

TIPS FOR ANSYS

Carl Howard
14/06/2000

CONTENTS

WHERE TO GET HELP.....	1
INSUFFICIENT DISK SPACE.....	1
BULK COMPRESSION MODE.....	2
CHOICE OF SOLVER.....	2
ELEMENT NORMALS.....	2
BOOLEAN OPERATIONS ON SOLID MODELS.....	3
MESHING.....	3
LOADING AN ARRAY FROM A FILE.....	4
USING ETABLE AND EXTRACTING THE RESULTS TO AN ARRAY.....	5
OUTPUT WHEN USING THE GUI.....	5
GENERATING EDITABLE WMF FIGURES.....	5

Where to Get Help

There are some excellent resources on the net. Look at:

<http://www.infotech.tu-chemnitz.de/~messtech/ansys/ansys.html>

The xansys news group has heaps of solutions to problems. If you are stuck trying to solve a problem, start here.

<http://www.escribe.com/software/xansys/index.html>

Has the latest discussions on the xansys news group.

http://www3.sympatico.ca/peter_budgell/ANSYS_tips.html

Has some great hints and tips on using ANSYS.

<http://www6.50megs.com/imaokas/ansys.html>

More tips, tricks, macros, undocumented commands, inquiry functions.

Insufficient Disk Space

When trying to do a modal analysis using the Lanczos solver, the job might crash with the error, "INSUFFICIENT DISK SPACE", even though there is heaps of disk space. I tried to solve a model with 20,000 DOF and had 5Gb free and I was unable to solve the first mode. The problem was fixed by increasing the virtual memory setting to greater than 500Mb.

Below is a message posted on the XANSYS news group.

[xansys] Ansys5.5.3 and Memory Requirments for Lanczos Solver

I thought I'd pass the following information along for those who may have trouble with memory requirements in Ansys5.5.3. I've analyzed 3 different modal analysis problems ranging in size from 20,000dof's to 280,000dof's on models with and without CE's with the Block Lanczos solver. For all cases, 5.5.3 required 30-75% more -m to solve and took 10-20% longer to solve. The good news is the major file sizes, (ln07, ln22) are the same, wavefront the same, and the results are the same between the two versions. It almost appears as if the "MEMORY TO PERFORM EIGENEXTRACTION" estimate has been updated and now requires that additional ram to successfully run. Below are the memory estimates for a 59,388 dof solid45 model.

Ansys5.5.3 results, this version requires a -m 170 to solve, -m 160 failed

```
MEMORY TO PERFORM EIGENEXTRACTION:
  MIN. FOR PART IN-CORE AND OUT-OF-CORE = 17806368 135.85 (MB)
  MIN. FOR IN-CORE = 53694372 409.66 (MB)
  RECOM. FOR PART IN-CORE AND OUT-OF-CORE = 19061128 145.42 (MB)
  RECOM. FOR IN-CORE = 54749419 417.70 (MB)
```

Ansys5.5.2 results, this version takes a -m 110 to solve, -m 100 failed

```
MEMORY TO PERFORM EIGENEXTRACTION:
  MIN. FOR PART IN-CORE AND OUT-OF-CORE = 9734064 74.265 (MB)
  MIN. FOR IN-CORE = 37732140 287.87 (MB)
  RECOM. FOR PART IN-CORE AND OUT-OF-CORE = 10751272 82.026 (MB)
  RECOM. FOR IN-CORE = 37969692 289.69 (MB)
```

The 282,828 dof solid95, solid92 model with CE's required a "MIN. FOR PART IN-CORE AND OUT-OF-CORE" of 1350MB for 5.5.3 and 779MB for 5.5.2. I guess if you want to solve large vibration problems and have limited resources stick with Ansys5.5.2.

Bulk Compression Mode

The Lanczos solver is useful for extracting the 0Hz (bulk) mode of the acoustic cavity. This mode MUST be extracted in order to get accurate results below the first cavity mode. I was not able to get this result using the subspace method.

Choice of Solver

The only solver that I have been able to use for FSI problems is the frontal solver. None of the other solvers would converge after running the job for a day.

Element Normals

A common problem that occurs when meshing with shell elements is that the normal of elements have varying orientation. The way to check the orientation of the normal is to turn off all the numbering (element, volume, etc) and turn on the power graphics option. The shell element will have a purple colour on one side and a light blue colour on the other side. If the element normal are not correctly orientated, then you can flip them. Select the problem elements and use the command, Preprocessor | Move / Modify | Reverse Normals | Of Shell Elems. An alternative command is to select all the shell elements and locate a shell element that is orientated properly, then use the command Preprocessor | Move / Modify | Shell Normals, which will orient all the shell elements the same way as a chosen element.

This problem can also occur for line elements. A symbol can be turned on to indicate the direction of line normal. Use the command /psymb,ldir,1 or PlotCtrls | Symbols.

The /eshape,1 command can also be used to identify line elements such as beams with offset centroids that have incorrectly orientated normals.

Boolean Operations on Solid Models

- Perform all the boolean operations on the solid model BEFORE meshing. You can easily corrupt your model if you try to perform a boolean operation with entities that are meshed.
- Concatenate lines AFTER performing boolean operations such as glue and add.
- NEVER write a batch file that uses volume, area, line, etc numbers to identify an object. The numbering of objects changes with slight variations in geometry and can also change between machines using the same operating system! The only way to identify a part is by location or by using attributes such as real, type and material.

Meshing

As stated above, try to mesh your model after performing all the boolean operations on the solid model, otherwise you can easily contaminate your model.

By assigning attributes to parts of your solid model can save an enormous amount of time. Once all the attributes are assigned, you can type commands like vmesh,all and all the volumes will be meshed with the correct element types, material and real numbers.

If you have problems meshing a volume, try meshing the areas of the volume with dummy shell elements first and then mesh the volume. Alternatively, mesh one of the areas of the volume with a dummy shell element, then change the selected element type to the 3D element, extrude the area and generate the 3D elements at the same time.

Loading an array from a file

The documented ANSYS command for loading an array from a file stored on disk is next to useless. Fortunately ANSYS has improved the command but has not updated the documentation in V5.5.3. Here is how the command works:

*VREAD enhancement

The *VREAD command is now:

*VREAD,ParR,fname,fext,directory,NCOL

it is modified to allow:

*VREAD,ParR,fname,fext,directory,IJK,n1,n2,n3

where IJK may be any of:

I,J, or K if n2 and n3 are blank or zero and n1 > 0
IJ, JI, IK, KI, JK, KJ if n3 blank or zero and n1 and n2 > 0
IJK, JIK, JKI, KJI if n1,n2,n3 are all > 0

0, blank, or a positive number – NCOL

The use is as follows:

((ParR(K,J,I),K=1,n1),J=1,n2),I=1,n3)

Example usage:

/batch,list

*dim,a,,3,5

*create,test,dat

1.1 1.2 1.3 1.4 1.5

2.1 2.2 2.3 2.4 2.5

3.1 3.2 3.3 3.4 3.5

*end

*vread,a(1,1),test,dat,,ji,5,3

(5f6.1)

*stat,a(1,1)

*dim,b,,3,5

*create,test,dat

1.1 2.1 3.1

1.2 2.2 3.2

1.3 2.3 3.3

1.4 2.4 3.4

1.5 2.5 3.5

*end

*vread,b(1,1),test,dat,,5

(3f6.1)

*stat,b(1,1)

*dim,c,,3,5

*vread,c(1,1),test,dat,,ij,3,5

(3f6.1)

*stat,c(1,1)

*dim,d,,2,3,4

*create,test,dat

111. 112. 113. 114.

121. 122. 123. 124.

131. 132. 133. 134.

211. 212. 213. 214.

221. 222. 223. 224.

231. 232. 233. 234.

*end

*vread,d(1,1,1),test,dat,,kji,4,3,2

(4f6.0)

```
*stat,d(1,1,1)
/exit,nosave
```

Using ETABLE and Extracting the Results to an Array

When an etable is formed, the results from the whole model are loaded into the etable. If only a portion of the model is selected, then only the results relating to that portion are displayed to the screen. HOWEVER, this is not the case when trying to load the etable into an array parameter (a matrix). There are a couple of tricks that must be done. See the code below.

```
!find out how many elements are selected
*GET,length,elem,,COUNT

!find out the minimum and maximum element numbers
emin=elnext(0)
*get,emax,elem,,num,max

! Find number of sub steps = numst
set,last
*GET,numst,active,0,set,sbst

! Does not matter if the numbering is not sequential

!define a filtering mask
*DIM,mask,array,emax-emin+1,1
*vget,mask(1),elem,emin,esel
*vmask,mask(1)

! define an array of length equal to the number of elements
*DIM,SPL1,array,emax-emin+1,1 !this is the large array
*DIM,SPL2,array,emax-emin+1,1 !this is the compressed array
*DIM,SPL3,array,length,1      !this is the reduced array

*DIM,areabig,array,emax-emin+1,1 !this is the large array for the area
*DIM,area2,array,emax-emin+1,1  !this is the compressed array
*DIM,area,array,length,1        !this is the reduced array for the area

!
!       Determine the area of each element
!
ETABLE,artab,VOLU,          !defines an element table called areaetab
*VGET,areabig(1),ELEM,emin,ETAB,artab,,2
!the array can be further reduced to the required size
*vmask,mask(1)
*vfun,area2(1),compress,areabig(1)
!extract the first 1 to "length" elements
*VOPER,area(1),area2(1),ADD,
```

Output when using the GUI

If you are using the ANSYS in interactive mode and you try to run a batch file that has commands to redirect the screen output to a file, such as /output, prns, nlis, etc. ANSYS will open a new window to display the results and the file will NOT be created. These commands can only be used in batch mode.

Generating Editable WMF Figures

If you are using an NT version of ANSYS, through the interactive menu system, you can create a windows meta-file (WMF) that can be edited using Corel Draw. This can ONLY be done using the GUI and not in batch mode. Select PlotCtrls | Write Metafile | Invert White / Black. I had problems importing the WMF files (generated by ANSYS) directly into Corel Draw. I had to import the WMF file into Word97, then copy the figure into Corel Draw.

Generating Editable Postscript Figures

The default postscript output from ANSYS is a bitmap encapsulated in postscript and cannot be easily edited using graphics software such as Corel Draw and Adobe Illustrator. To generate editable postscript images, the display type must be changed. The display types are listed below and a tick indicates that the postscript output can be edited.

Editable Postscript	Type	Description
á	0	Basic display (no hidden or section operations)
á	1	Section display (plane view) use /cplane
á	2	Centroid hidden display (based on item centroid sort)
á	3	Face hidden display (based on face centroid sort)
á	4	Precise hidden display (like 3 but with more precise checking)
á	5	Capped hidden display (same as combined 1 and 3 with model in front of section plane removed)
	6	Z-buffered display (like 3 but using software Z buffering)
	7	Capped Z buffered display (same as combined 1 and 6 with model in front of section plane removed)
	8	Qslice Z buffered display (same as 1 but the edge lines of the remaining 3-D model are shown)
á	9	Qslice precise hidden display (like 8 but using precise hidden)

1. In ANSYS set up your screen to display your required output figure.
eplo
2. Select the desired display type
/type,1,2 !window 1, centroid hidden display
3. Change the output to a file
/show,filename,grph !output written to filename.grph
4. Replot the image
/rep
5. Print any other screen outputs to this file
plesol,nmisc,4
6. Return the output to the screen
/show,x11

Start the display program, then enter the following commands

```
file,filename,grph
/show,pscr
pscr,color,2
```

Then enter the following commands with the appropriate numbers depending on your desired page orientation

	Portrait	Landscape
pscr,tranx,	40	540
pscr,trany,	200	100
pscr,rotate,	0	90
pscr,scale,	0.18	0.2

To print out all the plots type the command

```
plot,all
```

The directory will contain the encapsulated postscript files pscr*.grph from the screen captures.