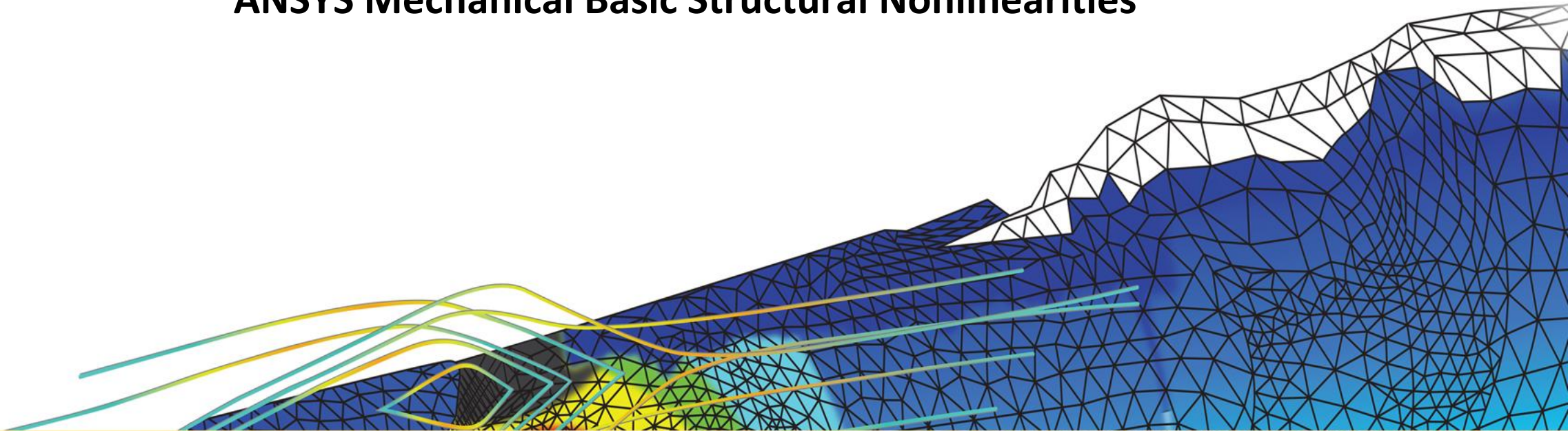# Module 07: Nonlinear Diagnostics

ANSYS Mechanical Basic Structural Nonlinearities

Module 07 Topics
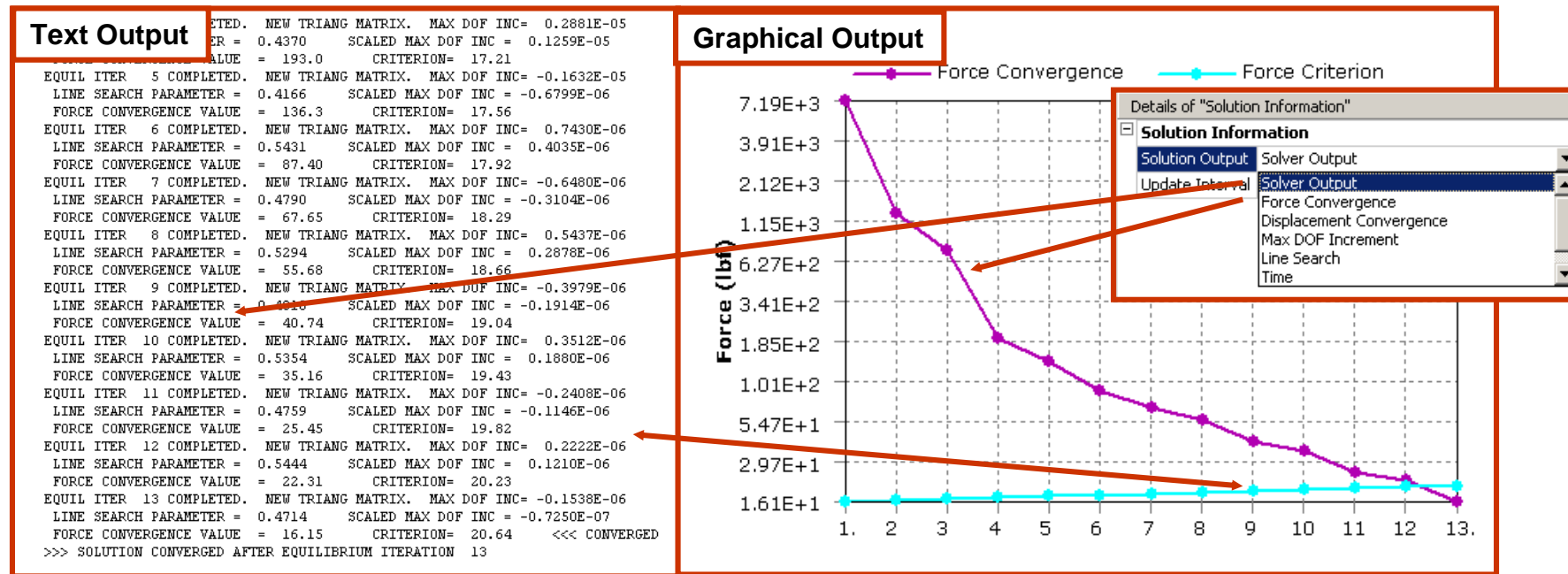
1. **Solver Output**

2. **Monitoring the Solution**

3. **Newton-Raphson Residuals**

4. **Example Cases**

5. **Tips on Solving the Model**

6. **Summary**

7. **Workshop 07.1: Diagnostic Tools for Contact**

Since some nonlinear structural analyses can be challenging to solve, understanding how to diagnose nonconvergence is critical in obtaining useful answers.

07.01 Solver Output

## In Module 01, the Solution Information branch was introduced.

- Recall that with the Solution Information branch, the detailed Solver Output can be reviewed, and convergence graphs, such as the Force Convergence behavior, can be plotted.

07.01 Solver Output

A "Messages" Window located directly below the Solution Information Worksheet offers a summary listing of some general warnings and errors.

- **RMB on any message to:**
  - **Go to Object (Highlights Project Tree Object relevant to the message)**
  - **Show Full Message in separate expanded window**
  - **Copy message (to separate text file)**
  - **Delete message from list**

 May 9, 2017
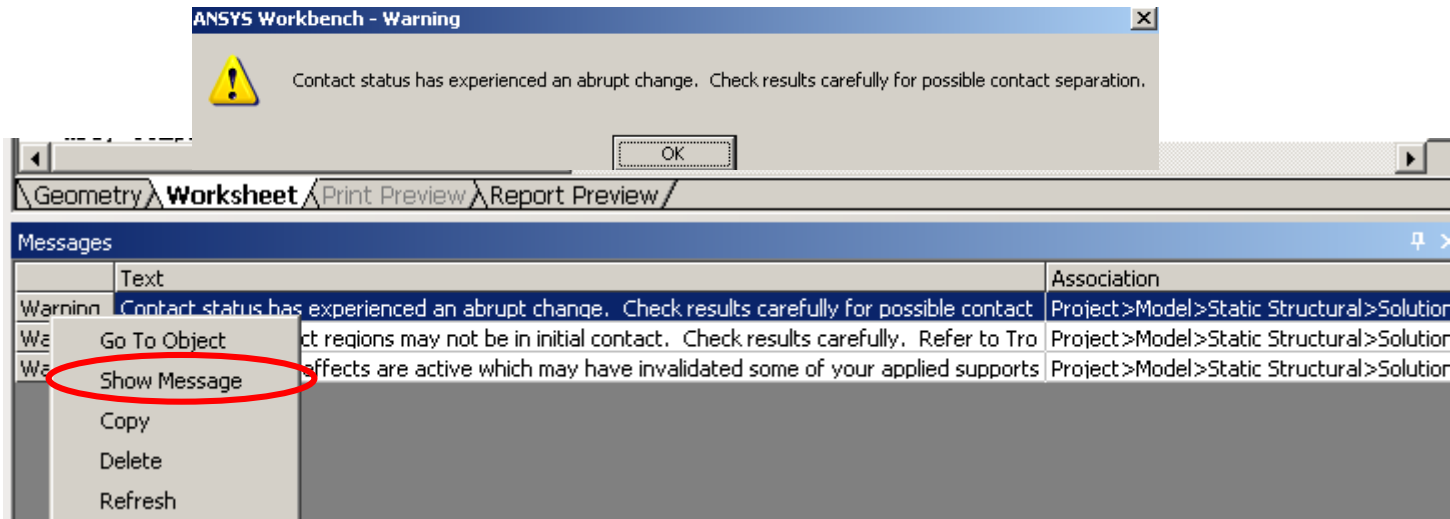
07.01 Solver Output

- **The Solver Output can provide detailed text output about the solution. It is useful to become familiar with how to read this file.**

  - **The beginning of the Solver Output simply shows the ANSYS license used (in this case, ANSYS Multiphysics) and the version number.**

© 2017 ANSYS, Inc.   May 9, 2017

07.01 Solver Output

- **Scrolling down from the top, the user directory where solver files will be stored is recorded along with a record of solver units**

```
PARAMETER _WB_PROJECTSCRATCH_DIR(1) = E:\1269286445\_ProjectScratch\Scr4CE5\

SET PARAMETER DIMENSIONS ON  _WB_SOLVERFILES_DIR
 TYPE=STRI  DIMENSIONS=        248          1          1

PARAMETER _WB_SOLVERFILES_DIR(1) = E:\1269286445\WB6a-diagnostics_files\dp0\SYS\MECH\

SET PARAMETER DIMENSIONS ON  _WB_USERFILES_DIR
 TYPE=STRI  DIMENSIONS=        248          1          1

PARAMETER _WB_USERFILES_DIR(1) = E:\1269286445\WB6a-diagnostics_files\user_files\
--- Data in consistent NMM units.

MPA UNITS SPECIFIED FOR INTERNAL
 LENGTH       = MILLIMETERS (mm)
 MASS         = TONNE (Mg)
 TIME         = SECONDS (sec)
 TEMPERATURE = CELSIUS (C)
 TOFFSET      = 273.0
 FORCE        = NEWTON (N)
 HEAT         = MILLIJOULES (mJ)

 INPUT  UNITS ARE ALSO SET TO MPA
L
```

ANSYS®

07.01   Solver Output

- Scrolling down further until a series of asterisks are encountered, the reading of the finite element data by the solver can be seen.

  This listing is useful, as will be shown later, because it not only provides information on how many parts are in the model, but the Contact Region ID numbers are listed here
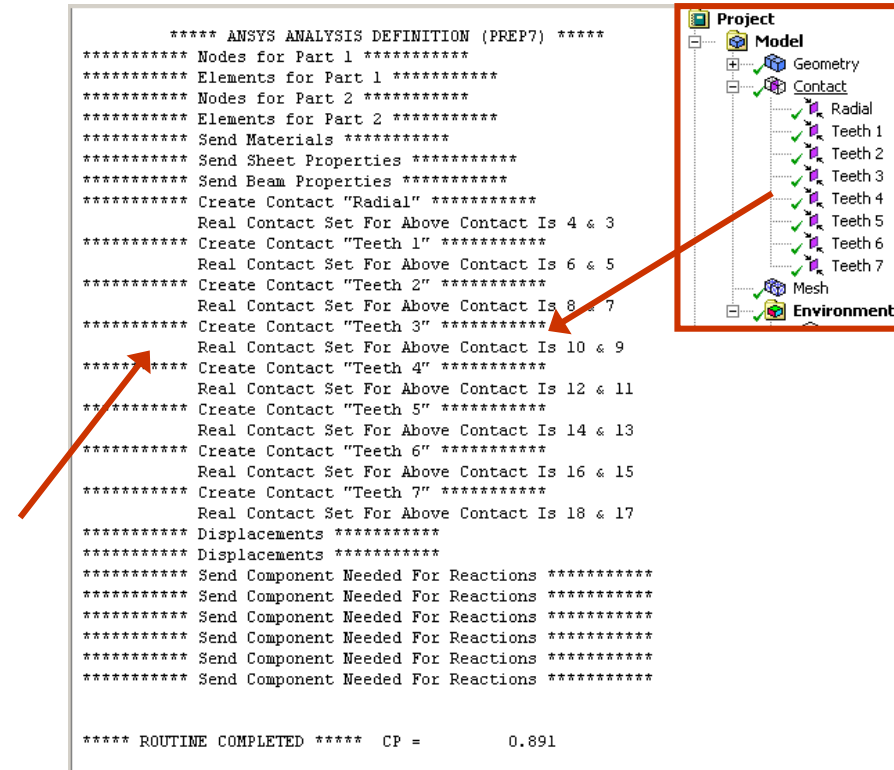
It is instructive to note that while Contact Regions can be given any name in Mechanical, the ANSYS solver treats each Contact Region with a unique number (ID). *For debugging purposes, it is useful to find out which Contact Region has which ID number. For example, in the above snippet, Contact Region "Teeth 3" is referenced by contact ID 9 and 10.*

07.01 Solver Output

- **Solver Output also records the element technology being activated based on the element order chosen (midside nodes) and the material association. See course ANSYS Mechanical Material Nonlinearities for details on Element Technology.**

Elastic material or metal plasticity with higher order elements

2D plane stress/strain metal plasticity with lower order elements

2D plane strain elastic material with lower order elements

```
*** SELECTION OF ELEMENT TECHNOLOGIES FOR APPLICABLE ELEMENTS ***
      --- GIVE SUGGESTIONS AND RESET THE KEY OPTIONS ---

ELEMENT TYPE     1 IS PLANE183 WITH PLANE STRESS OPTION. NO SUGGESTION IS
AVAILABLE AND NO RESETTING IS NEEDED.     Default URI
```

```
ELEMENT TYPE   2 IS PLANE182 WITH PLANE STRESS OPTION. IT IS ASSOCIATED WITH
ELASTOPLASTIC MATERIALS ONLY. KEYOPT(1)=2 IS SUGGESTED AND HAS BEEN RESET.
   KEYOPT(1-12)=    2     0    3    0    0    0    0    0    0    0    0    0
```
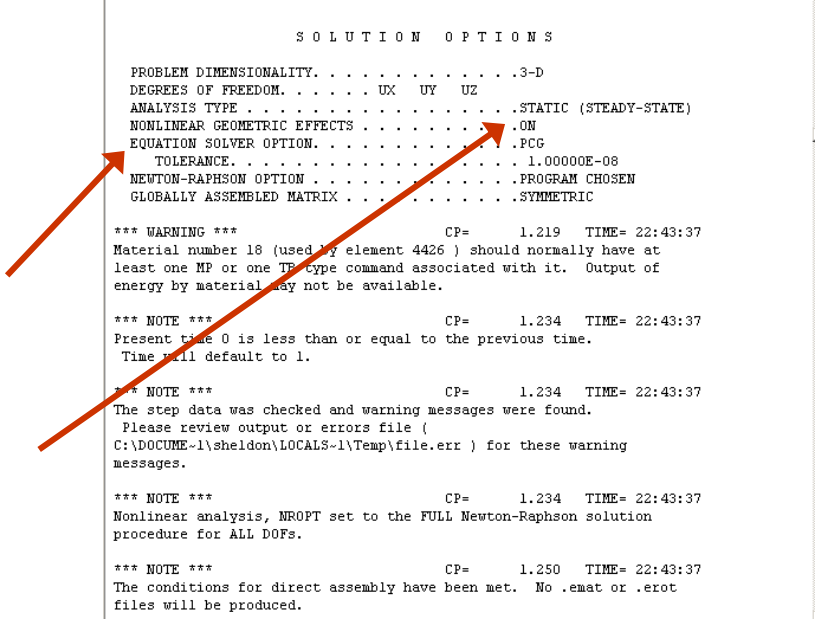*Enhanced Strain*

```
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.
   KEYOPT(1-12)=    3     0    2    0    0    0    0    0    0    0    0    0
```
Simplified Enhanced Strain

ANSYS®

07.01   Solver Output

- When the equation solution is initiated, the section of the output will be shown as on the right

  The useful things to review here are the equation solver used (if left at "Program Chosen" or manually specified), whether large deflection effects are on or off, whether nonlinear material effects are considered (if plasticity is present), and the number of substeps used.

```
                S O L U T I O N   O P T I O N S

PROBLEM DIMENSIONALITY. . . . . . . . . . . .3-D
DEGREES OF FREEDOM. . . . . . UX   UY   UZ
ANALYSIS TYPE . . . . . . . . . . . . . . . .STATIC (STEADY-STATE)
NONLINEAR GEOMETRIC EFFECTS . . . . . . . . .ON
EQUATION SOLVER OPTION. . . . . . . . . . . .PCG
   TOLERANCE. . . . . . . . . . . . . . . 1.00000E-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.
```

The review of this section of the Solver Output is not critical, but it indicates when the matrices are being solved and what the solution options specified in WB-Mechanical were.

ANSYS®

07.01   Solver Output

- **Details of contact elements are then printed next. Here, various options related to contact elements, including the contact Normal Stiffness and Pinball Radius will be listed**

Any NOTE or WARNING messages printed in this section are useful to review.

For example, initial penetration or gaps (in active unit length) will be shown in this area

```
*** 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-01
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=      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
```

```
*** NOTE ***                          CP=      9.333   TIME= 08:33:24
Max.  Initial penetration 8.249982762E-02 was detected between contact
element 18611 and target element 18504.
You may move entire target surface by : x= -5.839095973E-02, y=
6.44109704E-18, 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 1.548585102E-02 was detected between contact element
19330 and target element 19283.
The gap is closed due to initial adjustment.
*****************************************
```

Recall previously that the Contact Region name and the contact set ID are listed. From that example, we know that the above contact set 9 is part of "Teeth 3" region.

07.01   Solver Output

- As the nonlinear solution progresses, the equilibrium iteration information is shown at the bottom (sample below)

  – Note that for each equilibrium iteration, the residual forces (FORCE CONVERGENCE VALUE) must be lower than the CRITERION

  – Ideally, the residual or out-of-balance forces should be zero for a system to be in equilibrium. However, because of machine precision and practical concerns, WB-Mechanical determines a value small enough to result in negligible error. This value is the CRITERION, and the FORCE CONVERGENCE VALUE must be smaller than the CRITERION for the substep to be converged.

  – In the example below, after 3 equilibrium iterations, the residual forces are lower than the criterion, so the solution is converged.

  – *Informative messages (such as convergence or bisection) are noted with ">>>" and "<<<" in the output.*

```
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
```

ANSYS®

07.01   Solver Output

Warning and error messages will also be printed in the output

- When contact status changes abruptly, this is just a warning indicating that the contact elements enter or exit the 'pinball region' drastically. This may be due to parts sliding or separating drastically if the load is too high. WB-Mechanical may automatically bisect the solution, if necessary.

- Element distortion messages are usually severe problems due to excessive loading or over-constraints. Bisection of the load is automatically performed, but sometimes corrective measures may need to be taken to fix the problem.

```
*** WARNING ***                        CP=      880.286   TIME= 09:02:16
Contact element 18718 (real ID 51) status changes abruptly from contact
(with target element 19150) -> no-contact.
     LINE SEARCH PARAMETER =   0.2883      SCALED MAX DOF INC = -0.1944E-02
     FORCE CONVERGENCE VALUE   = 0.1015E+06  CRITERION=   19.88
     EQUIL ITER   12 COMPLETED.   NEW TRIANG MATRIX.   MAX DOF INC= -0.6402E-02

*** WARNING ***                        CP=      965.959   TIME= 09:05:06
Contact element 19035 (real ID 51) status changes abruptly from
no-contact -> contact (with target element 19191).
     LINE SEARCH PARAMETER =    1.000      SCALED MAX DOF INC = -0.6402E-02
     FORCE CONVERGENCE VALUE   = 0.1195E+07  CRITERION=   20.31
     EQUIL ITER   13 COMPLETED.   NEW TRIANG MATRIX.   MAX DOF INC=  0.2628E-01

*** ERROR ***                          CP=     1034.287   TIME= 09:07:14
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.
```

ANSYS®

07.01   Solver Output

Lastly, after the solution has completed, the very end of the Solver Output provides some statistics on the analysis run.

The percentage of solver time used for pre-processing vs. other elements can be determined here, including equation solver time.

```
 NUMBER OF WARNING MESSAGES ENCOUNTERED=          0
 NUMBER OF ERROR    MESSAGES ENCOUNTERED=          0

+--------- D I S T R I B U T E D   A N S Y S   S T A T I S T I C S -----------+

Release: Release 17.0        Build: 17.0      Update: UP20151214   Platform: WINDOWS x64
Date Run: 04/27/2016   Time: 16:54
Operating System: Windows 7 Service Pack 1 (Build: 7601)
Windows Process ID: 6588

Processor Model: Intel(R) Core(TM) i7-4810MQ CPU @ 2.80GHz

Compiler: Intel(R) FORTRAN Compiler Version 15.0.2  (Build: 20150121)
          Intel(R) C/C++ Compiler Version 15.0.2  (Build: 20150121)
          Intel(R) Math Kernel Library Version 11.2.3 Product Build 20150413

Number of machines requested          :    1
Total number of cores available       :    4
Number of physical cores available    :    4
Number of processes requested         :    2
Number of threads per process requested :  1
Total number of cores requested       :    2 (Distributed Memory Parallel)
MPI Type: INTELMPI
MPI Version: Intel(R) MPI Library 5.0 Update 3 for Windows* OS


GPU Acceleration: Not Requested

Job Name: file0
Input File: dummy.dat

  Core            Machine Name   Working Directory
  ---------------------------------------------------
    0
    1

Latency time from master to core     1 =    0.555 microseconds

Communication speed from master to core     1 =  4951.05 MB/sec

Total CPU time for main thread        :         5.4 seconds
Total CPU time summed for all threads :         5.7 seconds

Elapsed time spent pre-processing model (/PREP7) :     0.0 seconds
Elapsed time spent solution - preprocessing      :     0.0 seconds
Elapsed time spent computing solution            :     3.9 seconds
Elapsed time spent solution - postprocessing     :     0.1 seconds
Elapsed time spent post-processing model (/POST1) :    0.0 seconds

Equation solver used                  :         Sparse (symmetric)
Equation solver computational rate    :    11771.3 Mflops

Maximum total memory used             :       267.0 MB
Maximum total memory allocated        :      3136.0 MB
Maximum total memory available        :        32 GB

+------ E N D   D I S T R I B U T E D   A N S Y S   S T A T I S T I C S ------+
```

The very end shows the total Elapsed time.

- If elapsed time is much larger than CPU time (such as 2x) for a single processor, then that shows that much of the computational time may have been affected by slow disk I/O.

- If dual processors are used, the CPU time will not be as accurate, as it is the *sum* of the time used by both processors.

            May 9, 2017

ANSYS®

07.01   Solver Output

Becoming familiar with the contents and structure of the Solver Output can be very useful in understanding the finite element solution.

- Users can review the Solver Output *during solution* to find detailed information of the current equilibrium iteration.

- The Solver Output can be reviewed *after solution* to determine reasons for nonconvergence (if the solution had failed), to obtain information on solver performance, and find out specifics of contact element settings.

- Detailed warnings or error messages (marked with ***), if present, will also be printed in the Solver Output. Also, during solution, review of the substep status (denoted with >>>) will show the reasons for bisection, if any.

ANSYS®

07.02 Monitoring the Solution

While solving a nonlinear model, because many iterations may be needed, it is useful to see what the nonlinear solution trends are.

- If the solution seems to be behaving unexpectedly, the user can stop the analysis and investigate the problem rather than wait until the solution is complete. This helps to save time.

- Monitoring the solution also helps the user gain an understanding into the response of the system.

There are two ways in which users can monitor the solution in WB-Mechanical:

- Solution Information branch to provide equation solver behavior.

- Results Tracker to provide response of system during the solution.

07.02   Monitoring the Solution

The most useful way to evaluate the solution convergence behavior is to view the Force Convergence graph:

- As shown in the previous section, the Force Convergence graph is available from the Solution Information branch
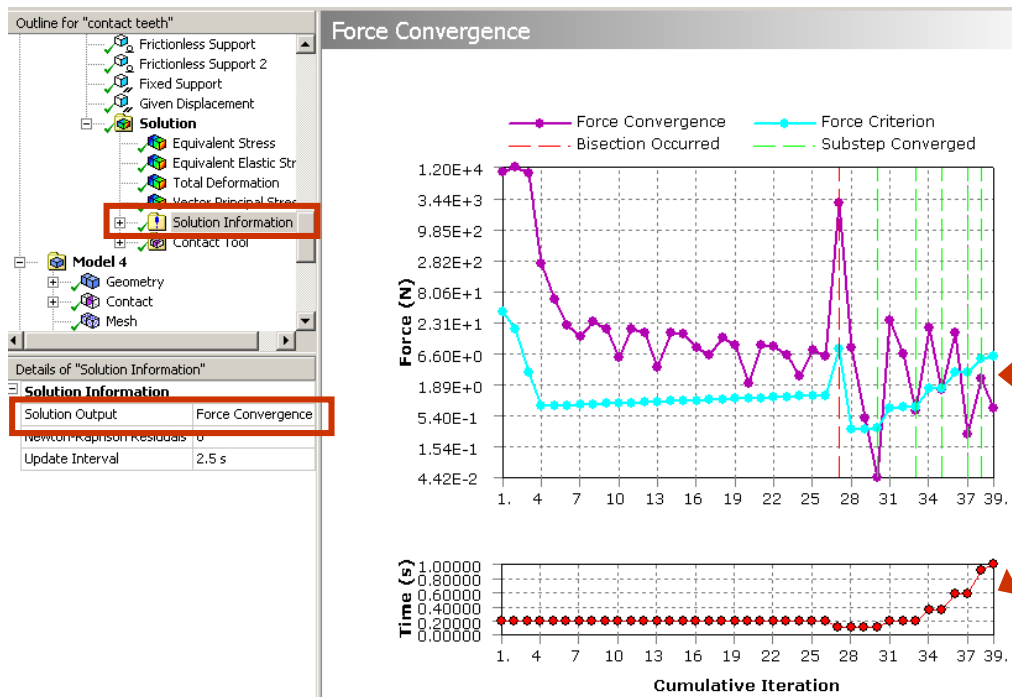
Recall that, in the Newton-Raphson method, force equilibrium is sought. If the out-of-balance (residual) forces is below the force criterion, the substep is assumed to be converged.

During solution, one can review the Force Convergence graph to monitor the progress.

The top Force Convergence graph shows that a bisection occurred. This means that a smaller load increment needed to be applied.

The bottom "TIME" graph represents the fraction of total load. Although Time has no significance in a static analysis, it is used as a counter where Time=1.0 is the final solution. If the Time is currently 0.2, that means that 20% of the load is being applied.

07.02   Monitoring the Solution

One common item is when the force convergence starts to "plateau" during solution.

- **This usually indicates either that (a) a smaller increment of the load should be applied or (b) contact Normal Stiffness may be too high.**

- **WB-Mechanical will take care of case (a) by bisecting the solution.**

- **If the reason is related to case (b), either bisection or manually lowering the contact Normal Stiffness will help.**
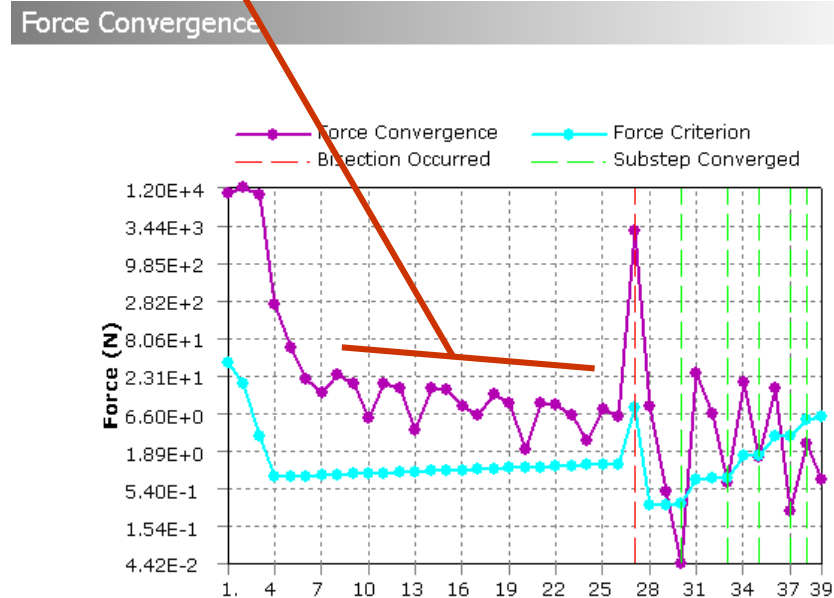
07.02   Monitoring the Solution

Besides monitoring the out-of-balance forces, a *Results Tracker* is available from the "Solution Information" branch.

- The Results Tracker enables users to monitor deformation at a vertex and/or contact region information during the solution.

- For "Results Tracker > Deformation," select a vertex of interest and specify whether x, y, or z deformation is to be monitored.

- For "Results Tracker > Contact," a pull-down menu enables users to select a contact region. Then, the quantity to track (such as number of contacting elements) can be displayed.

Contact Result Trackers can be added before solving or when the solution is in progress.

ANSYS®
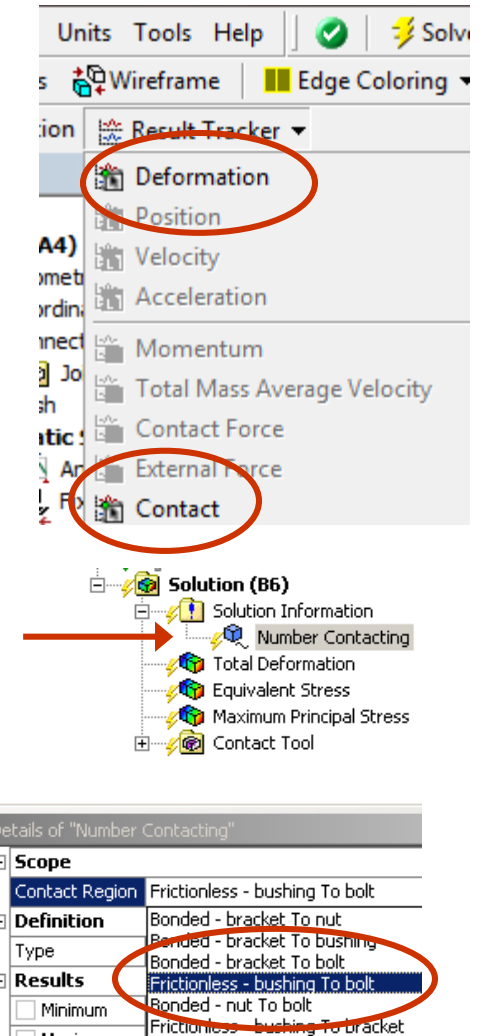
07.02   Monitoring the Solution

After the Results Tracker items are requested and solution initiated, users may "track" the deformation or contact results during the course of the solution.

In this example, the number of contacting elements is monitored for a particular contact region. As is apparent in the graph on right, between Time=1.4 and 1.7, the number of contacting elements jumps from zero to 29. Since "Time" is a "placeholder" in a nonlinear static analysis, this means that, after the first load step (Time=1.0), between 40% and 70% of the load, contact is established.
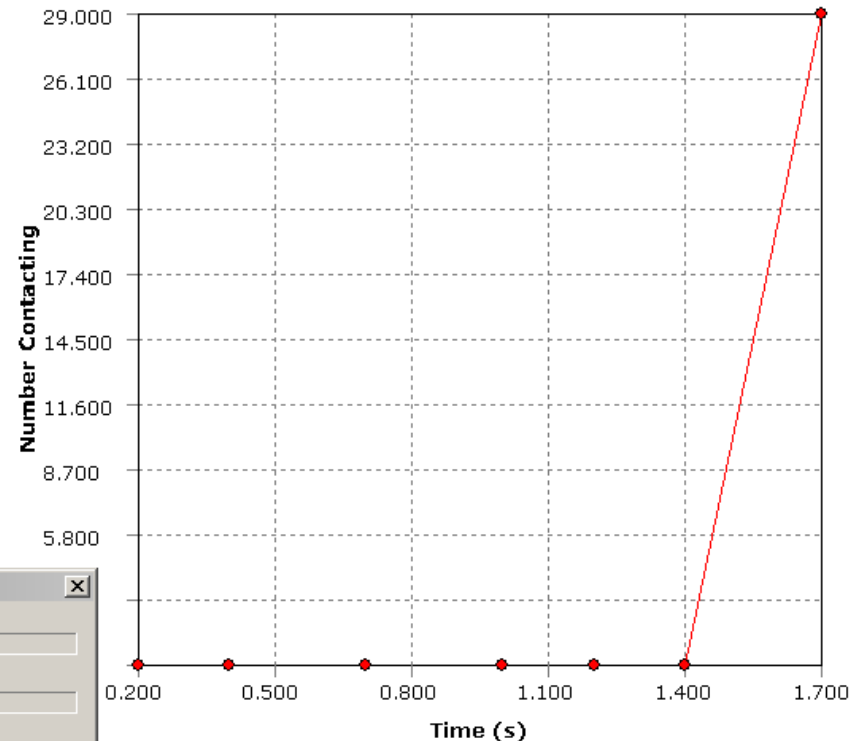
07.02   Monitoring the Solution

Several items (of a similar type) may also be selected and reviewed at once during solution

In this case, the penetration of eight Contact Regions is shown

One can review whether maximum penetration is too great – if so, the solution can prematurely be stopped, and the Normal Stiffness of the *appropriate* Contact Region increased since the user knows which Contact Region has too much penetration.

Also, from this, if the user expects the Contact Region "Radial" to come into contact, but that is not reflected while monitoring, the user will also know what results to review after solution is complete.

In this example, Contact Regions "Teeth 1-4" are in contact by the second substep. "Teeth 7" doesn't seem to come into contact until the fourth substep. This shows the user during solution what Contact Regions are engaging and when.
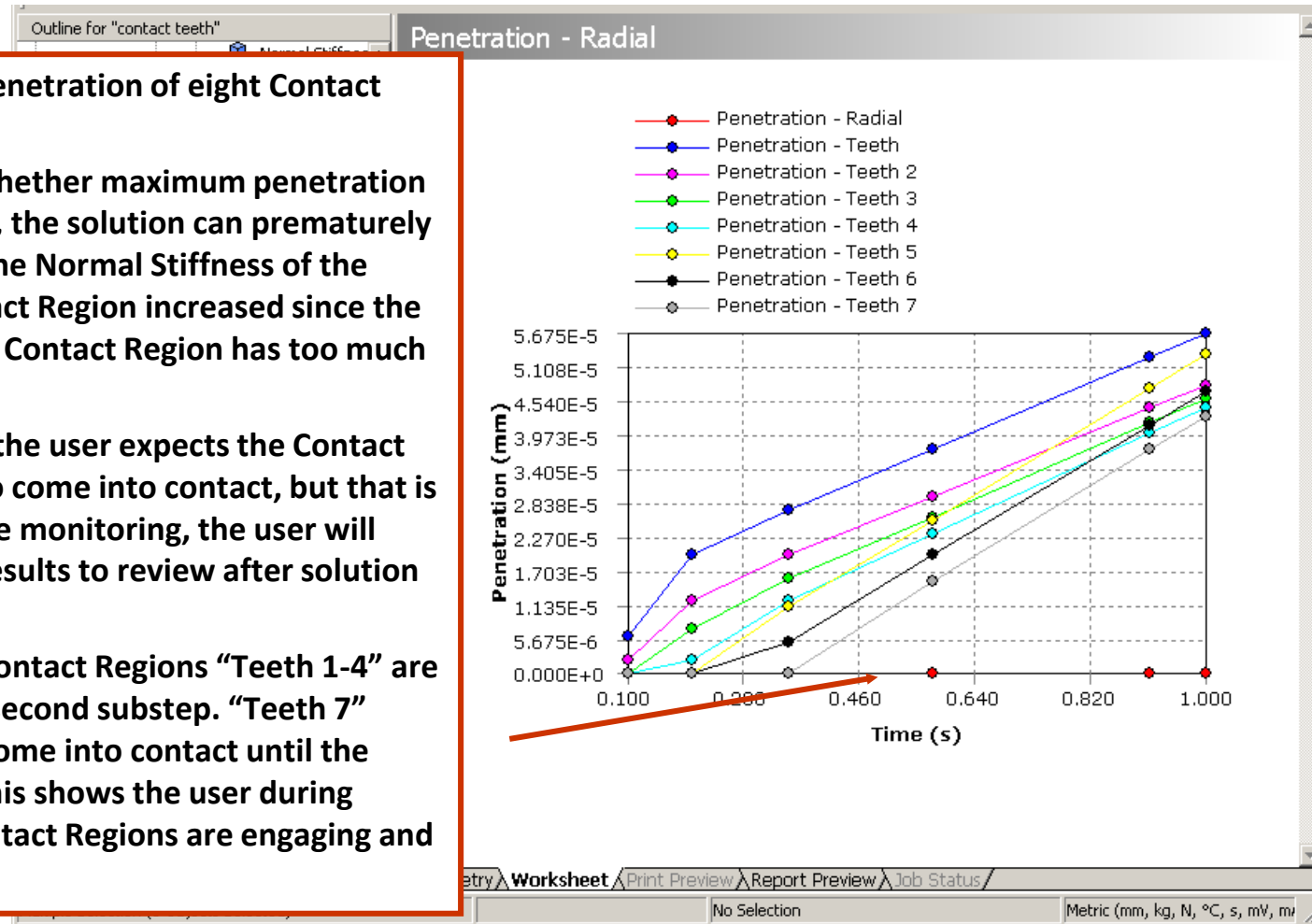
07.03 Newton-Raphson Residuals

So far, obtaining detailed solver information and monitoring the solution have been discussed. In the event that nonconvergence is encountered, the Newton-Raphson Residuals are very useful in locating possible problem areas.

- If the solution does not converge because of force equilibrium, this will be reflected in the Solver Output and Force Convergence graph. The Force Convergence (residuals) will be greater than the Force Criterion.

- In this situation of nonconvergence, the Newton-Raphson Residuals (if requested) will show which areas had high force residuals. This usually helps to pinpoint what locations prevented force equilibrium, usually either because of the Load and Supports at that location or because of Contact Region settings.

ANSYS

07.03 Newton-Raphson Residuals

In the "Solution Information" details view, enter the number of equilibrium iterations to retrieve Newton-Raphson Residuals. For example, if "4" is entered, the residual forces from the last four iterations will be returned if the solution is aborted or does not converge.

       May 9, 2017

07.03   Newton-Raphson Residuals

After solution is stopped or fails to converge, residuals will be available under the "Solution Information" branch, as shown below.
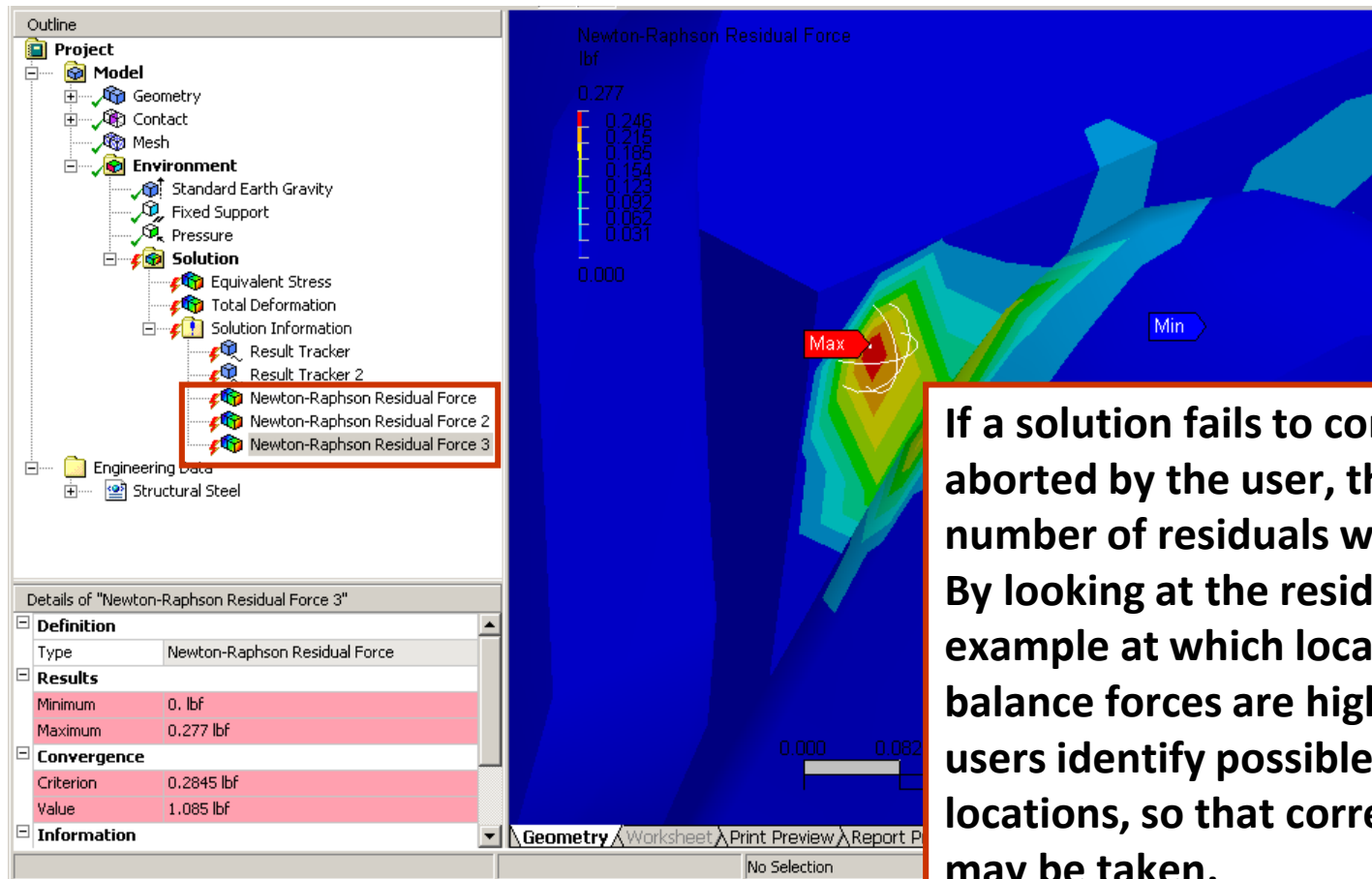

If a solution fails to converge or is aborted by the user, the requested number of residuals will be available. By looking at the residuals, one can example at which locations out-of-balance forces are high. This helps users identify possible problematic locations, so that corrective action may be taken.

07.03 Newton-Raphson Residuals

Another example is shown below. The force residuals are high (not in equilibrium), and the Newton-Raphson Residuals allow the user to see what areas may contribute to the high out-of-balance forces.

- **In this case, the 'ring' of high residual forces is part of a contact region, so the user knows where to examine.**
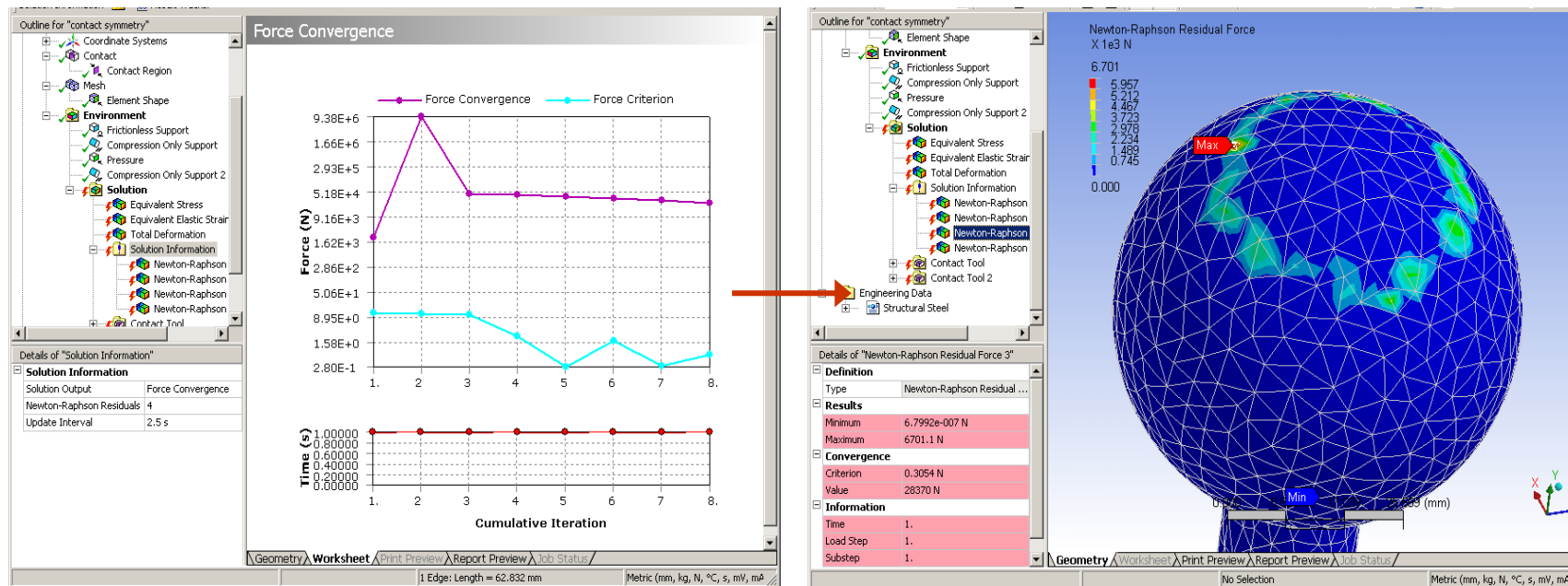
07.04   Example Cases

In the present section, some different scenarios will be briefly covered. Reviewing Solver Output, monitoring the solution, and performing nonlinear diagnostics will be discussed for the different cases.

- It is impractical to cover all of the different causes for nonconvergence. Instead, some common problems users encounter will be discussed.

May 9, 2017

07.04   Example Cases

A common error message that may appear is "internal solution magnitude limit was exceeded." Essentially, this means that rigid-body motion is encountered.

- **Insufficient constraints, either with Supports or Contact Regions, may allow for parts to 'fly off' into space**
- **The Solver Output may also provide the rigid-body direction**

07.04   Example Cases

- To check what parts are undergoing rigid-body motion, one can perform a *free vibration analysis* and look for near-zero frequency modes. This can be *computationally expensive*.

- *Another method is to look at contact pairs to see which ones are initially open.* Recall that the first part of the Solver Output relates contact ID with Contact Region name.

In this example, "contact offset block-2 To contact offset bolt-1" contact region (set #10 and 11) has an initial gap. One can check the model to see whether or not the gap should be there and if it may be causing the rigid-body motion because of lack of initial contact being established.

```
*** NOTE ***                              CP=      3.084   TIME= 12:27
Symmetric Deformable- deformable contact pair identified by real
constant set 10 and contact element type 10 has been set up.   The
companion pair has real constant set ID 11.  Both pairs should have
the same behavior.
For asymmetric contact analysis, you may keep the current pair and
deactivate its companion pair.
Contact algorithm: Augmented Lagrange method
Contact detection at: Gauss integration point
Contact stiffness factor FKN            1.0000
The resulting contact stiffness         0.31297E+06
Default penetration tolerance factor FTOLN   0.10000
The resulting penetration tolerance     1.0810
Frictionless contact pair is defined
Average contact surface length          7.3741
Average contact pair depth              10.810
Pinball region factor PINB              1.0000
The resulting pinball region            10.810
Initial penetration will be ramped during the first load step.


*** NOTE ***                              CP=      3.084   TIME= 12:27:33
Min.  Initial gap 0.997786449 was detected between contact element 9003
and target element 9157.
You may move entire target surface by: x= 0.816899846, y=
1.934807504E-15, z= 0.572933191, to bring it in contact.
*************************************
```

```
******* Send Beam Properties ***********
******* Create Contact "contact offset block-2 To contact offset block-1" *
        Real Contact Set For Above Contact Is 5 & 4
******* Create Contact "contact offset bolt-1 To contact offset block-2" **
        Real Contact Set For Above Contact Is 7 & 6
******* Create Contact "contact offset bolt-1 To contact offset block-1" **
        Real Contact Set For Above Contact Is 9 & 8
******* Create Contact "contact offset block-2 To contact offset bolt-1" **
        Real Contact Set For Above Contact Is 11 & 10
******* Create Contact "contact offset block-1 To contact offset bolt-1" **
        Real Contact Set For Above Contact Is 13 & 12
******* Displacements ***********
******* Create Pressure(s) ***********
******* Displacements ***********
******* Create Pressure(s) ***********
******* Displacements ***********
```

ANSYS®

07.04   Example Cases

Sometimes, contact Normal Stiffness may be too high and contribute to nonconvergence

- A threaded fastener solves the first substep until 20% of the load, but then diverges.
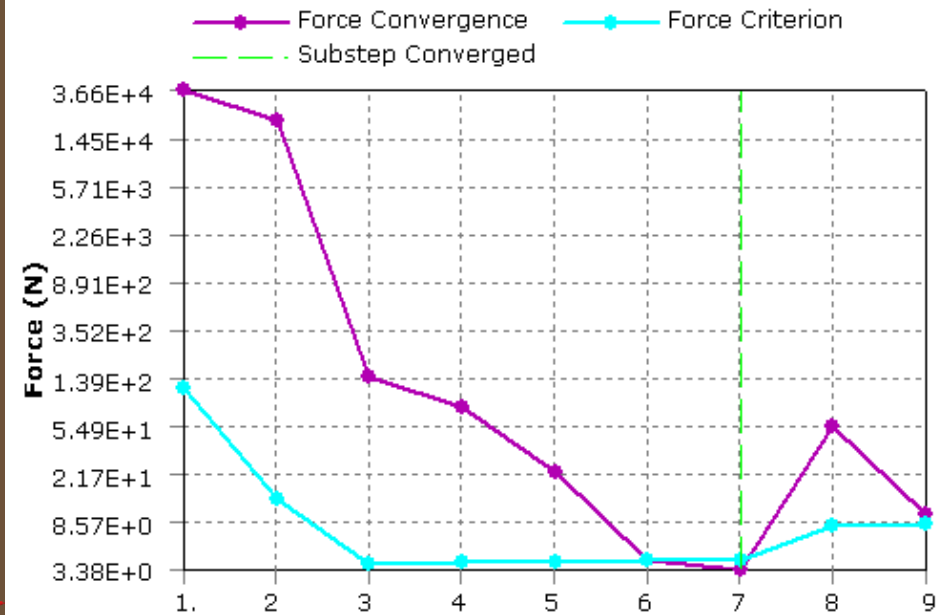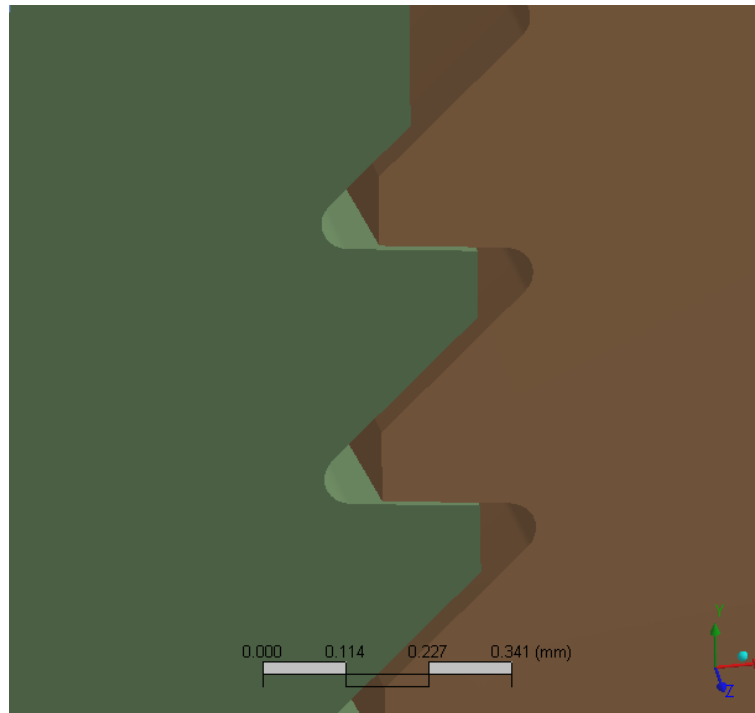
07.04   Example Cases

- **By looking at the Newton-Raphson residuals, it seems that force balance could not be achieved at areas of Contact Regions. The mesh also looks very coarse.**

- **By lowering the contact Normal Stiffness and refining the mesh near regions of contact, the problem can be solved.**

Unconverged Newton-Raphson Residuals

Converged Solution after Adjustments

May 9, 2017
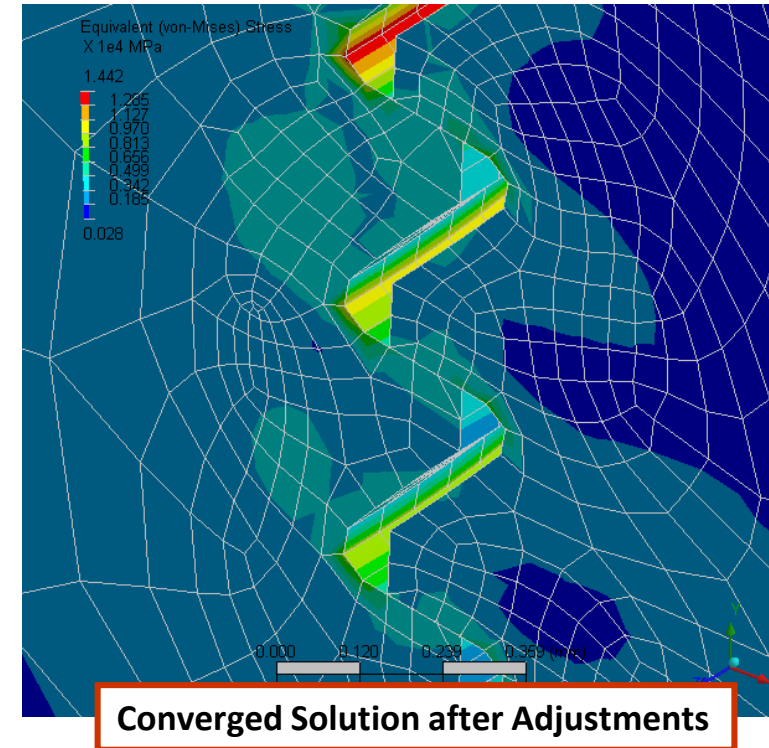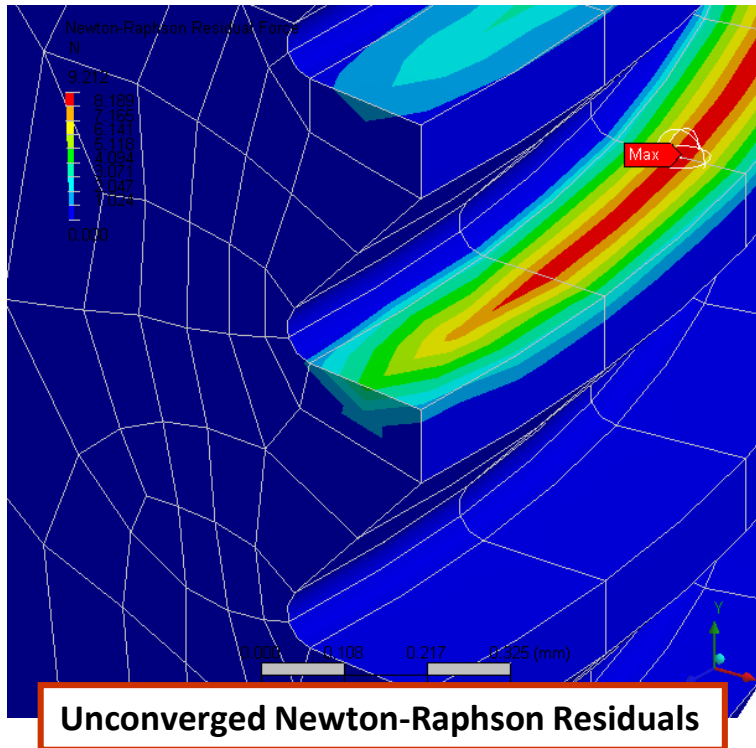
07.04   Example Cases

The error message "solver engine was unable to converge on a solution for the nonlinear problem as constrained" can be due to several factors, although it is often a good idea to double-check the model setup if this message is encountered.

In this model, Pretension Bolt Loads are present, so there are two load steps.
The first load step (preload) converged without problems. The second load step, however, had problems where not even one substep converged.
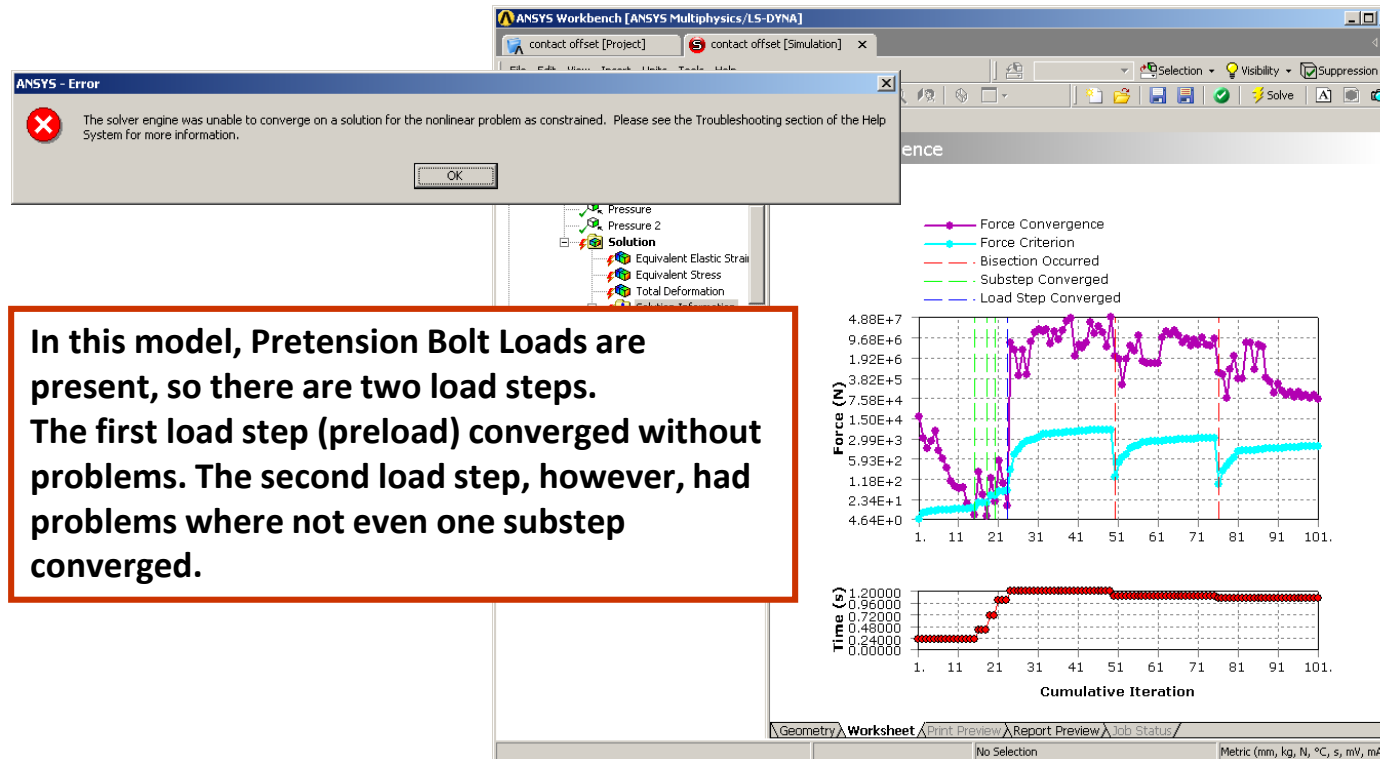
07.04   Example Cases

By examining the Results Tracker and Solver Output, one can determine the cause of nonconvergence:

- During the first load step, only one Contact Region is holding the parts together, as shown in the Results Tracker on left.

- From the Solver Output on right, the Contact Region holding the parts together has lost contact. This indicates that, most likely, the Loads applied have made the parts lose contact.

Things to check:
Were other Contact Regions supposed to be in initial contact? Was load magnitude incorrectly applied? Were any other sources of constraint omitted?
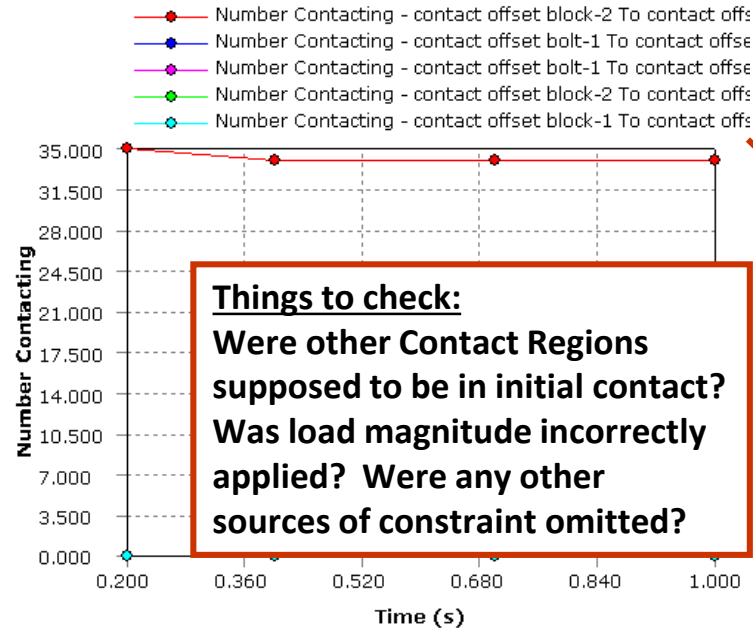
```
*** ERROR ***                        CP=    922.656   TIME= 11:34:37
Solution not converged at time 1.05 (load step 2 substep 1).
 Run terminated.

*****************************************************
 SUMMARY FOR CONTACT PAIR IDENTIFIED BY REAL CONSTANT SET          4

*** NOTE ***                         CP=    922.656   TIME= 11:34:37
No contact was detected.
85 contact points have abrupt change in contact status.
Contact element 7245 has the highest chattering level 4.
Max.  Pinball distance 21.8710032.
*****************************************************


*****************************************************
 SUMMARY FOR CONTACT PAIR IDENTIFIED BY REAL CONSTANT SET          5

*** NOTE ***                         CP=    922.656   TIME= 11:34:37
No contact was detected.
85 contact points have abrupt change in contact status.
Contact element 7352 has the highest chattering level 4.
Max.  Pinball distance 21.8710032.
*****************************************************


*****************************************************
 SUMMARY FOR CONTACT PAIR IDENTIFIED BY REAL CONSTANT SET          6
```

ANSYS®

07.05   Tips on Solving the Model

As shown in the previous section, the following combination of steps help to pinpoint possible sources of nonconvergence:

- Look at Force Convergence graph to see how solution is behaving.

- Check Solver Output for contact information (are Contact Regions initially in contact? What is the initial gap/penetration, if any?) and any warnings or errors during solution.

- Use Results Tracker to monitor contact information or deformation at vertex. Is the model behaving as expected?

- If nonconvergence occurs, check Newton-Raphson Residuals to find locations of high residuals, which reflect possible problem areas. Are there Loads or Supports applied in those problem areas, or are the areas part of a Contact Region? Double-check model setup.

07.05   Tips on Solving the Model

If those steps still don't provide enough information on what the problem is, there are other things that can be done:

- If plasticity is present and excessive element distortion occurs, try running the model without plasticity first to determine if the material model is the cause of the problem.

  - If the problem can be isolated to the plasticity material, check the plasticity definition. Does it become perfectly plastic? If so, can a plastic hinge (mechanism) form? Is it possible that elements have such little stiffness that they can distort too much?

- For contact problems, duplicate the branch and run the model with Bonded Type contact first to see if the problem can be isolated to the contact definition.

  - If it is contact causing problems and force residuals are too high, change all contact to Pure Penalty and enter a low Normal Stiffness (0.01 to 0.1). Try solving the model then – there may be excessive penetration, but if results are obtained, the user can get a sense of how the parts are interacting with contact.

ANSYS®

07.05   Tips on Solving the Model

If Normal Lagrange Formulation is used for contact, check chattering status.

- Chattering information is printed during solution as well as at the end if nonconvergence is met – the user can also request Contact Chattering in the Results Tracker. If there is too much chattering, one can switch to Augmented Lagrange method or use a command object to adjust chattering controls (FTOLN) for Normal Lagrange as necessary contact to fit the application.

May 9, 2017

07.06   Summary

Mechanical provides many tools in helping users to monitor nonlinear analyses as well as diagnose any problems.

- Oftentimes, it may be better to start simple and add complexity as you go, so that sources of problems can be isolated more readily. Adding lots of complexity to the first analysis can result in wasted time down the road.

- Do not randomly change settings. Use recommended settings first, then change contact or solver settings only if there is clear reason to do so, as illustrated in the Solver Output, Results Tracker, or Newton-Raphson Residuals.

- Although Contact Regions are automatically created, always verify all Contact/Target surfaces and contact settings to ensure that Contact Regions are defined as expected. Review the detailed contact output in the Solver Output to verify the initial contact status of Contact Regions and what value of the penetration or gap is, if present.

          May 9, 2017

07.07   Workshop 07.1: Diagnostic Tools for Contact

- **Please refer to instructions for *Workshop 07.1 - Diagnostic Tools for Contact***