Collaborative Solutions Inc

An ANSYS Support Distributor

Date	June 10, 2000	Memo Number	STI51:000610
Subject	**ANSYS Tips & Tricks: Nonlinear Monitoring**		
Keywords	Nonlinear: SOLCON: MONITOR		

1. Introduction:

ANSYS 5.4 through 5.6 utilizes Solution Control (SOLCON), which has many default values to provide some intelligence to nonlinear analyses. Besides setting more robust nonlinear options, SOLCON also has an analysis monitoring feature which the author finds quite useful.

This memo will briefly discuss Solution Control and provide some information on using the Monitor capability. It is assumed that the reader is already familiar with performing nonlinear structural and/or thermal analyses in ANSYS. CSI also provides training and mentoring services in this area to help engineers solve these classes of problems.

2. Background on Solution Control:

Solution Control, introduced around v5.4, provides default settings on various ANSYS nonlinear commands such as NROPT, CNVTOL, LNSRCH, PRED, EQSLV, etc. At 5.6, Solution Control (SOLCON) is on by default. It can be deactivated via the /CONFIG,NLCONTROL,0 or with SOLCON,OFF commands.

Because Solution Control is automatically "on," the user should not need to issue SOLCON,ON. Please note that issuing this command will *reset* all nonlinear solution options to the "default" values as provided by SOLCON, so any changes to solution options made prior to issuing SOLCON,ON will be ignored.[1]

Between revisions 5.4 and 5.6, the behavior and defaults for SOLCON have changed. With each revision of ANSYS, "smarter" and more robust values of SOLCON are included, so the user should keep this in mind when running the same nonlinear problem in different versions of ANSYS.

Solution Control also provides the MONITOR functionality which is otherwise unavailable in ANSYS with SOLCON,OFF. Because of this and the fact that SOLCON provides much more robust solution options, the author strongly recommends leaving SOLCON,ON (default). The user can refer to the online help for SOLCONTROL for information on what default command settings are and change any options, as needed.[2]

3. Discussion on MONITOR Capability:

During a solution of a structural or thermal nonlinear problem, a file called "jobname.mntr" will be created in the working directory. This is the monitor file which provides the status/progress of a nonlinear analysis, posting information on converged substeps/loadsteps. An example is shown below:

```
SOLUTION HISTORY INFORMATION FOR JOB: preload1.mntr
 ANSYS RELEASE  5.6             .1      20:36:31    04/30/2000

 LOAD SUB-  NO.   NO.  TOTL   INCREMENT    TOTAL       VARIAB 1    VARIAB 2    VARIAB 3
 STEP STEP  ATTMP ITER ITER   TIME/LFACT   TIME/LFACT  MONITOR     MONITOR     MONITOR

   1    1     1     5     5    1.0000       1.0000      18.517     0.45752E-02 0.23241E-02
   2    1     1     1     6    1.0000       2.0000      23.974     0.45752E-02 0.17538E-08
   3    1     1     3     9    838.83       840.83      35.661     0.45752E-02 0.78886E-30
   3    2     1     2    11    838.83       1679.7      43.863     0.45752E-02 0.78886E-30
   3    3     1     2    13    838.83       2518.5      52.155     0.45752E-02 0.78886E-30
   3    4     1     2    15    838.83       3357.3      60.177     0.45752E-02 0.78886E-30
   3    5     1     1    16    838.83       4196.1      65.254     0.45752E-02 0.78886E-30
   3    6     1     2    18    838.83       5035.0      73.406     0.45752E-02 0.87334E-05
   3    7     1     1    19    838.83       5873.8      78.343     0.45752E-02 0.52087E-04
   3    8     1     1    20    838.83       6712.6      83.410     0.45752E-02 0.90495E-04
   3    9     1     1    21    838.83       7551.4      88.227     -.46022E-02 0.10128E-03
   3   10     1     1    22    838.82       8390.3      93.144     -.50003E-02 0.13811E-03
```

[1] Issuing SOLCON,ON is usually performed to reset solution options to the default settings.
[2] As an example, the author usually does not like using line search (LNSRCH) because it is computationally expensive, so he turns it off since, by default, line search is activated by SOLCON for all contact problems.

2303 W. 190th St • Redondo Beach, CA 90278
310.896.1230 • FAX 310.896.1240 • http://www.csi-ansys.com

As noted in the example listing above, the far-right side of the monitor file lists the solved loadstep and substep numbers. This helps a user understand where the solution currently is, especially in the event that there are multiple load steps in a transient or static nonlinear analysis.

The third column lists the number of attempts made to solve the particular substep. When the solution does not converge or if ANSYS predicts that it will not converge with the maximum number of equilibrium iterations (NEQIT), automatic timestepping (AUTOTS) bisects the substep and tries again. If this number is greater than 1, it will give the user a good indication of the point at which the solution is having difficulty converging.

Likewise, the number of equilibrium iterations, as listed in the fourth column, also gives an indication of whether ANSYS is having an easy or difficult time solving the problem. If the number of equilibrium iterations is low (anywhere from 2-5), then the user may not need as small a timestep/substep to solve the problem.[3] On the other hand, if the number of equilibrium iterations needed to solve the substep is high (anywhere from 15-20+), the user should consider starting off with small substeps. If ANSYS is bisecting the problem a lot, this, too, gives the user an indication that the initial as well as minimum substep may be too large.

The sixth and seventh columns provide substep (a.k.a timestep) increments and totals.[4] This helps determine how large of a substep ANSYS is using. Also, if ANSYS has bisected or decreased/increased the substep, this is indicated in the sixth column. This information serves several purposes: (a) if the substep is too small, this may tell the user that the problem is having trouble converging, and he/she may want to abort the run cleanly at this point to examine the model or (b) it gives the current status of the analysis [i.e., How much of the problem has already been solved? Is ANSYS 75% done?], letting the user know when to expect completion of the problem.

The last three columns provide the user with customizable information. By default, the three columns list elapsed CPU solution time, maximum displacement of the model, and maximum equivalent plastic strain of the model. For thermal analyses, "MxDs" actually reports max temperature of the entire model, and column 3 is unused.

The CPU time provides the user with information regarding (a) how long ANSYS has been solving the problem and (b) about how long it may take in the future (looking at the TIME column in conjunction with the CPU column allows the user to estimate how long the total solution may take if it continues at the same rate). The maximum DOF value also helps the user understand how the system is responding, especially for large deflection problems.

The last three columns can also be customized through the use of the MONITOR command. The MONITOR command allows the user to report DOF values or reaction forces for any node, up to three nodes. The author has found this capability invaluable for many types of analyses. While the solution is running, the author can "tail" or "cat" the monitor text file to see how the system is responding. Examples of what nodes to "monitor" are the following:

- For large deflection/rotation problems, select a node at the "tip" of the structure to see how much the system has moved.
- For contact problems involving multiple bodies coming into contact, monitoring both the moving body and stationary body can help determine at what time the bodies come into contact. This also helps determine if the load increments are too high (or pinball region is too small), in the event that one body "passes through" the other and contact is not detected. This way, the user does not need to invest the whole solution time to find out that the load increment is too high – after reviewing the monitor file, the user can abort the solution and specify smaller substeps or larger pinball radius.
- For bolt preload problems (PRETS179), the reaction force (i.e., bolt preload) after fixing the bolt shrinkage can be monitored to ensure that the preload is not exceeded during the course of the analysis.
- For rigid-flexible contact (CONTA169-174) using a pilot node, the reaction force at the pilot node can be monitored to determine the contact force applied. Conversely, for force-driven problems, monitoring the DOF of the pilot node helps determine how much the rigid surface is moving.

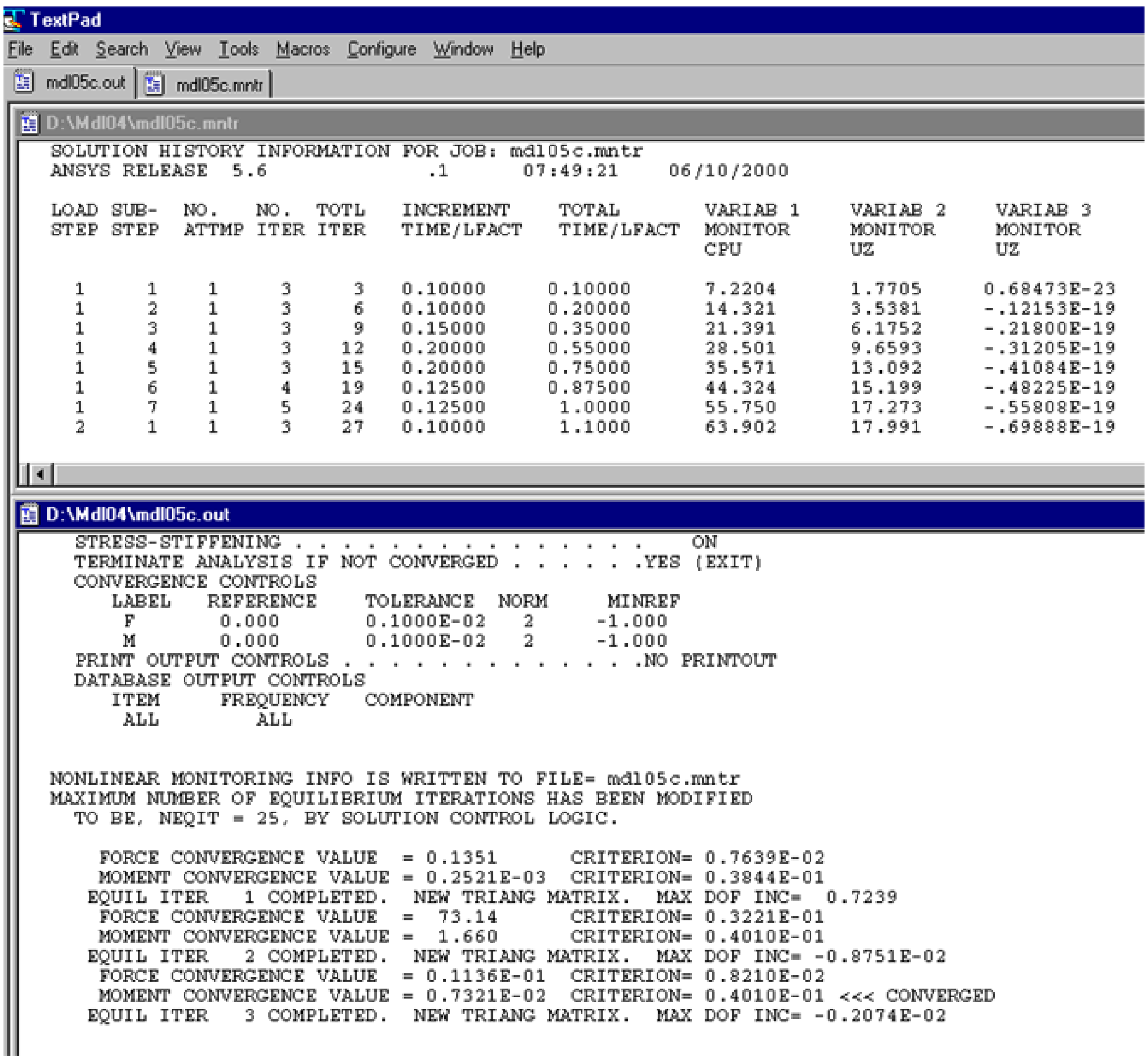
[3] This statement is, of course, assuming that the problem is not path-dependent. Any path-dependent problem (e.g., plasticity, friction) should have enough substeps to capture the proper response history.

[4] For simplicity, the author will only refer to substeps rather than timesteps throughout this memo, but these quantities are inversely related.

- For thermal problems, the temperature at critical regions can be reported. For example, in the analysis of electronic components, the junction temperatures can be monitored during a nonlinear transient run. If the temperatures are too high or too low than what is expected, the solution can be halted to verify tabular boundary conditions or any other possible problems.

These are a few examples of quantities of interest the user can monitor during the course of the analysis.

While the user can view the monitor file periodically with "tail" or "cat" in the DOS or xterm window,[5] the author has found using text editors such as TextPad quite helpful:



```
TextPad
File Edit Search View Tools Macros Configure Window Help
mdl05c.out | mdl05c.mntr

D:\Mdl04\mdl05c.mntr
 SOLUTION HISTORY INFORMATION FOR JOB: mdl05c.mntr
 ANSYS RELEASE  5.6              .1       07:49:21      06/10/2000

 LOAD SUB-  NO.   NO.  TOTL  INCREMENT     TOTAL      VARIAB 1    VARIAB 2    VARIAB 3
 STEP STEP  ATTMP ITER ITER  TIME/LFACT   TIME/LFACT  MONITOR     MONITOR     MONITOR
                                                      CPU         UZ          UZ

   1    1     1     3    3   0.10000     0.10000      7.2204      1.7705     0.68473E-23
   1    2     1     3    6   0.10000     0.20000      14.321      3.5381     -.12153E-19
   1    3     1     3    9   0.15000     0.35000      21.391      6.1752     -.21800E-19
   1    4     1     3   12   0.20000     0.55000      28.501      9.6593     -.31205E-19
   1    5     1     3   15   0.20000     0.75000      35.571      13.092     -.41084E-19
   1    6     1     4   19   0.12500     0.87500      44.324      15.199     -.48225E-19
   1    7     1     5   24   0.12500      1.0000      55.750      17.273     -.55808E-19
   2    1     1     3   27   0.10000      1.1000      63.902      17.991     -.69888E-19

D:\Mdl04\mdl05c.out
   STRESS-STIFFENING . . . . . . . . . . . . . . .    ON
   TERMINATE ANALYSIS IF NOT CONVERGED . . . . . .YES (EXIT)
   CONVERGENCE CONTROLS
      LABEL   REFERENCE    TOLERANCE  NORM     MINREF
       F        0.000      0.1000E-02   2      -1.000
       M        0.000      0.1000E-02   2      -1.000
   PRINT OUTPUT CONTROLS . . . . . . . . . . . . .NO PRINTOUT
   DATABASE OUTPUT CONTROLS
      ITEM      FREQUENCY    COMPONENT
       ALL         ALL


 NONLINEAR MONITORING INFO IS WRITTEN TO FILE= mdl05c.mntr
 MAXIMUM NUMBER OF EQUILIBRIUM ITERATIONS HAS BEEN MODIFIED
   TO BE, NEQIT = 25, BY SOLUTION CONTROL LOGIC.

     FORCE CONVERGENCE VALUE   = 0.1351      CRITERION= 0.7639E-02
     MOMENT CONVERGENCE VALUE  = 0.2521E-03  CRITERION= 0.3844E-01
    EQUIL ITER   1 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC=  0.7239
     FORCE CONVERGENCE VALUE   =  73.14      CRITERION= 0.3221E-01
     MOMENT CONVERGENCE VALUE  =  1.660      CRITERION= 0.4010E-01
    EQUIL ITER   2 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.8751E-02
     FORCE CONVERGENCE VALUE   = 0.1136E-01  CRITERION= 0.8210E-02
     MOMENT CONVERGENCE VALUE  = 0.7321E-02  CRITERION= 0.4010E-01 <<< CONVERGED
    EQUIL ITER   3 COMPLETED.  NEW TRIANG MATRIX.  MAX DOF INC= -0.2074E-02
```

A sample analysis session is shown in the figure above. TextPad (and some other text editors) can be configured to "reload" any text file. As soon as ANSYS updates/writes to the jobname.mntr and/or jobname.out file, TextPad can automatically reload the new, updated information. As a result, the author views the monitor file on the top to examine the status of the entire solution and the output file at the bottom to review progress of the current equilibrium iteration. Also, note that the last three columns provide CPU and UZ information at two nodes. In this particular example, two bodies are contacting one another. Upon reviewing the last two columns, it is apparent that they

[5] There are freeware versions of "tail" and "cat" available for Windows/DOS. UNIX systems already have these utilities, usually located in /usr/bin.

have not come into contact yet (which explains why ANSYS is having little trouble solving the problem with 3-5 equilibrium iterations per substep).[6]

4. Conclusion/Recommendations:

The MONITOR capability of SOLCON is quite useful to help determine how well a solution may be progressing. Instead of investing time in the whole solution run, potential problems can be determined more quickly and the analysis halted, if necessary.[7] This saves a lot of time, especially during the first stages of setting up a nonlinear problem where one may not be sure of how the structure is responding or may have incorrectly posed the problem. Also, the monitor file provides concise status/summary of the analysis and is easier to read than parsing through the output file which usually contains a large amount of information.

While this memo cannot cover all of the specifics of SOLCON, the author hopes that the MONITOR command may prove useful to engineers currently performing structural and thermal nonlinear analyses. For users unfamiliar with nonlinear analyses, the author strongly recommends taking training and/or mentoring sessions from CSI engineers to better familiarize themselves with all of the various nonlinear solution options and powerful capabilities of ANSYS.

Sheldon Imaoka
Collaborative Solutions, Inc. (LA Office)
Engineering Consultant

[6] For this specific example, the two bodies are separated by a distance of 20 units, and the solution has progressed to the point that one body has moved 18 units. Hence, contact of the bodies is expected to occur shortly.

[7] To cleanly stop a solution in batch mode, create a text file in the working directory called "jobname.abt" with the word "nonlinear" in the first line. A simple way to accomplish this in DOS or xterm is with the command:

```
echo nonlinear > jobname.abt
```

where "jobname" is the actual jobname of the model.

ANSYS Tip of the Week

"ANSYS Tip of the Week" (TOTW) is provided for customers of Collaborative Solutions, Inc. (CSI) with active TECS agreements, distributed weekly in Adobe Acrobat PDF format via email. Unless otherwise stated, information contained herein should be applicable to ANSYS 5.4 and above, although usage of the latest version (5.6 as of this writing) is assumed. Users who wish to subscribe/unsubscribe or to view older TOTW archives can visit

`http://www.csi-ansys.com/tip_of_the_week.htm`

Corrections, comments, and suggestions are welcome and can be sent to operator@csi-ansys.com [they will be distributed to the appropriate person(s)]. While CSI engineers base their TOTW on technical support calls and user questions, ideas on future topics are appreciated. Users who wish to submit their own TOTW are encouraged to do so by emailing the above address for more information.

XANSYS Mailing List

The XANSYS mailing list is a forum for questions and discussions of the use of ANSYS. As of 04/00, there are more than 1000 subscribers with topics ranging from Structural, Thermal, Flotran, to Emag analyses, to name a few. Users are encouraged to subscribe to evaluate the usefulness of the mailing list for themselves. Also, either (a) using the mail program to filter [xansys] messages or (b) using the "digest" option to receive one combined email a day is strongly recommended to minimize sorting through the volume of postings.

```
This list is for *ALL* users of the ANSYS finite element analysis program from around the
world. The list allows rapid communication among users concerning program
bugs/ideas/modeling techniques. This list is NOT affiliated with ANSYS, Inc. even though
several members of the ANSYS, Inc. staff are subscribers and regular contributors.

To SUBSCRIBE: send blank email to xansys-subscribe@onelist.com
To unsubscribe send blank email to xansys-unsubscribe@onelist.com
Archived on http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html
ANOTHER archive on http://www.eScribe.com/software/xansys/
(A poor archive is also at http://www.onelist.com/archives.cgi/xansys)
```

CSI ANSYS Technical Support, Training, & Mentoring

Collaborative Solutions, Inc. is committed to providing the best customer support in our industry. Three people will be devoted to technical support from 8:00 a.m. to 5:00 p.m. PST every working day. CSI customers with active TECS (maintenance) agreements may contact CSI by any of the following ways:

```
Phone: 760-431-4815                                WWW: http://www.csi-ansys.com
Fax:   760-431-4824                                FTP:  ftp://ftp.csi-ansys.com
E-mail: firstname.lastname@csi-ansys.com

CSI Engineers:
Karen Dhuyvetter       Greg Miller       Sean Harvey        Alfred Saad
Bill Bulat             Sheldon Imaoka    David Haberman     Mike Rife
```

CSI believes strongly in the value of training and mentoring to help make customers successful using ANSYS. Training classes are usually 2-3 days in duration and provide instruction on various topics, including structural nonlinearities, heat transfer, and dynamics. Mentoring sessions involve working with a CSI engineer one-on-one on specific projects. These sessions help reinforce applicable subject matter covered in training classes or help ensure that the customer is using ANSYS most efficiently and effectively.

Training class schedules are posted at: `http://www.csi-ansys.com/training.htm`

Please contact your account manager for more details on training, mentoring, consulting, and other CSI services.