

Contact Analysis Tips

Sheldon Imaoka

Memo Number: STI0902

ANSYS Release: 11.0

January 10, 2009

1 Introduction

Generally speaking, nonlinear analyses involving contact can be quite challenging to solve when the contacting area changes during the load history. However, ANSYS Workbench Simulation has very robust contact technology, along with diagnostic tools that can help the user obtain converged, accurate solutions. This memo hopes to discuss some tips related to contact analyses in hopes of aiding the user who encounters difficulties when solving contact analyses.

2 Background

In finite element analysis, if two independent parts are present, there is no stiffness relationship defined between them, and the resulting stiffness matrices will be *uncoupled* — consequently, one part may pass through the other during the course of the simulation. Contact elements are required to define the interaction of two or more sets of meshes to prevent such penetration.

ANSYS contact elements typically support four different algorithms: augmented Lagrangian, pure penalty, Multipoint constraint, and Lagrange multiplier methods. The default and most commonly-used option is the augmented Lagrangian formulation, which can be thought of as a variation of the pure penalty method.

2.1 Penalty-Based Methods

When two parts come into contact, ideally, no penetration will occur. The pure penalty method can be thought of as placing stiff springs between the two parts that have come into contact with each other:

$$p = k_n x_n \quad (1)$$

One can see from Equation 1 that as the contact stiffness k_n is increased, the resulting penetration x_n decreases for a finite amount of contact pressure p . Ideally, the penetration x_n should be zero, but Equation 1 would result in contact stiffness k_n being infinite. However, if the penetration x_n is small, the results are still very accurate.

The dilemma faced by the user is that choice of the contact stiffness k_n affects both accuracy and convergence.¹ Too low of a value of k_n results in large penetration x_n ; one can imagine that if the penetration is on the same order of magnitude as the calculated displacements, the results will be quite suspect. On the other hand, selection of k_n that is too high leads to convergence difficulties; the reason for this is because any variation of penetration Δx_n leads to a large change in contact pressure Δp .

The good news is that, by default, ANSYS automatically calculates contact stiffness k_n , based on the underlying solid element's size and material properties. The analyst is left with the task of monitoring the nonlinear solution — if convergence difficulties are met due to too high of an k_n value, ANSYS provides tools on determining which contact region's contact stiffness should be reduced. Likewise, postprocessing penetration x_n is available after a converged solution to ensure that k_n was not too low.

For experienced users wishing to manually change the default contact stiffness, in the Details view of a Contact Region, the “Advanced: Normal Stiffness” option should be changed to “Manual”, and the user may input a “Normal Stiffness Factor”.²

2.1.1 Augmented Langrangian Method

As noted earlier, the augmented Lagrangian method is a penalty-based approach. Specifically, Equation 1 is modified as follows during contact:

$$p = k_n x_n + \lambda \quad (2)$$

During the Newton-Raphson iterations, the contact penetration x_n is checked against an automatically-calculated *maximum allowable penetration tolerance* ϵ_n .³ If $x_n \geq \epsilon_n$, then λ is increased, and a message is printed in the

¹For surface-to-surface contact, the units of contact stiffness are *force/length*³

²In ANSYS, this is controlled via the 3rd real constant FKN, where a negative value represents an absolute contact stiffness whereas a positive value reflects a factor multiplied to the automatically-calculated stiffness.

³In ANSYS, this value can be manually specified via real constant 4 FTOLN

Solution Output indicating the number of contact points that have excessive penetration (lines 2 and 6 below):

```

1 LINE SEARCH PARAMETER = 1.000      SCALED MAX DOF INC = 0.2996E-01
2 3D CONTACT ELEMENTS: 640 CONTACT POINTS HAVE TOO MUCH PENETRATION
3 FORCE CONVERGENCE VALUE = 0.9108E+05 CRITERION= 3570.
4 EQUIL ITER 6 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= 0.2696E-01
5 LINE SEARCH PARAMETER = 1.000      SCALED MAX DOF INC = 0.2696E-01
6 3D CONTACT ELEMENTS: 416 CONTACT POINTS HAVE TOO MUCH PENETRATION
7 FORCE CONVERGENCE VALUE = 0.6736E+05 CRITERION= 4061.
8 EQUIL ITER 7 COMPLETED. NEW TRIANG MATRIX. MAX DOF INC= -0.2354E-01
9 LINE SEARCH PARAMETER = 1.000      SCALED MAX DOF INC = -0.2354E-01
10 FORCE CONVERGENCE VALUE = 0.6458E-03 CRITERION= 4405. <<< CONVERGED
11 >>> SOLUTION CONVERGED AFTER EQUILIBRIUM ITERATION 7

```

The benefit of the augmented Lagrangian method is that the results are less sensitive to the value of contact stiffness k_n . To understand why, consider the example below of a finite contact pressure 10 with contact stiffness 1000:

$$p = k_n x_n + \lambda \quad (3)$$

$$10 = 1000(0.01) \quad (4)$$

$$= 1000(x_n) + 5 \quad \text{if } x_n \leq \epsilon_n \quad \text{define } \lambda = 5 \quad (5)$$

$$= 1000(0.005) + 5 \quad (6)$$

From Equation 6, the use of λ results in lower penetration if the penetration is found to be greater than the maximum allowable value. This reduction in penetration without the need to increase k_n makes the augmented Lagrangian approach an attractive alternative to the pure penalty method, although it should be noted that more equilibrium iterations may be required as a result of this penetration control.

In Workbench Simulation, in the Details view of a Contact Region, the “Advanced: Formulation” field allows the user to change this option, as “Pure Penalty” is the default.⁴ For Workbench Simulation users wishing to change the default contact formulation to “Augmented Lagrange” for all newly-created models, this can be accomplished via the “Tools menu → Options ...: Simulation → Connections → Default” field.

⁴In ANSYS, the augmented Lagrangian algorithm is the default method, but in Workbench Simulation, pure penalty is the default.

2.1.2 Automatic Contact Stiffness Update

During the course of the analysis, material response or contacting areas may change, so it may be desirable to modify the contact stiffness k_n as the solution progresses. While the user may manually redefine the contact stiffness between load steps, an automatic adjustment option is available. This updating procedure is based on heuristics, and the user can choose whether updating is done per substep or per equilibrium iteration.

In Workbench Simulation, in the Details view of a Contact Region, there is a pull-down menu in “Advanced: Update Stiffness” with relevant options.⁵

2.2 Friction and Elastic Slip

When friction is present, an analogous situation exists for behavior in the tangential direction. Until the frictional shear stress τ exceeds the limiting shear stress τ_{lim} , the contact points should be “sticking”. While zero slip is desired, in a penalty-based method, the slip x_t is related to the frictional stress τ by the tangential contact stiffness k_t as follows:

$$\tau = k_t x_t \quad \text{if } \tau < \mu p \quad (7)$$

Similar to the case in the normal direction with penetration x_n , if the elastic slip x_t is small, it will not compromise accuracy in frictional models.

In ANSYS, the tangential contact stiffness k_t is automatically calculated but can be manually specified via the 12th real constant FKT — in Workbench Simulation, a user would insert a “Commands” object underneath the relevant contact region to accomplish this setting. When KEYOPT(10) is set to a value of 2 or 3, the maximum allowable slip SLTO (defined by the 23rd real constant) is also used to update FKT to reduce elastic slip.

Understanding how frictional models are implemented with a penalty-based approach is useful, although the adjustment of tangential contact stiffness k_t and use of maximum allowable slip SLTO is usually not needed for most problems.

⁵In ANSYS, this option is controlled via KEYOPT(10). It is worth noting that in ANSYS, a more aggressive updating scheme can be activated through the addition of KEYOPT(6).

3 Tip: Initial Contact Information

Rigid-body motion due to parts not initially in contact is a common convergence problem. Prior to solving, the user should always check the initial contact status to determine whether or not parts that are thought to be in initial contact truly are touching. While the calculation of initial contact status may take a few seconds or minutes, depending on the size of the model, it will lead to time savings when detecting problems early.⁶

In Workbench Simulation, a user may insert a “Contact Tool” underneath the “Connections” branch, as shown in Figure 1. The user may select or deselect specific contact regions, and plotting/listing of only the contact or target side is possible from this worksheet. Multiple “Contact Tool” branches may also be inserted to allow the user to review contact regions in different groups.

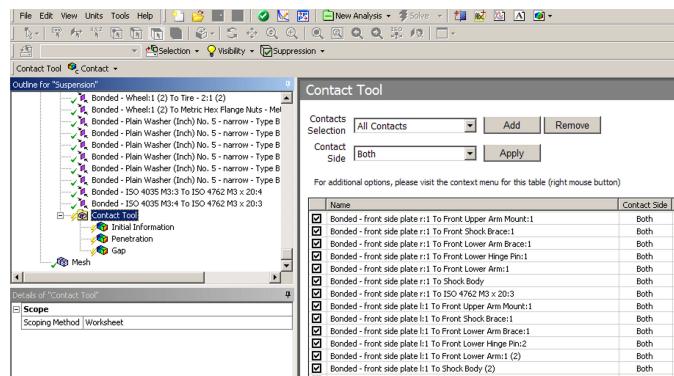


Figure 1: Connections branch with Contact Tool

The “Initial Information” branch is included by default, although a user may also insert contour results of initial “Status,” initial “Penetration,” or initial “Gap” as well. If the user right-clicks on “Contact Tool” and selects “Generate Initial Contact Results”, the initial contact information will be calculated and presented in tabular form, as shown in Figure 2. The rows are conveniently summarized where possible problems are highlighted in orange (possibly large penetration or gap), yellow (frictionless or frictional contact pair having an initially open state), or red (bonded or no-separation contact initially having an open state). This allows for more convenient

⁶This discussion centers on use of ANSYS Workbench Simulation, but for ANSYS users, refer to the CNCHECK command for similar functionality that is discussed in this section.

examination of models with very large number of contact regions, as shown in this example.

Initial Information							
For additional options, please visit the context menu for this table (right mouse button)							
Name	Contact Side	Type	Penetration (mm)	Gap (mm)	Geometric Penet...	Geometric Gap (...	Resulting Pinball (mm)
Bonded - Shock Shaft:1 To Piston:1	Contact	Bonded	6.4837e-015	0.	7.1214e-004	0.2033	
Bonded - Shock Shaft:1 To Piston:1	Target	Bonded	5.0466e-015	0.	5.5755e-004	7.1058e-004	0.1788
Bonded - Shock Shaft:1 To Ball End:1	Contact	Bonded	2.1056e-015	0.	9.0913e-002	6.2656e-002	0.1504
Bonded - Shock Shaft:1 To Ball End:1	Target	Bonded	1.0282e-014	0.	8.3755e-002	5.6963e-002	0.1361
Bonded - Shock Shaft:1 To Spring Cup:1	Contact	Bonded	9.1238e-015	0.	0.10217	0.25344	
Bonded - Shock Shaft:1 To Spring Cup:1	Target	Bonded	8.9593e-015	0.		7.2879e-002	7.3943e-002
Bonded - E-Clip:1 To Piston:1	Contact	Bonded	9.0623e-015	0.	8.6165e-009	6.7117e-009	9.1652e-002
Bonded - E-Clip:1 To Piston:1	Target	Bonded	9.0623e-015	0.	8.1576e-009	8.4217e-009	0.19104
Bonded - E-Clip:2 To Piston:1	Contact	Bonded	1.4854e-014	0.	7.8736e-009	7.8119e-009	9.525e-002
Bonded - E-Clip:2 To Piston:1	Target	Bonded	4.9514e-015	0.	6.4577e-009	6.8869e-009	0.20406
Bonded - Ball End:1 To 0.25 Ball:1	Contact	Bonded	9.3212e-015	0.	1.0000e-002	1.0000e-002	0.11205
Bonded - Ball End:1 To 0.25 Ball:1	Target	Bonded	9.3212e-015	0.	2.8342e-002	9.3801e-002	0.4597
Bonded - Ball End:1 To Spring Cup:1	Contact	Bonded	1.9806e-014	0.	1.9635e-003	0.20737	0.27899
Bonded - Ball End:1 To Spring Cup:1	Target	Bonded	1.9806e-014	0.	1.7406e-003	0.11672	0.12109
Frictionless - Ball Body:1 To Plain Washer (Dish):1	Contact	Bonded	0.	0.	0.	0.	0.01073
Frictionless - Ball Body:1 To Plain Washer (Dish):1	Target	Bonded	0.	0.	0.	0.	0.00001
Bonded - 0.25 Ball:1 To ISO 4762 M3 x 12 12	Contact	Bonded	1.4949e-014	0.	2.6615e-002	8.9457e-002	0.3079
Bonded - 0.25 Ball:1 To ISO 4762 M3 x 12 12	Target	Bonded	7.5064e-014	0.	0.10051	0.10945	0.11615
Frictionless - Shock Spring Adjust Nut:1 To 1C1	Contact	Frictionless	2.2712e-003	0.	2.2712e-003	N/A	0.9505
Frictionless - Shock Body (1) To Shock Cartridge ...	Contact	Frictionless	7.6203e-005	0.	7.6203e-005	N/A	0.00001
Frictionless - Shock Body (2) To Shock Cartridge ...	Contact	Frictionless	0.4643	0.	0.4643	N/A	4.219
Frictionless - Shock Body (2) To Shock Cartridge ...	Target	Frictionless	0.73535	0.	0.73535	N/A	0.73716
Frictionless - Shock Body (2) To Shock Cartridge ...	Contact	Frictionless	0.	0.10171	0.	N/A	2.7907
Frictionless - Shock Body (2) To Shock Cartridge ...	Target	Frictionless	0.2685	0.	0.2685	N/A	0.0002
Bonded - Shock Body (1) To Piston:1 (2)	Contact	Bonded	1.9823e-014	0.	0.26625	0.23984	0.30927
Bonded - Shock Body (1) To Piston:1 (2)	Target	Bonded	1.5784e-013	0.	0.23564	0.2403	0.24046
Bonded - Shock Body (2) To Plain Washer (Dish):1	Contact	Bonded	0.	0.	0.	0.	0.23976
Bonded - Shock Body (2) To Plain Washer (Dish):1	Target	Bonded	0.	0.	0.	0.	0.00001
Bonded - Shock Cartridge:1 (2) To O-Ring:1 (2)	Contact	Bonded	1.9806e-014	0.	0.31462	0.43157	
Bonded - Shock Cartridge:1 (2) To O-Ring:1 (2)	Target	Bonded	1.3394e-014	0.	1.9993e-003	0.17087	0.17457

Figure 2: Initial Contact Information

The user should not only review the highlighted rows, but the user should also check to see how much penetration or gap may be present, along with the number of contacting points. Experienced analysts using auto-asymmetric behavior can see which side has been deactivated (grey), and the verification of contact offset or automatic adjustment is possible. Columns can be hidden or shown by right-clicking in the worksheet.

Contour plots of initial status, penetration, and gap are also quite helpful. For example, in Figure 3, the user can see that the cylinder is only in “near contact” with the bar, so either automatic adjustment or a finer mesh is required to establish initial contact.

4 Tip: Contact Result Tracker

Nonlinear solutions of large models may take considerable CPU time. There may be cases where a user may invest a lot of time solving a model only to find that incorrect model setup or unanticipated contacting areas leads to an invalid solution. Other scenarios may involve the solution bisecting or progressing slowly, and the user may not be able to check results without prematurely stopping the solution.

Being able to track results can help in the above situations. Prior to solving, the user can request certain results for specific contact regions and monitor these results during the course of the analysis. If the contact solution starts to deviate from the expected behavior, the user can stop the

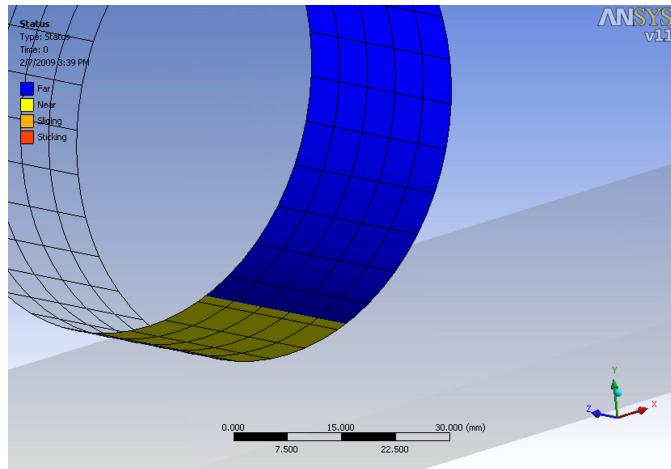


Figure 3: Contour of Initial Contact Status

analysis without having to wait until the end of the run to find out that the analysis setup may not be correct.⁷

Prior to solving, the user may highlight the “Solution Information” branch and select the “Result Tracker”. The ability to monitor deformation at vertices or overall kinetic and potential energy are possible. Also, checking contact results is helpful in monitoring nonlinear simulations.

While the user can select the “Results Tracker → Contact”, the author prefers to drag-and-drop contact regions of interest from the “Connections” branch to the “Solution Information” branch. In the Details view of the “Result Tracker” that appears, the user may select any of the following items for a given contact region:

- Maximum contact pressure
- Maximum contact penetration
- Maximum contact gap
- Maximum frictional stress (frictional contact only)
- Maximum sliding distance (frictional contact only)
- Number of contact elements in a sticking state

⁷Discussion is focused on Workbench Simulation. For ANSYS users, refer to the NLHIST command for equivalent functionality.

- Number of contact elements in contact (sticking or sliding)
- Maximum chattering level
- Maximum elastic slip (in the sliding direction)
- Maximum normal contact stiffness
- Maximum tangential contact stiffness
- Minimum tangential contact stiffness

The user may add as many “Result Tracker” items as desired.

For example, Figure 4 shows the number of contacting elements for seven contact regions while the nonlinear solution is progressing. One can see that the contact region “Frictional - seal3” is in near-field (open) contact throughout the solution. On the other hand, the contact region “Frictional - opening” was open until a time of 0.4, where a large number of elements came into contact. This helps the user understand whether each contact region is increasing or decreasing in the contacting area, and if the behavior is unexpected, the solution may be stopped to examine the intermediate results.

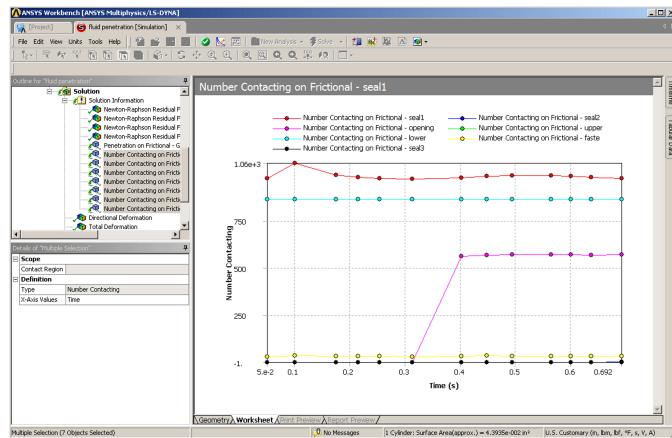


Figure 4: The Results Tracker During Solution

5 Tip: Nonlinear Diagnostics

As alluded to earlier, the contact stiffness k_n is the most important contact parameter for the penalty-based approach, influencing both convergence behavior and accuracy. During the equilibrium iterations, if the force residuals “plateau”, an example of which is shown in Figure 5, chances are high that contact stiffness is preventing force convergence from being achieved. While contact stiffness *may* be a cause for the high residuals, the user may not be certain by simply looking at the force convergence behavior.

During the Newton-Raphson iteration, convergence is achieved when force equilibrium is satisfied. The user can request ANSYS to save the Newton-Raphson residuals, so regions of high out-of-balance forces can be reviewed. This helps the user in determining where force imbalance is high — if the area is associated with a contact region, then the user will have an understanding of which contact regions may have too high of a contact stiffness defined. Of course, if the high residuals are in an area not associated with contact but loads or boundary conditions, the user knows to pay closer attention to these constraints instead of worrying about contact.⁸

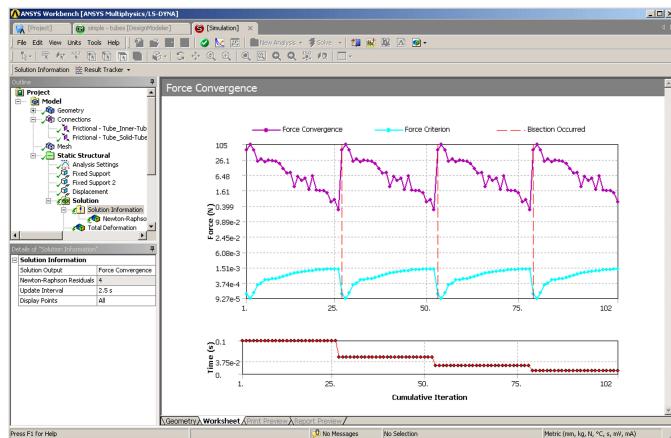


Figure 5: Force Convergence Graph

In Workbench Simulation, prior to initiating a solution, the user should select the “Solution Information” branch. In the Details view, a value of “4” or “5” can be entered for the “Newton-Raphson Residuals”. This tells ANSYS to save the last 4 or 5 N-R residuals in the event of non-convergence or

⁸While usage in Workbench Simulation is covered here, for ANSYS users, refer to the **NLDIAG** command this and additional functionality.

the user stopping the solution. In cases of an incomplete solution, contours of Newton-Raphson residuals will be available under the “Solution Information” branch, and the user can plot the contour as shown in Figure 6. (Note that for converged solutions, the Newton-Raphson residuals are not needed and hence hidden from view.) By examining the N-R residuals, the user can determine what regions have high out-of-balance forces for cases of non-convergence due to force equilibrium not being satisfied. In this example, a solid cylinder is pushing down on two hollow cylinders, where half of the model is displayed. The highest residuals are in-between the two concentric hollow cylinders, indicating that the contact stiffness defined for that region may be too high. Because the full Newton-Raphson method regenerates the stiffness matrix at each iteration, the author prefers reviewing the last 4 or 5 N-R residuals to ensure that the problematic region is consistent (i.e., if 3 of the 4 N-R residuals indicate the same area, that gives the user confidence that these nodes are related to force convergence difficulties).

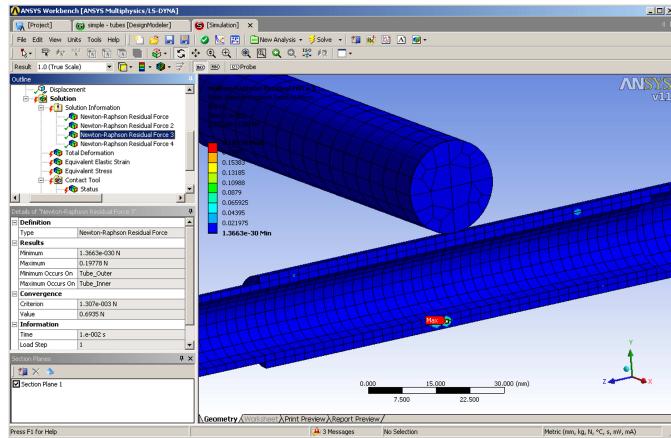


Figure 6: Contour Plot of Newton-Raphson Residuals

6 Tip: Contact Postprocessing

Postprocessing is the most important step of any analysis, and contact problems are no different. The user should always review contour plots of contact status, pressure, and penetration in order to verify that the mesh adequately captures the contact behavior and that results are correct.

Contact penetration is in units of length. One can compare deformation in the same direction as contact; if the penetration is a small fraction of the

deformation, one may conclude that any variation in penetration would not affect results significantly.

In Figure 7, the maximum penetration is 4.256e-3 mm. The user can compare this value to the deformation on the same contact surface to verify that the penetration is negligible. Checking the contact status may indicate that contact detection is occurring at a very localized region, which may warrant a finer mesh.

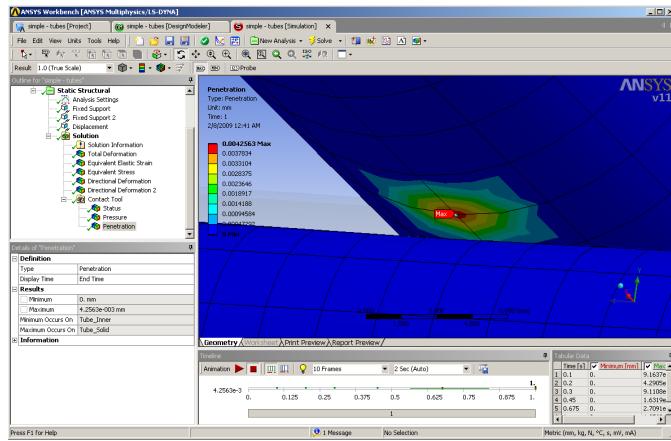


Figure 7: Contour Plot of Contact Penetration

7 Conclusion

This memo attempted to cover four tips that can aid the analyst in solving complex contact problems:

- Check the initial contact status to ensure that contact regions that should be engaged are in initial contact
- Monitor results of interest during the nonlinear solution
- Specify storage of Newton-Raphson residuals in case of non-convergence
- Verify results by plotting contact pressure distributions and, most importantly, contact penetration

Obtaining convergence is but one important aspect to contact analyses, as the analyst should also remember to verify the results.

Sheldon's ansys.net Tips and Tricks

Sheldon's ansys.net Tips and Tricks are available at the following URL:

http://ansys.net/sheldon_tips/

Please remember that, with each release of ANSYS, new features and techniques may be introduced, so please refer to the ANSYS documentation as well as your local ANSYS support office to verify that these tips are the most up-to-date method of performing tasks.

Disclaimer: the author has made attempts to ensure that the information contained in this memo is accurate. However, the author assumes no liability for any use (or misuse) of the information presented in this document or accompanying files. Please refer to ansys.net for the latest version of this document. Also, this memo and any accompanying input files are not official ANSYS, Inc. documentation.

ANSYS Training

ANSYS, Inc. as well as ANSYS Channel Partners provide training classes for ANSYS, Workbench, CFX, FLUENT, ANSYS LS-DYNA, AUTODYN, ASAS, AQWA, TAS, and ICEM CFD products. Information on training classes and schedules can be found on the following page:

<http://www.ansys.com/services/ts-courses.asp>

ANSYS Customer Portal

Customers on active maintenance (TECS) can register for a user account and access the ANSYS Customer Portal. Here, browsing documentation, downloading software (including service packs), and submitting technical support incidents are possible: <http://www1.ansys.com/customer/>

XANSYS Mailing List

The XANSYS mailing list is a forum for exchanging ideas, providing and receiving assistance from other users, and general discussions related to ANSYS and Workbench. (Note that it is recommended to contact your local ANSYS support office for technical support.) You can obtain more information by visiting the following URL: <http://www.xansys.org/>