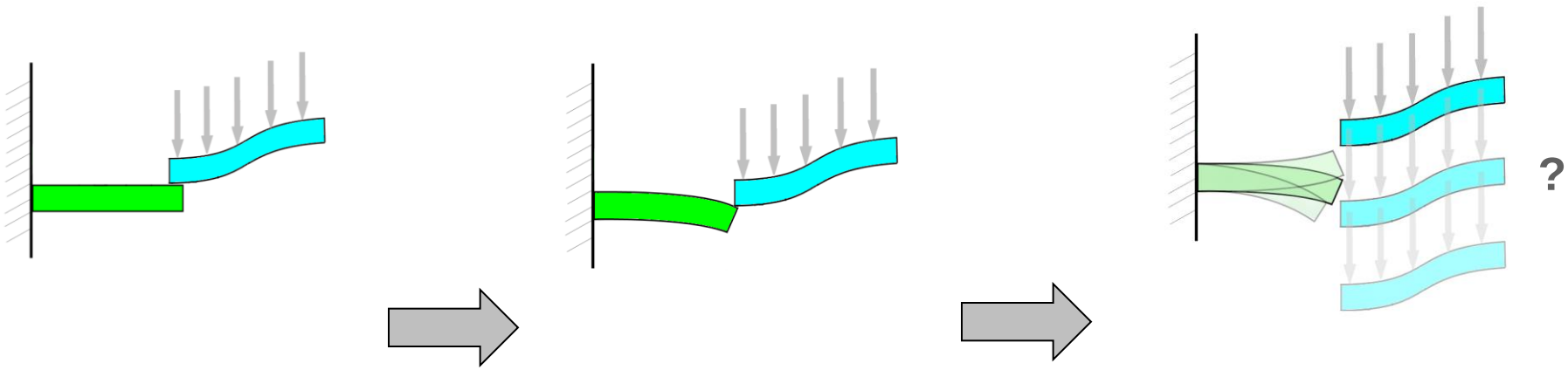
Figure 5: Solved Mesh

Case Study 8

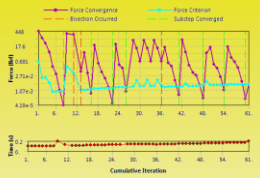


DOF exceeded at nth Substep (contact)

- Look for pass-through of contact surfaces
  - Increase pinball region (little cost to have very large pinball)
  - Use displacement-based BC's rather than Force-based
  - Use `STABILIZE` command to impede large deflections
  - Use weak springs to impede large deflections
- Look for problems in load application / stability
  - See images for example

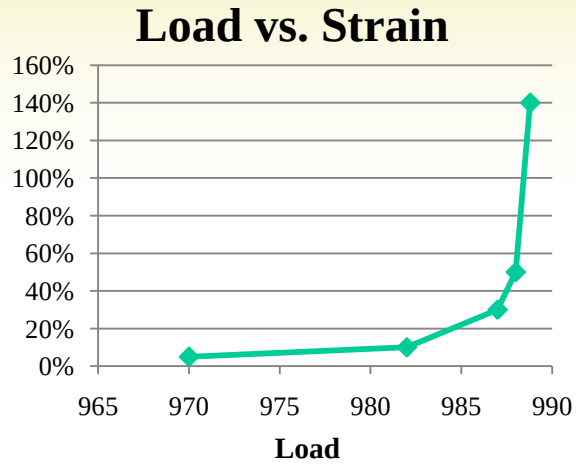


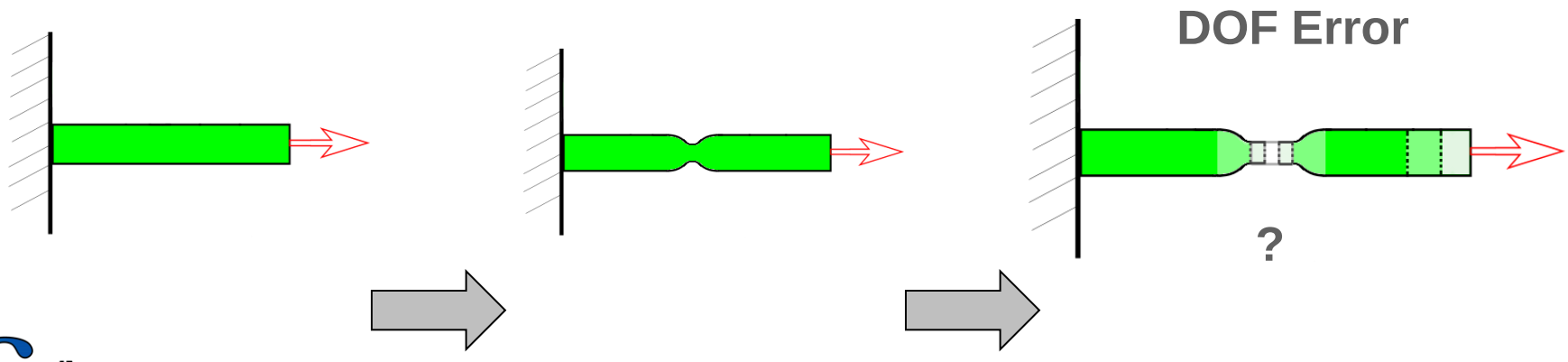
Case Study 9



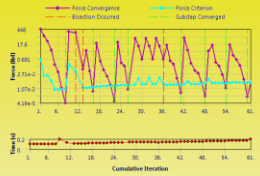
DOF exceeded at nth Substep (material)

- Use displacement-based BC's rather than Force-based
- Analysis goal if running out to plastic failure
  - Represents material failure
  - Modeling actual failure is usually done with explicit (e.g. LS-DYNA)
  - Try Arc-Length Method (See Case 12)
  - Approach failure to sufficient accuracy
    - Example (right): failure is around 990 lbf



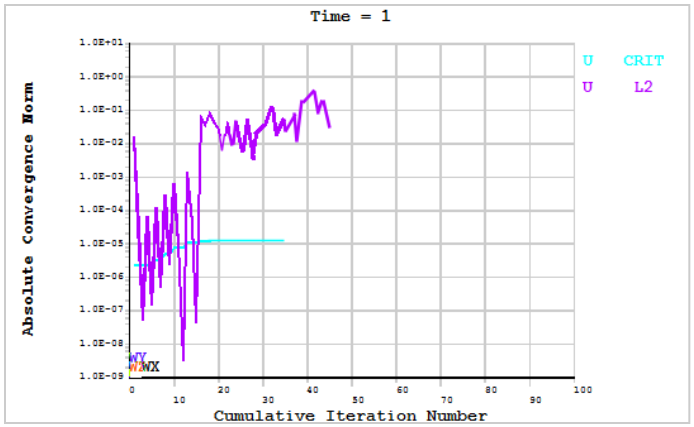


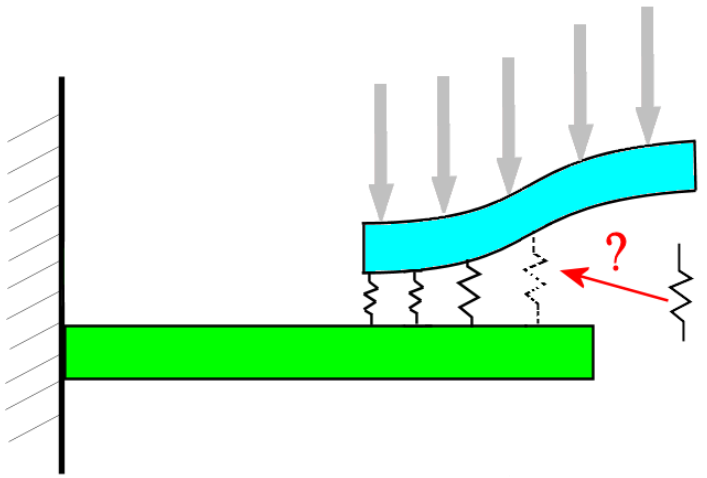
Case Study 10



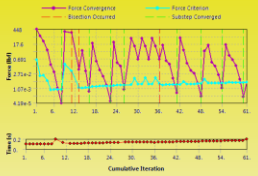
Residuals Too High (contact)

- Very Common!
- Smaller time steps (shouldn't take more than 20)
  - If fails at <5% load, it likely will never make it
- **Soften contact stiffness (FKN) to 0.01**
  - **Magic Bullet!**
  - Default stiffness is quite aggressive (to avoid user error)
  - Check for acceptable contact penetration
    - (with 0.01 stiffness typical penetration is <0.0005" for metal)



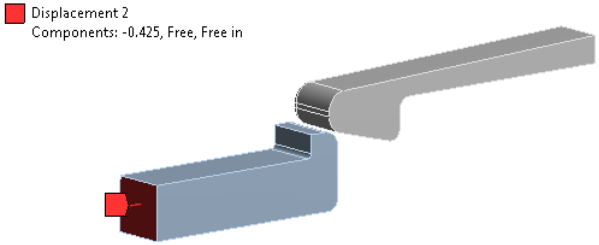


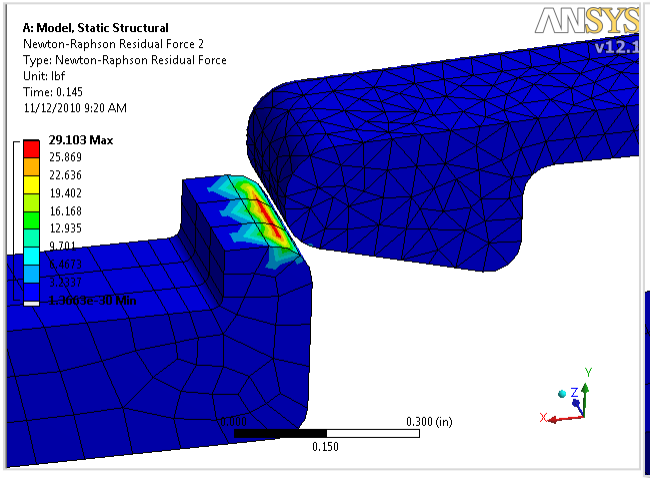
Case Study 10

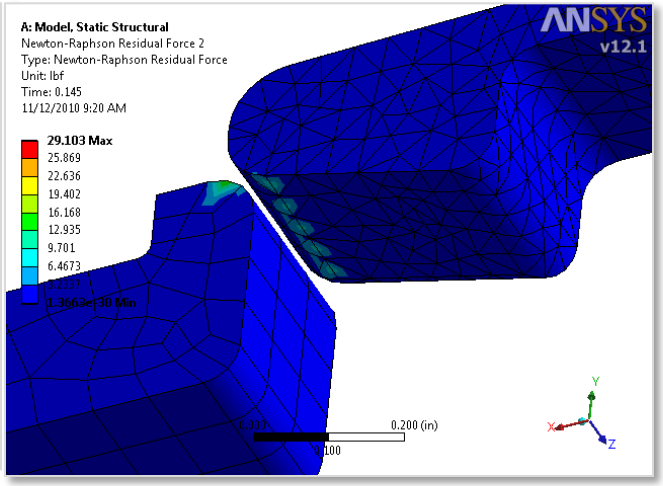


Residuals Too High (contact) cont'd

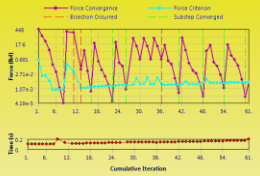
- Pinpoint the offending location
  - Look at last converged solution as well as unconverged (999999)
  - Check status/penetration/pressure
  - View the residuals of the unconverged load step
    - NLDIAD,NRRES,ON







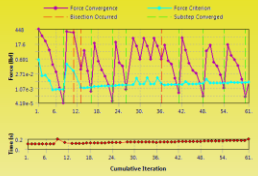
Case Study 10



Residuals Too High (contact) cont'd

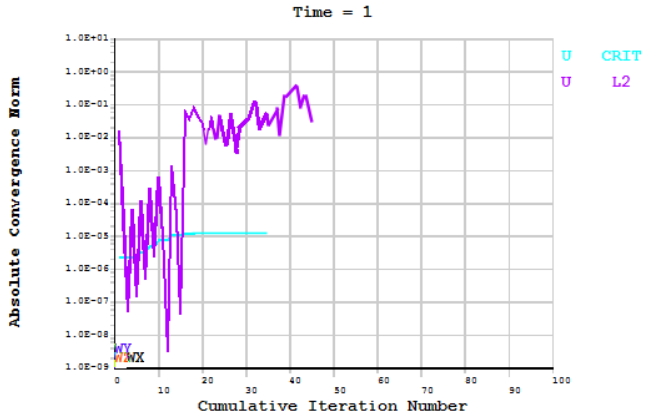
- Examine the location of high residuals – often solution is apparent
  - Look for regions to set to bonded contact
  - Soften contact stiffness (FKN) for just a few elements
    - Make new contact pair for small location
  - Switch to symmetric contact
  - Refine/alter the mesh
  - Switch to penalty method
    - Higher penetration than Augmented Lagrange
  - Switch to contact detection at nodes
  - Switch to spurious contact detection
  - Use U Convergence instead of F(orce).
    - Best to turn on F convergence to stabilize final state
    - Loses some path-dependent accuracy

Case Study 11

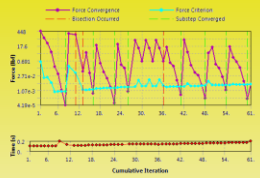


Residuals Too High (material)

- Very Common!
- Smaller time steps (shouldn't take more than 20)
  - If fails at <5% load, then it likely will never make it!
- Pinpoint the offending location
  - View the residuals of the unconverged load step (see Case 10)
- Try alternate material models
  - Different models may have nearly identical results, but converge easier. (such as hardening model, vs. strain energy model).
  - Simplify the material model – should I really bother with a 9-parameter Curve fit?



Case Study 11



Residuals Too High (material) cont'd

- Switch to Linear Material for offending elements if they are not critical
  - Especially singularities
- Try VT Accelerator
  - Requires extra licensing (HPC)
  - See PADT's Focus article (image right)
  - Only effective for some material models
- Be *SURE* you understand the material model
  - Check a given nodal stress/strain
    - Remember to use the 'unaveraged' or 'element' results
  - Build simple block test case
    - Should match hand-calcs
    - Look for unit errors

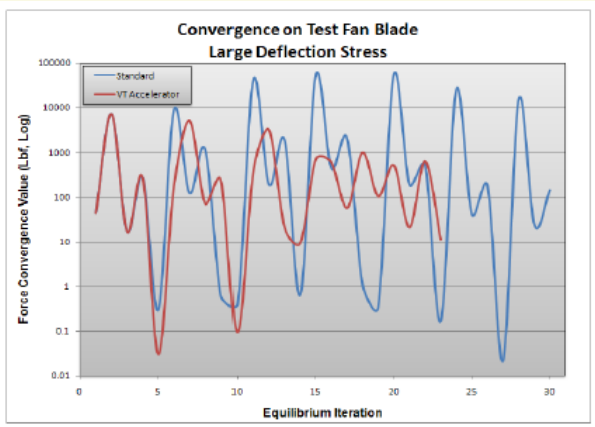
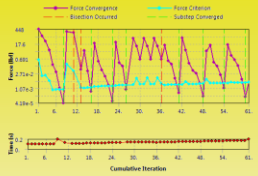


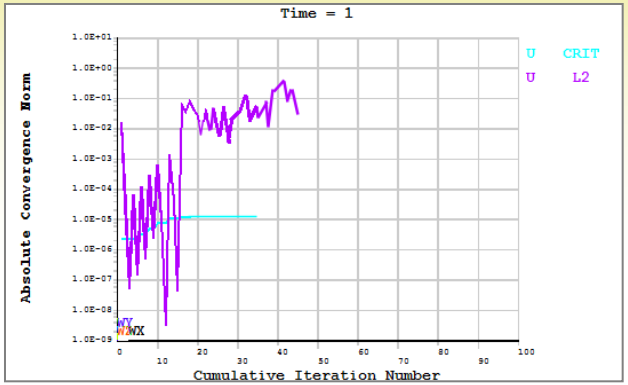
Figure 2: Convergence History

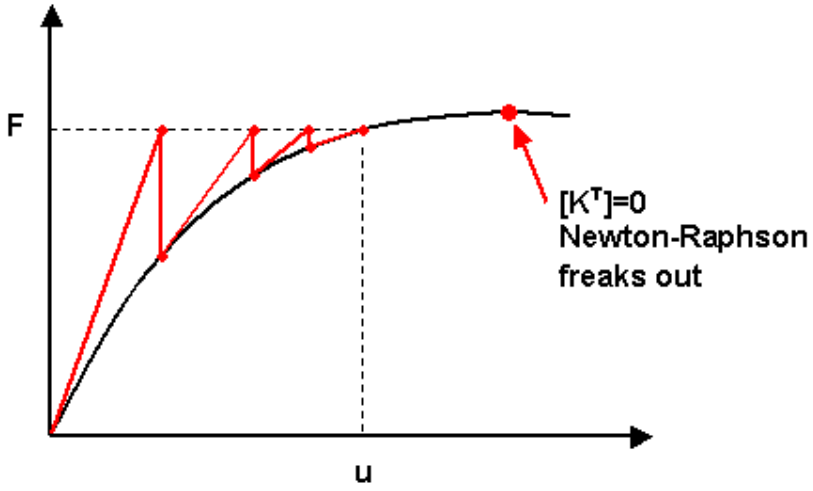
Case Study 12



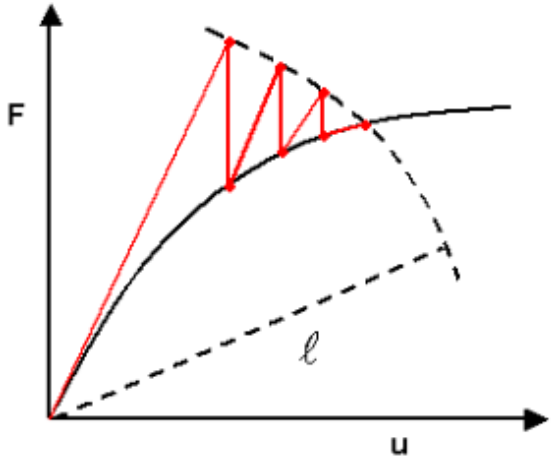
Arc-Length Method (Material)

- A few classes of problems are unsuited for Newton Raphson
- Arc-length method is alternative algorithm to develop new $[K_n]$
- See PADT's Focus article for details



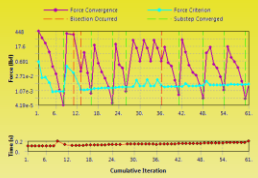


Newton Raphson Method



Arc-Length Method

Leverage Expertise



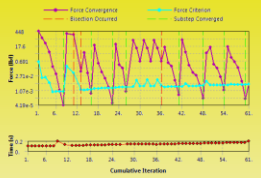
Send us your Problem Models

- One hour of help, can save a week of struggles
  - Convergence can require trial and error
- We transfer the knowledge and models back to you

- **Establish a Purchase Agreement with Epsilon FEA**
  - Setup Epsilon FEA as approved supplier (plus NDA)
  - 5 hour agreement would cover almost any single convergence case ($600).
  - **Do the paperwork today**, then just pick up a phone when problems arise

- **Epsilon FEA is your local ANSYS partner!**

Contact : Rod Scholl, Principal Analyst
Rod.Scholl@EpsilonFEA.com
612.605-6894



Discussion