

Date July 12, 2000 Memo Number SMH01:000711
Subject **ANSYS Tip of the Week: New Multiframe Restarts in 5.6**
Keywords Restart: Multiframe: rescontrol: antype:

1. Introduction

At ANSYS 5.6 a new restart capability has been added called multiframe restart. This restart capability archives all the required files to restart an analysis at user control points within the solution. When used effectively, this capability can be very useful, especially on models that have long solution times. For example, a user has a non-linear contact analysis that takes several days to run. If the license server goes down, or power is lost, the user will be able to easily pick up from any of the restart files written out. Since the original way of doing restarts (esav, emat files) only has one set of saved restart files, an unexpected termination of ANSYS would likely result in these files being unusable. Since the new restart capability allows as many files as desired written out per load step, a user can pick up an analysis from nearly any time in the solution as desired.

2. Limitations

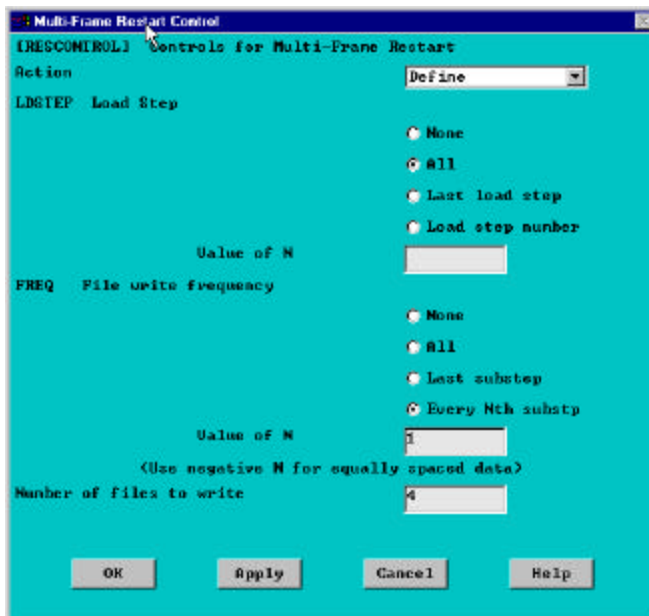
- Only available at 5.6 and up.
- Available only for nonlinear static or full transient structural analyses. Not available for thermal analyses.
- Parameters defined after the initial solve need special treatment (discussed later).

3. Commands and Usage

- Rescontrol – To define, list restart controls
- Antype – To specify that the solution is to be a restart

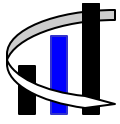
Examples of rescontrol syntax:

- **rescontrol,define,all,1,4** – For **all** load steps, every **1** sub step, write a restart, but only allow a maximum of **4** files to be written for each load step. This leaves the user with 4 restart files for the last 4 sub steps in the load step.

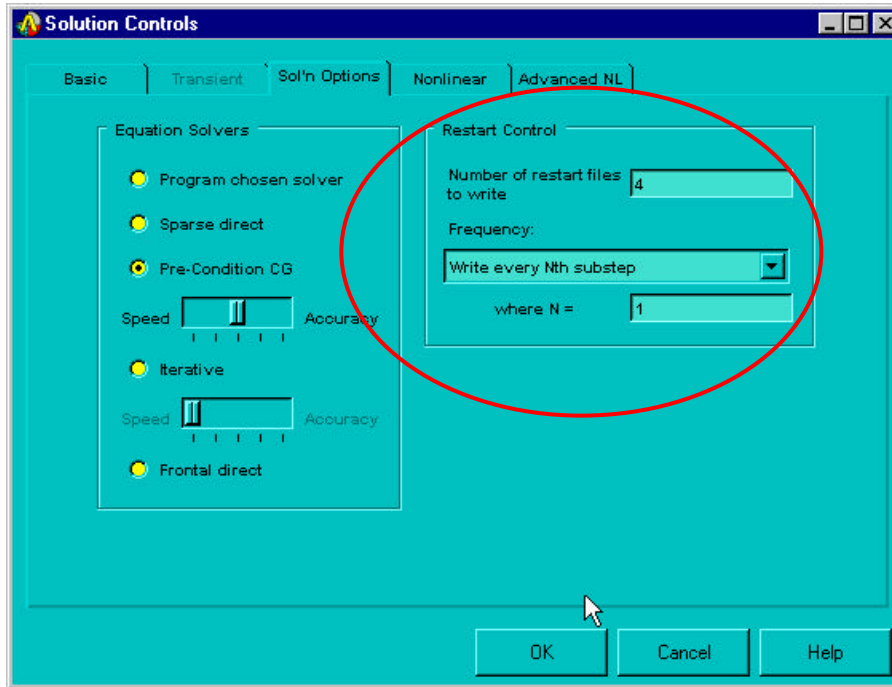


Using the Unabridged Menu

Solution>Nonlinear>Restart Control



or



Using the Abridged Menu

Solution>Sol'n Control

Examples (continued)

- **rescontrol,file_summary** – List information of all the restarts available
- **rescontrol,status** – List the current restart settings

Note: These command options can also be executed in the GUI by going to **Solution>Nonlinear>Restart Control** in the unabridged menu

4. Files created and necessary for multiframe restart

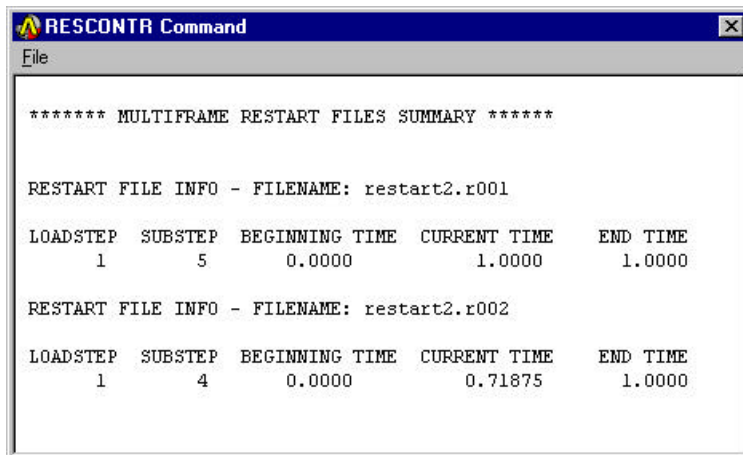
- *.rdb -- Database of model saved after the first iteration of the solve
- *.ldhi – Load history file contain all loads and boundary conditions for each load step
- *.rnnn – Element saved record files, where nnn is 001,002,003,etc. - similar to *.esav

5. Performing the restart

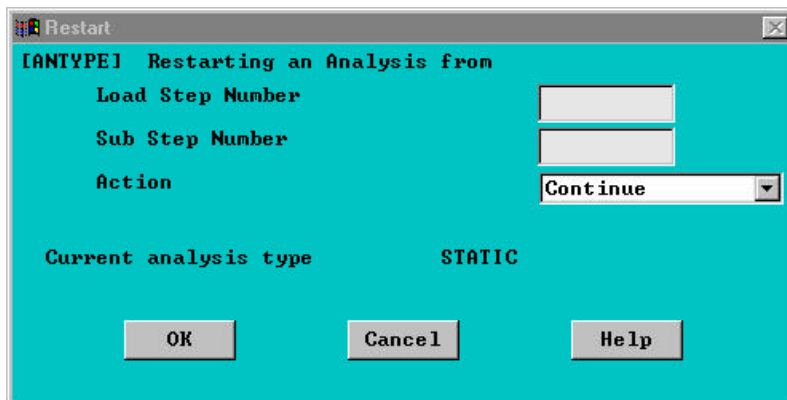
1. To perform the restart, go to Solution and pick restart. Two boxes will pop up. One showing a summary of the files you can restart from, and the second one allowing you to enter what load step and sub step to restart from.



Summary File

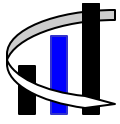


Restart Box



2. In most cases, all that is required is to enter the load step and sub step numbers. The action button set to continue allows ANSYS to pick up where it left off. Note: if you leave the sub step number box blank, ANSYS defaults to the last sub step.
3. If you are restarting from within a load step (as in load step 1 sub step 4 above) put in 1 for load step, and 4 for sub step. Pick OK. After that, you can specify changes, like add some equilibrium iterations or change the number of sub steps say for an un-converged solution. Finally, pick solve.

Command syntax –
antype,,rest,1,4
!make changes
solve



4. If you are adding an additional load step, like load step 2, put load step 1 sub step 5 in the boxes. Pick OK, then specify/pick all the commands necessary for the next load step. Finally, pick solve.

Command syntax –

antype,,rest,1,5

time,25 !specify time at end of this load step

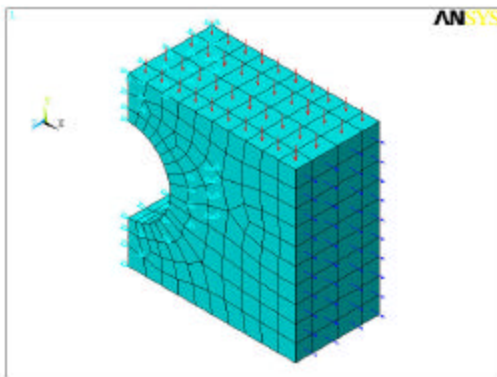
!add new loads, etc

solve

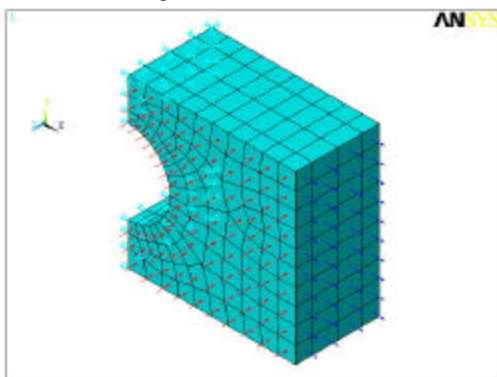
6. Example file

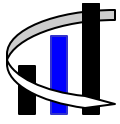
A block with a hole has two pressure loads applied as shown. The block has plasticity (Bi-Linear Kinematic Hardening) material properties defined, making this a non-linear analysis. In the first load step, the end pressure of –20000 psi is applied (blue arrows below). In the second load step, a pressure on the top face of 200 psi is applied. After solving this problem, the user realizes he/she meant to load the front face with 200 psi, not the top face. Instead of re-running the entire solution, why not just restart after the first load step where we applied –20000 psi? That is what is demonstrated by running the following input file;

Incorrect Loading of top face



Correct Loading of front face



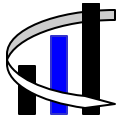


```
!MULTIFRAME RESTART EXAMPLE FILE
!
FINISH
/PREP7
BLOCK,0,2,0,1,0,.5,          !CREATE BLOCK WITH HOLE
CYL4,1,.5,.25,,,.5
VSBV, 1, 2
KWPAVE,5,6
wpro,,,90.000000
VSBW, 3
VDELE, 1,,1
ET,1,45                      !DEFINE SOLID 45 ELEMENT TYPE
UIMP,1,EX,,.10E6             !LINEAR MATERIAL PROPERTIES
UIMP,1,NUXY,,.3,
T,BKIN,1,1,,                !BILINEAR KINEMATIC HARDENING MAT. PROP.
TBMODIF,2,1,40000
TBMODIF,3,1,200000
!
ESIZE,0.1,0,                !MESH SIZE CONTROLS
SMRT,2
VSWEEP,ALL                   !SWEEP MESH OVER VOLUME
!
/SOLU
DA,13,SYMM                   !BOUNDARY CONDITIONS
DA,14,SYMM
DL,31,,UZ,
DL,33,,UZ,
DL,28,,UY,
SFA,6,1,PRES,-20000
!
NSUBST,20,40,10             !SET SOLUTION OPTIONS
OUTRES,ALL,LAST
RESCONTROL,DEFINE,ALL,1,1    !WRITE A RESTART FILE FOR EVERY LOAD STEP, AT
                                !EVERY SUB STEP WITH A MAXIMUM OF ONE RESTART FILE
                                !PER LOAD STEP

SOLVE
!
SFA,18,1,PRES,200           !APPLY 200 PSI TO AREA 18
SOLVE                        !RUN SOLUTION
!
FINISH
/POST1
PLDISP,2                    !PLOT DEFORMED SHAPE
                                !REALIZE THAT I MEANT TO APPLY PRESSURE TO AREA 22
                                !NOT AREA 18. OOPS! WAIT, I CAN RESTART FROM THE
                                !FIRST LOAD STEP SO I DON'T HAVE TO RUN IT ALL OVER
                                !AGAIN

FINISH
!
/SOLU
ANTYPE,,REST,1              !SPECIFY THIS IS A RESTART FROM THE LAST SUB STEP OF
                                !LOAD STEP 1
                                !APPLY PRESSURE TO THE CORRECT AREA 22
SFA,22,1,PRES,200
SOLVE
!
```

7. Issues with Parameters



An issue arises sometimes with parameters and the new multiframe restart. Let's say we run a model and we want to have ANSYS extract a results quantity using the `*get` command and store it in a parameter for us. We then plan to start up ANSYS again, and continue from where we left off, using this parameter somehow in the next solution. Possibly this parameter is a reaction force we extracted using `*get` from a displacement control run, and now we are going to remove the displacement and replace it with the force stored in this parameter. When you tell ANSYS you are doing a restart, first ANSYS resumes from the model `jobname.rdb`. It then looks in the `jobname.ldhi` to find the loads from the appropriate load step/sub step. It then uses the appropriate `jobname.rnnn` file to restart from. When ANSYS resumes from the `jobname.rdb`, it has lost the parameter you just defined! So what do you do? After you define the parameters/arrays, issue the **`parsav,all`** command. This will save all the parameters and the arrays to a text file name `jobname.parm`. Next, issue your restart command **`antype,,rest`**, with the proper arguments. Finally, issue the **`parres`** command to read the parameters back in.

8. Additional usage of the `antype,,rest` command

In addition to the above, the `antype,,rest` command can be used to end a load step at a given sub step even if the end of the load step has not yet be reached. Use the **`endstep`** option on the `antype` command to do this. The `antype,,rest` command can also be used to write out a results set for a load step and sub step that were not previously written out. Use the **`rstcreate`** option on the `antype` command to do this. Please see the `antype` command for more details

9. Conclusions/Recommendations

The multiframe restart capability at 5.6 can be very powerful, potentially saving a user lots of time, especially on models with long solution times. Implementing the multiframe restart is easy, as shown above. One word of caution, writing out the `jobname.rnnn` files takes space on your hard drive. Use caution not to fill your disk with too many of these files. To limit the number of these files created, use the `maxfiles` variable on the `rescontrol` command.

Sean Harvey
Collaborative Solutions, Inc.
Engineering Consultant



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

Fax: 760-431-4824

E-mail: firstname.lastname@csi-ansys.com

WWW: <http://www.csi-ansys.com>

FTP: <ftp://ftp.csi-ansys.com>

CSI Engineers:

Karen Dhuyvetter
Bill Bulat

Greg Miller
Sheldon Imaoka

Sean Harvey
David Haberman

Alfred Saad
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.



Collaborative
Solutions Inc

An ANSYS Support Distributor

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.