Obtaining and Optimizing Structural Analysis Convergence

John Higgins, PE
Senior Application Engineer
Obtaining and Optimizing Structural Analysis
Convergence

1. Introduction

2. Example Problems – Characterize Convergence Difficulty

3. Step-by-Step Convergence Procedure

4. How to leverage the robustly converging model

5. Q&A
Obtaining & Optimizing Structural Analysis Convergence

Example Problems
Characterize Convergence Difficulty
Before you start...

→ Which parts and/or model components are to be evaluated?

→ What are the design objectives?

→ What engineering criteria will be used to evaluate the design?
Plan Ahead!

→ What are you trying to find out?
  → Nominal or Peak Stress, displacement, forces etc.

→ How much of the structure needs to be modeled?

→ What are the boundary conditions and loads?

→ What are the critical locations and when are they critical?

→ What is a practical maximum node count and solution time?
Linear vs. Nonlinear definition

Frequency Domain Analysis vs. Time Domain Analysis
All products exhibit nonlinearities

Always start with the simplest linear analysis!
Large deflection affects stiffness
Material properties cannot always be assumed to be linear.
Contact is the most common source of nonlinearity and often the most difficult to solve!

Status changes, friction, pressure

When, where?
What is the pressure?
Disciplines within Structural Mechanics

**Static analysis**
- Stresses & deformations
- Linear model

**Dynamics**
- Vibrations - eigenmodes
- Vibrations - Harmonic
- Vibrations - Probabilistic
- Time histories
- Drop test, shocks
- Mechanisms

**LINEAR**
- Static analysis

**NONLINEAR**
- Nonlinear behavior
Characterize Convergence Difficulty

→ Easier Problems
  → Deformations are relatively small
  → Nonlinear strains (plasticity, creep, swelling) are small
  → Contact status does not oscillate
  → Models are small and simplified (2D, Axisymmetric)
  → Symmetric boundary conditions are utilized
  → Displacement based loads
  → Loads result in tensile member stresses
  → Nonlinear buckling to the point of instability

→ Harder Problems
  → Very large deformation
  → Large strains with large distortion
  → Contact chatter
  → Contact sliding with high friction coefficient
  → Post buckling response
  → Large 3D models with complex geometry
  → No symmetry boundaries
  → Force based loads
Pin Insert Model

Hard Solution
- Model the entire Pin / socket assembly
- Mesh fine enough to capture local stress concentrations
- Use a force based analysis to model pin insertion and removal
- Determine critical locations / load steps

Easier Solution
- Model a single axi-symmetric Pin/Socket assembly
- Create mapped mesh with refinement on contact surfaces and areas of high stress
- Use a displacement controlled solution
- Use auto time stepping and smart output controls since max stress might not occur at the final solution step
Gasket Assembly

Hard Problem

- Large model
- Complex loading sequence
- Multiple bolt loads
- Frictional Contact
- Nonlinear material response
Manifold Assembly

Easier Problem

- Small symmetric model
- Uniform mesh
- Sequential bolt pre-tension
- Frictionless Contact
- Small strain material response
Automatic contact between bodies reduces modeling time

409 Parts
967 Contact Pairs
Building Collapse Simulation

Evaluating the thermal-structural response of the World Trade Center collapse
Clevis Pin Pullout

Total Deformation
Type: Total Deformation
Unit: in
Time: 4
11/1/2009 3:00 PM

- 4.6423 Max
- 4.1265
- 3.6107
- 3.0949
- 2.579
- 2.0632
- 1.5474
- 1.0316
- 0.5158
- 0 Min
ANSYS Nitinol Material Behavior

Nitinol tensile bar
Rubber boot with self-contact

Plot No. 1

Nodal Solution

Step = 1
Sub = 1
Time = .2
SeqV (AVG)
DMX = .408514
SMN = .1325e-03
SMX = .19922
Vena Cava Filter

IVC filters are used in case of contraindication to anticoagulation

I.e. – it captures clots!
FSI Example – Vena Cava Filter
FSI Example – Vena Cava Filter

Streamlines

Deformation
Obtaining & Optimizing Structural Analysis Convergence

Step-by-Step Convergence Procedure
My analysis did not converge. Now what?

➔ First step is to determine the cause:

1. Rigid body motion
2. Force balance not obtained
3. Material Instabilities
4. Element formulation error
5. Combination of items 1-4 above

➔ We will examine each in detail including identifying the problem, determining the cause and providing solutions
Rigid body motion error messages

→ DOF limit exceeded.
→ Negative main diagonal.
→ Pivot error.

*** WARNING ***
CP = 11.703 TIME= 16:15:15
Smallest negative equation solver pivot term encountered at UX DOF of node 98. Check for an insufficiently constrained model.
Examples of rigid body motion cont.

*** WARNING ***
CP = 873.336 TIME = 13:07:32
There are 1 small equation solver pivot terms.

---

*** WARNING ***
CP = 897.090 TIME = 13:08:57
There are 2 small equation solver pivot terms.

---

*** ERROR ***
CP = 897.230 TIME = 13:08:57
The value of UX at node L14305 is 2.393821741E+10. It is greater than the current limit of 10000000. This generally indicates rigid body motion as a result of an unconstrained model. Verify that your model is properly constrained.

---

*** ERROR ***
CP = 897.230 TIME = 13:08:57
*** MESSAGE CONTINUATION ---- DIAGNOSTIC INFORMATION ***
If one or more parts of the model are held together only by contact verify that the contact surfaces are closed. You can check contact status in the SOLUTION module for the converged solutions using CQCHECK.

---

********************************************************************************
SUMMARY FOR CONTACT PAIR IDENTIFIED BY REAL CONSTANT SET 4 Max. Penetration of -2.734581805E-05 has been detected between contact element 7173 and target element 7449.
Causes of rigid body motion

→ Insufficient supports
→ Individual parts of an assembly not supported
→ Insufficiently connected dissimilar element types

<table>
<thead>
<tr>
<th>Element Type</th>
<th>Rigid Body Motion Permitted</th>
<th>Rigid Body Motion Prevented</th>
</tr>
</thead>
<tbody>
<tr>
<td>2-D Plane Stress/Strain Solid (UX,UY)</td>
<td><img src="image" alt="Diagram" /></td>
<td><img src="image" alt="Diagram" /></td>
</tr>
<tr>
<td>2-D Axisymmetric Solid (UX,UY)</td>
<td><img src="image" alt="Diagram" /></td>
<td><img src="image" alt="Diagram" /></td>
</tr>
<tr>
<td>3-D Solid (UX,UY,UZ)</td>
<td><img src="image" alt="Diagram" /></td>
<td><img src="image" alt="Diagram" /></td>
</tr>
</tbody>
</table>
Find the rigid body motion

- Plot the unconverged or last converged displacement solution and verify displacement scaling
Use modal analysis to find rigid body motion

If it is not clear what constraint is required to eliminate rigid body motion, a modal analysis can be performed.

- A modal analysis determines the vibration modes including rigid body motion.
- Each rigid body mode is predicted as a zero frequency mode.
- Animating the zero frequency mode shape illustrates the rigidly moving model.
Examples of Force equilibrium not obtained

- Convergence value is greater than criterion after min. load increment and max. number of iterations are solved

![Graph showing force convergence](image)

![Oscillating convergence](image)
Equilibrium Iterations

• A nonlinear structure is analyzed using an iterative series of linear approximations, with corrections.

• ANSYS uses an iterative process called the *Newton-Raphson Method*. Each iteration is known as an *equilibrium iteration*.

A full Newton-Raphson iterative analysis for one increment of load. (Four iterations are shown.)
Convergence Procedure

The difference between external and internal loads, \( \{F_a\} - \{F_{nr}\} \), is called the *residual*. It is a measure of the force imbalance in the structure.

The goal is to iterate until the residual becomes acceptably small; less than the criterion, where the solution is then considered *converged*.

When convergence is achieved, the solution is in *equilibrium*, within an acceptable tolerance.
Causes of Force balance not obtained

→ Contact Stiffness is too large
→ Load is stepped too rapidly
→ For small load increments, MINREF criterion exceeded
→ Material instability
→ Buckling

\[ \|\{R}\|_2 < (0.5\% \times \|\{F}\|_2) \]
Contact Stiffness is too large

\[ F = KU \]

- Using too great a contact stiffness usually leads to oscillating convergence, and often to outright divergence.

\[ F_{KN} = 1 \quad \text{vs} \quad F_{KN} = 0.1 \]
Load is stepped too rapidly
Examples of Material Instabilities

*** ERROR ***
One or more elements have become highly distorted. Excessive
distortion of elements is usually a symptom indicating the need for
corrective action elsewhere. Try incrementing the load more slowly
(increase the number of substeps or decrease the time step size). You
may need to improve your mesh to obtain elements with better aspect
ratios. Also consider the behavior of materials, contact pairs,
and/or constraint equations. If this message appears in the first
iteration of first substep, be sure to perform element shape checking.

1 U-P ELEMENTS DO NOT SATISFY THE VOLUMETRIC COMPATIBILITY
FORCE CONVERGENCE VALUE = 0.9198 CRITERION= 0.2732E-03
EQUIL ITER 21 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.1203E-03
LINE SEARCH PARAMETER = 1.000 SCALED MAX DOF INC = 0.1203E-03
1 U-P ELEMENTS DO NOT SATISFY THE VOLUMETRIC COMPATIBILITY
FORCE CONVERGENCE VALUE = 0.3655 CRITERION= 0.6662E-04
EQUIL ITER 22 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.2881E-03
LINE SEARCH PARAMETER = 0.6785 SCALED MAX DOF INC = 0.1955E-03
FORCE CONVERGENCE VALUE = 0.1778 CRITERION= 0.3772E-04

*** ERROR ***
Solution not converged at time 1.925E-02 (load step 1 substep 6).
Run terminated.

*** WARNING ***
The unconverged solution (identified as time 1 substep 999999) is
output for analysis debug purposes. Results should not be used for
any other purpose.

The solver engine was unable to converge on a solution for the nonlinear problem as constrained. Please see the Troubleshooting section of the Help System for more information.
Causes of Material Instabilities

→ Too large of plastic or creep strain increment
→ Elements with mixed u-P constraints not satisfied
→ Force balance not obtained

1. Material law cannot handle large load and/or time increment
2. Element distorted shape results in negative volume calculation
Causes of Element Formulation Errors

- Element Shape distortion
- Excessive strain
- Volumetric locking
- Hourglass modes
- Buckling
- Very large force unbalance
Basic Corrective Action

→ Use CHECK command for overall verification including missing elastic properties, unconstrained model, and element shape checks.
→ MCHECK command can help you identify defects in the mesh such as holes or cracks.
→ CNCHECK command evaluates the initial contact status of contact pairs, identifying whether the contacts are initially open or closed.
→ Plot the unconverged or last converged displacement solution and/or response history
→ Determine the unbalanced force location by plotting Newton Raphson Residuals using the NLDIAG tool
Nonlinear Solution Corrective Action

ANSYS WB Mechanical offers a toolbox of options under the analysis settings branch for achieving successful convergence.

- **Step Control** - Load steps and substeps
- **Solver Control** - Solver types
- **Nonlinear Controls** - N-R convergence criteria
- **Output Controls** - control data saved during the load history
Plot Newton Raphson Residuals to determine critical contact pair
Avoid mistakes from wrong assumptions!

Make sure to include large deflections when displacements are significant

Check Displacement Scaling

Large deflections included!

Linear solution
Make sure to use the correct material input!

If test data is supplied using the engineering stress-strain measure, you must convert it to true stress-log strain data before entering it into ANSYS for large-strain applications.
Correct the rigid body motion

→ Add supports and/or use displ. controlled solution
→ Use bonded contact for debugging
→ Adjust the parts to all start in contact
→ Increase pinball region
→ Add friction
→ Use bonded contact to tie dissimilar element types
→ Turn on Line Search for large deflection models
→ Add/adjust weak springs
→ Add nonlinear stabilization
→ Use the Arc-Length solution method (Post-buckling)
→ Add contact stabilization damping
Take advantage of symmetries

Axisymmetry
Rotational
Planar or reflective
Repetitive or translational
Connections that prevent rigid body motion

- Spot Welds
- Constraint equations
- Springs
- Beam connections
- Spot Welds
Review Initial Contact Settings and eliminate gaps

*** NOTE ***

CP = 2.312 TIME = 22:43:38

Symmetric Deformable - deformable contact pair identified by real constant set 9 and contact element type 9 has been set up. The companion pair has real constant set ID 10. Both pairs should have the same behavior. ANSYS will keep the current pair and deactivate its companion pair, resulting in asymmetric contact.

Contact algorithm: Augmented Lagrange method
Contact detection at: Gauss integration point
Contact stiffness factor FKN 0.10000
The resulting contact stiffness 0.34244E+16
Default penetration tolerance factor FTOLN 0.10000
The resulting penetration tolerance 0.10890E-04
Max. initial friction coefficient MU 0.20000
Default tangent stiffness factor FKT 1.0000
Default elastic slip factor SLTOL 0.10000E-05
The resulting elastic slip 0.36301E-05
Update contact stiffness at each iteration
Default Max. friction stress TAUMAX 0.10000E+21
Average contact surface length 0.36301E-03
Average contact pair depth 0.10890E-03
Pinball region factor PINB 1.0000
The resulting pinball region 0.10890E-03
Initial penetration will be ramped during the first load step.

*** NOTE ***

CP = 9.333 TIME = 08:33:24
Max. Initial penetration 8.249982762E-02 was detected between contact element 12811 and target element 18554:
You may move entire target surface by: x = -5.839095973E-02, y = 6.44109704E-10, z = 5.828136391E-02 to reduce initial penetration.

*** WARNING ***

CP = 9.333 TIME = 08:33:24
The initial penetration/gap is relatively large. Using bonded/no separation option may cause an accuracy issue. You may use the PSOLVE command to move the contact nodes towards the target surface.

*** NOTE ***

CP = 9.343 TIME = 08:33:25
Min. Initial gap 1.543585102E-02 was detected between contact element 4260 and target element 4282:
The gap is closed due to initial adjustment.

*** NOTE ***

CP = 2.328 TIME = 22:43:38

Min. Initial gap 1.965754579E-06 was detected between contact element 4260 and target element 4282.
You may move entire target surface by: x = 1.644161332E-07, y = 1.951139673E-06, z = -1.738170837E-07, to bring it in contact.

*** NOTE ***

CP = 2.328 TIME = 22:43:38

Symmetric Deformable - deformable contact pair identified by real
Add Contact Stabilization Damping

- Rigid body motion often can occur in the beginning of a static analysis due to the fact that the initial contact condition is not well established.

- Contact Stabilization introduces a viscous damping traction proportional to but opposite to the relative pseudo velocities between the two surfaces along contact normal and/or tangential directions.

Where:

- \( d_n \) = damping coefficient in normal direction
- \( d_t \) = damping coefficient in tangential direction
- \( \dot{u} \) = pseudo velocity
Contact Stabilization Damping

Example: Consider a fixed pin interfacing with a hole in plate with initial radial clearance and under a force based load

- Stabilization captures localized stress distribution more accurately

Conventional Adjust to Touch

Contact Stabilization Damping
# Adjust Contact Pinball Region

<table>
<thead>
<tr>
<th>Default PINB*</th>
<th>Contact Classification</th>
<th>Contact Surface Behavior</th>
<th>Initial Penetration Behavior KEYOPT(9)</th>
</tr>
</thead>
<tbody>
<tr>
<td>3 x Depth**</td>
<td>Rigid/Flex</td>
<td>Standard</td>
<td>Default (Include Everything)</td>
</tr>
<tr>
<td>4 x Depth</td>
<td>Rigid or Flex</td>
<td>Standard</td>
<td>Include Everything/with Ramped effects</td>
</tr>
<tr>
<td>2 x Depth</td>
<td>Flex/Flex</td>
<td>Standard</td>
<td>Default (Include Everything)</td>
</tr>
<tr>
<td>0.50x Depth</td>
<td>Flex/Flex</td>
<td>Bonded or No Separation</td>
<td>Default (Include Everything)</td>
</tr>
<tr>
<td>0.75x Depth</td>
<td>Rigid/Flex</td>
<td>Bonded or No Separation</td>
<td>Default (Include Everything)</td>
</tr>
<tr>
<td>1.00x Depth</td>
<td>Rigid/Flex</td>
<td>Bonded or No Separation</td>
<td>Include Everything/with Ramped effects</td>
</tr>
</tbody>
</table>

* Values are for NLGEOM,ON and are reduced by 50% for NLGEOM,OFF

**Depth = Underlying element depth (for solid elements)

**Depth = 4 x element thickness (for shells and beams)
Correct the Force unbalance

→ Reduce Contact Stiffness
→ Increase the number of Substeps
  → Change Contact algorithm
  → Change contact stiffness update
→ Ramp on the interference fit
→ Use Higher Order Elements
→ Refine the mesh
→ Increase the MINREF criterion
→ Extend the Stress-Strain curve
→ Adjust the convergence tolerance
→ Change solver – direct vs. iterative
→ Change residual combination method
Ideal load stepping – convergence after 3 to 5 iterations each substep

EQUIL ITER 1 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC = 0.3269E-03
LINE SEARCH PARAMETER = 0.9982  SCALED MAX DOF INC = 0.3263E-03
FORCE CONVERGENCE VALUE = 58.44  CRITERION = 9.017

EQUIL ITER 2 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC = 0.3632E-05
LINE SEARCH PARAMETER = 1.000  SCALED MAX DOF INC = 0.3632E-05
FORCE CONVERGENCE VALUE = 9.204  CRITERION = 9.201

EQUIL ITER 3 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC = 0.9539E-06
LINE SEARCH PARAMETER = 1.000  SCALED MAX DOF INC = 0.9539E-06
FORCE CONVERGENCE VALUE = 5.769  CRITERION = 9.389 <<< CONVERGED

>>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 3

\[ \{ F \} \]
\[ \{ F^{nr} \} \]
\[ \{ \Delta u \} \]

Load

Load Step 1
Load Step 2

Substeps

“Time”
equilibrium iterations
Correct the Force unbalance

→ Reduce Normal Stiffness Factor

→ Increase the number of Substeps

→ General Rule – more nonlinearities – use more substeps
Change contact algorithm

→ Use reduced Penalty Stiffness in improve convergence
→ Use Augmented Lagrange to restrict penetration

Pure Penalty: \[ F_{\text{normal}} = k_{\text{normal}} x_{\text{penetration}} \]

Augmented Lagrange: \[ F_{\text{normal}} = k_{\text{normal}} x_{\text{penetration}} + \lambda \]

→ Validate using sensitivity analyses and reviewing penetration data
Surface Projection Based Contact

- More accurate distribution of contact stresses.
- Satisfies moment equilibrium when an offset exists between contact and target surfaces with friction.
- Can help with contact convergence.
- Better handling of sliding contact.
- KEYOPT,<contact element type>,4,3

Default Contact Settings

Surface Projection Contact
Higher Order elements for curved surface contact for faster, more accurate results
Solver Type and Tolerance

SOLUTION OPTIONS

PROBLEM DIMENSIONALITY..............3-D
DEGREES OF FREEDOM.................UX UY UZ
ANALYSIS TYPE......................STATIC (STEADY STATE)
NONLINEAR GEOMETRIC EFFECTS........ON
EQUATION SOLVER OPTION..............PCG
TOLERANCE...........................1.000000E-08
NEWTON-RAPHSON OPTION..............PROGRAM CHosen
GLOBALLY ASSEMBLED MATRIX.........SYMmetric

*** WARNING ***
CP= 1.219  TIME= 22:43:37
Material number 18 (used by element 4426) should normally have at least one MP or one TB type command associated with it. Output of energy by material may not be available.

*** NOTE ***
CP= 1.234  TIME= 22:43:37
Present time 0 is less than or equal to the previous time.
Time will default to 1.

*** NOTE ***
CP= 1.234  TIME= 22:43:37
The step data was checked and warning messages were found.
Please review output or errors file (C:\DOCUME~1\sheldon\LOCALS~1\Temp\file.err) for these warning messages.

*** NOTE ***
CP= 1.234  TIME= 22:43:37
Nonlinear analysis, NROPT set to the FULL Newton-Raphson solution procedure for ALL DOFs.

*** NOTE ***
CP= 1.250  TIME= 22:43:37
The conditions for direct assembly have been met. No .emat or .erot files will be produced.
Correct the Element Distortion

→ Add intermediate substeps
→ Rezoning
→ Adjust the starting mesh shapes
→ Change Element type / formulation
→ Use shell or beam elements
→ Reduce delta stiffness reduction from Element Birth and Death
→ Change the HyperElastic material model
→ Modify creep law or coefficients
Select the appropriate element type to maximize results efficiency and quality

Complex 3-D geometries

Shell elements

Slender structures (twisted pipe model)
Remesh the model part way through the analysis

Before rezoning

After rezoning
Material nonlinearities occur because of the nonlinear path-dependent (except for the case of nonlinear elastic materials). The program can account for many material nonlinearities:

1. **Rate-independent plasticity** is characterized by a yield surface that is independent of the strain rate.
2. **Rate-dependent plasticity** allows the plastic deformation to be strain-rate sensitive.
3. **Creep** is also an irreversible straining that occurs much larger than that for rate-dependent plasticity.
4. **Gasket material** may be modelled using specific constitutive models that account for the interaction between the gasket material and the contacting surfaces.
5. **Nonlinear elasticity** allows a nonlinear stress-strain relationship to be defined.
6. **Hyperelasticity** is defined by a strain-energy function that relates the strain energy to the current deformation state.
7. **Viscoelasticity** is a rate-dependent material characteristic that allows for the representation of the time-dependent behavior of materials.
8. **Concrete** materials include cracking and crushing behavior.
9. **Swelling** allows materials to enlarge in the presence of moisture or other external factors.
## Hyperelastic material models

<table>
<thead>
<tr>
<th>Model Name</th>
<th>Abbreviation</th>
<th>Order/Options</th>
<th>No. of Coefficients</th>
<th>Linear/Nonlinear Fitting</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mooney-Rivlin</td>
<td>moon</td>
<td>2, 3, 5, 9</td>
<td>2/3/5/9+1</td>
<td>Linear</td>
</tr>
<tr>
<td>Polynomial</td>
<td>poly</td>
<td>1 to N</td>
<td>see below [2]</td>
<td>Linear</td>
</tr>
<tr>
<td>Yeoh</td>
<td>yeoh</td>
<td>1 to N</td>
<td>N+N</td>
<td>Linear</td>
</tr>
<tr>
<td>Neo-Hookean</td>
<td>neoh</td>
<td>-</td>
<td>1+1</td>
<td>Linear</td>
</tr>
<tr>
<td>Ogden</td>
<td>ogde</td>
<td>1 to N</td>
<td>2*N+N</td>
<td>Nonlinear</td>
</tr>
<tr>
<td>Arruda-Boyce</td>
<td>boyc</td>
<td>-</td>
<td>2+1</td>
<td>Nonlinear</td>
</tr>
<tr>
<td>Gent</td>
<td>gent</td>
<td>-</td>
<td>2+1</td>
<td>Nonlinear</td>
</tr>
<tr>
<td>Blatz-Ko</td>
<td>blat</td>
<td>-</td>
<td>1</td>
<td>Nonlinear</td>
</tr>
<tr>
<td>Ogden Hyper-foam</td>
<td>foam</td>
<td>1 to N</td>
<td>2*N+N</td>
<td>Nonlinear</td>
</tr>
</tbody>
</table>

Hyperelastic materials may only be used in solid elements for explicit dynamic simulations.

<table>
<thead>
<tr>
<th>Model</th>
<th>Applied Strain Range</th>
</tr>
</thead>
<tbody>
<tr>
<td>Neo-Hookean</td>
<td>30%</td>
</tr>
<tr>
<td>Mooney-Rivlin</td>
<td>30%-200% depending on order</td>
</tr>
<tr>
<td>Polynomial</td>
<td></td>
</tr>
<tr>
<td>Ogden</td>
<td>Up to 700%</td>
</tr>
</tbody>
</table>
The value of the one element test case

- Testing nonlinear material models
- Testing of macros and user-defined routines
- Evaluation of the impact of large aspect ratios, skew angles or warped elements
Robust well shaped elements can improve solution convergence and time
Numerous controls are available to locally modify the mesh to reduce stress gradients through multiple elements.
Automated mesh refinement will increase the solution quality and sometimes speed.
Element Formulation

Solver Output records the element technology being activated based on the element order chosen (midside nodes) and the material association.

Elastic material or metal plasticity with higher order elements

2D Plane Stress
Elastic material or Metal Plasticity with lower order elements

2D Plain Strain
Elastic material or Metal Plasticity with lower order elements

Fully incompressible hyperelasticity with higher or lower order elements

---

**Selection of Element Technologies for Applicable Elements**

--- Give Suggestions and Reset the Key Options ---

**Default URI**

- **Elastic material or Metal Plasticity with lower order elements**

- **Fully incompressible hyperelasticity with higher or lower order elements**

- **B-Bar with Mixed u-P**

---

**Enhanced Strain**

**Simplified Enhanced Strain**

---

**Element Type 1** is SOLID186. KEYOPT(2)=0 is suggested. Since it is associated with fully incompressible hyperelastic materials, KEYOPT(6)=1 must be used. They have been reset.

**Element Type 2** is PLANE182 with plane strain option. It is associated with linear materials only and Poisson's ratio is not greater than 0.49. KEYOPT(1)=3 is suggested and has been reset.

**Element Type 3** is PLANE183 with plane stress option. No suggestion is available and no resetting is needed.
You have a solution! How to make the solution more efficient

→ Change the mesh
  → Refine areas of steep gradient
  → Coarsen areas where stresses are low
  → Adjust initial element shapes to create better deformed shapes

→ Change the Substep settings
  → Add substeps to reduce bisections
  → Reduce substeps where convergence takes 1 or 2 iterations max.
The Force Convergence graph clearly indicates that starting with more substeps would eliminate the 27 iterations performed before the first bisection.
Thoroughly investigate your results
Check the quality of your results - forces

Applied loads

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0</td>
<td>0</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>2</td>
<td>1</td>
<td>2000.</td>
<td>1000.</td>
<td>0</td>
</tr>
<tr>
<td>3</td>
<td>2</td>
<td>0</td>
<td>500.</td>
<td>750.</td>
</tr>
<tr>
<td>4</td>
<td>3</td>
<td>2500.</td>
<td>0</td>
<td>1250.</td>
</tr>
<tr>
<td>5</td>
<td>4</td>
<td>750.</td>
<td>1500.</td>
<td>2500.</td>
</tr>
</tbody>
</table>

Matching reaction forces

<table>
<thead>
<tr>
<th>Time [s]</th>
<th>Force Reaction 3 (X) [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-2000.</td>
</tr>
<tr>
<td>2</td>
<td>-1.8111e-006</td>
</tr>
<tr>
<td>3</td>
<td>-2500.</td>
</tr>
<tr>
<td>4</td>
<td>-750.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Time [s]</th>
<th>Force Reaction 5 (Z) [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>8.3937e-005</td>
</tr>
<tr>
<td>2</td>
<td>-750.</td>
</tr>
<tr>
<td>3</td>
<td>-1250.</td>
</tr>
<tr>
<td>4</td>
<td>-2500.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Time [s]</th>
<th>Force Reaction 4 (Y) [N]</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>-1000.</td>
</tr>
<tr>
<td>2</td>
<td>-500.</td>
</tr>
<tr>
<td>3</td>
<td>4.4345e-005</td>
</tr>
<tr>
<td>4</td>
<td>-1500.</td>
</tr>
</tbody>
</table>
Check the quality of your stress results

Integration Point Results
- Display Option: Unaveraged
- Results:
  - Averaged
  - Minimum
  - Nodal Difference
  - Nodal Fraction
  - Maximum
  - Elemental Difference
  - Elemental Fraction

Equivalent Stress
- As: Static Structural (ANSYS)
- Equivalent Stress
- Type: Equivalent (von-Mises) Stress
- U1, MPA
- Time: 1
  - 2009/10/01 11:29
- Max:
  - 373.16
  - 220.29
  - 205.06
  - 201.93
  - 196.61
  - 191.48
  - 186.24
  - 94.428
  - 43.213
  - 1.8365 MPA

Equivalent Stress 2
- As: Static Structural (ANSYS)
- Equivalent Stress 2
- Type: Equivalent (von-Mises) Stress
- U1, MPA
- Time: 1
  - 2009/10/01 11:29
- Max:
  - 373.16
  - 220.29
  - 205.06
  - 201.93
  - 196.61
  - 191.48
  - 186.24
  - 94.428
  - 43.213
  - 1.8365 MPA

Equivalent Stress 3
- As: Static Structural (ANSYS)
- Equivalent Stress 3
- Type: Equivalent (von-Mises) Stress
- Nodal Difference
- U1, MPA
- Time: 1
  - 2009/10/01 11:29
- Max:
  - 20.622
  - 10.129
  - 16.195
  - 13.882
  - 11.560
  - 9.244
  - 6.928
  - 4.627
  - 2.316
  - 0 MPA

Equivalent Stress 4
- As: Static Structural (ANSYS)
- Equivalent Stress 4
- Type: Equivalent (von-Mises) Stress
- Nodal Fraction
- U1, MPA
- Time: 1
  - 2009/10/01 11:29
- Max:
  - 0.02916
  - 0.2106
  - 0.0705
  - 0.5407
  - 0.4725
  - 0.2495
  - 0.27435
  - 0.1829
  - 0.09415
  - 0 MPA

Geometry / Report Preview
Understand the data generated

Displacements are based on original coordinate system

Stresses and strain component data rotate with the elements
Obtaining & Optimizing Structural Analysis
Convergence

How to leverage the robustly converging model
Why Do Parametric Simulations?

Reduce design cycle
- Cost savings
  - Man power
  - Time to market
  - Prototyping

Design verification of existing or new designs

Design optimization to design a better device.

Sensitivity studies on product performance

Quality assurance
- Set manufacturing tolerances
- Meet six sigma requirements
- Satisfy ISO 9001:2000 regulations
You often need to go beyond single point solutions to answer the “what if” questions!
Parametric studies are a few mouse clicks away.
In Workbench; Check the box to create Parameters
Create custom parameters to create new performance criteria

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property</td>
<td>Value</td>
</tr>
<tr>
<td>Description</td>
<td>Price per 1000 units</td>
</tr>
<tr>
<td>Expression Type</td>
<td>Derived</td>
</tr>
<tr>
<td>Usage</td>
<td>Expression Output</td>
</tr>
<tr>
<td>Quantity Name</td>
<td>Money</td>
</tr>
</tbody>
</table>
Design points can be defined manually

**Table of Design Points**

<table>
<thead>
<tr>
<th>A</th>
<th>B</th>
<th>C</th>
<th>D</th>
<th>E</th>
<th>F</th>
<th>G</th>
<th>H</th>
<th>I</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Name</td>
<td>P1 - NeckWidth</td>
<td>P2 - BlendRadius</td>
<td>P3 - Total Deformation Reported Frequency</td>
<td>P4 - NeckStress Maximum</td>
<td>P5 - Total Deformation Maximum</td>
<td>P6 - Solid Mass</td>
<td>P7 - Temperature Minimum</td>
</tr>
<tr>
<td>2</td>
<td>Hz</td>
<td>MPa</td>
<td>mm</td>
<td>tonne</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Current</td>
<td>12</td>
<td>10</td>
<td>2273.5</td>
<td>347.6</td>
<td>0.055918</td>
<td>0.00090168</td>
<td>39.105</td>
</tr>
<tr>
<td>4</td>
<td>DP 1</td>
<td>12</td>
<td>8</td>
<td>2268</td>
<td>346.28</td>
<td>0.0556266</td>
<td>0.00089718</td>
<td>39.224</td>
</tr>
<tr>
<td>5</td>
<td>DP 2</td>
<td>15</td>
<td>10</td>
<td>2433.5</td>
<td>258.29</td>
<td>0.028904</td>
<td>0.00097182</td>
<td>37.33</td>
</tr>
<tr>
<td>6</td>
<td>DP 3</td>
<td>10</td>
<td>10</td>
<td>2171.2</td>
<td>386.73</td>
<td>0.082925</td>
<td>0.00085158</td>
<td>39.741</td>
</tr>
<tr>
<td>7</td>
<td>DP 4</td>
<td>18</td>
<td>10</td>
<td>257</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>DP 5</td>
<td>12</td>
<td>12</td>
<td>227</td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**Parameter Chart**

- **Y-axis**: Neck Stress Maximum (MPa)
- **X-axis**: Neck Width
Parametric analysis and DOE’s provide response surface data
Design of experiments will automate the development of response data.
Identify critical design parameters and how they relate to outputs
Use analyses to estimate the risk of failure

Fracture, creep

Buckling

Erosion, shocks
Automatically generate a detailed simulation report
Quantify the product life duration
End Goal - Better
Simulation-Driven Product Development

- Democartize Simulation
- Process Automation
- Enable Best Practices
- Focus on Engineering
- Complete Systems
- Simulated Environments
- Multiphysics
- Virtual Prototyping
- Fluids Dynamics
- Structural Mechanics
- Explicit Dynamics
- Low-Frequency Electromagnetics
- High-Frequency Electromagnetics
- Thermal Mechanics
- Acoustics

Dynamic CAE Collaboration
- Span Organizational and Geographic Silos
- Share Engineering Insights
- Better Decisions Faster