



A Publication for ANSYS Users

Contents

Feature Articles

- [Post Processing: Reading and Editing the Results File](#)
- [Hydrostatic and Deviatoric Stresses](#)
- [Search Operators in Help](#)

On the Web

- [2005 Western Region ANSYS Technical Conference](#)
- [ANSYS User Conference 2006](#)

Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About *The Focus*](#)
 - *The Focus* Library
 - Contributor Information
 - Subscribe / Unsubscribe
 - Legal Disclaimer



ANSYS
Support Distributor

The Focus



A Publication for ANSYS Users

Post Processing Tips & Hints: Reading and Editing the Results File

by Eric Miller, PADT

This series on Post Processing with ANSYS has taken a while, but with this article it comes to an end. Over the past year that we have been trickling these out, there has been a lot of good feedback and comments. We hope everyone has found them useful. The last topic will be a look at reading and changing the ANSYS result file, something that is not as hard as it seems and, when you need it, can be very useful. We will talk about reading a dumped version with /AUX2, some simple changes you can make with /AUX3, and point to some other references for more complex editing using FORTRAN routines. Your prize for reading through the bulk of this is learning how to change the Title in a result file, a much better solution than trying to remember to redefine the title every time you plot.



What Is in the Results File?

Well, mostly results. But the file also contains a copy of the finite element model (not geometry or BC's, but nodes and elements). Some users may not know this, but if you don't have a *.db file, you can still look at your model and post processes. If starting from a session with no nodes or elements, when you issue a SET command, ANSYS reads the model information from the *.rst file into the database (remember, the database is what ANSYS holds in memory, not what is in a *.db file). Inside the *.rst,



A Publication for ANSYS Users

after the FE model, results sets are stored. Each set contains the information that the user requested for substeps that the user requested with OUTRES. As you can imagine, this can be a lot of data, so ANSYS stores it in a fairly concise manner that makes it easy to read and write quickly.

Many third party applications actually read the result file instead of the *.db files. This is because the format of the *.rst file is documented and the *.db is not. You could probably figure out the *.db file but there is no commitment from ANSYS, Inc. to keep it the same. The *.rst file tends to stay consistent and new features are added rather than changed.

The format of the .rst file is documented in the manual “ANSYS Programming Guide, Interfacing with ANSYS” available from your ANSYS sales person. If you don’t like manuals and are a little braver, everything is also included in the ANSYS custom/include directory in FDRESU.INC. This documents all the headers and where things are stored in the *.rst. The key thing to know is that each chunk of data is stored in a record and that records can be headers that point to other records, or they can be data.

Before /AUX3 Comes, You Guessed It: /AUX2

ANSYS provides a group of utilities in the /AUX2 processor for converting the various binary files it creates into readable text files. Beware though, the amount of information created is **huge!** To get started, we will run a one element model (Listing 1) and look at its result file with /AUX2.

Listing 1. Simple 1-Element Model.

```
finish                                et,1,45
/clear                                e,1,2,3,4,11,12,13,14
/file,onelm                            ex,1,10e6
/prep7                                 nuxy,1,.23
n,1,0,0                                d,1,all
n,2,1,0                                d,2,all
n,3,1,1                                d,3,all
n,4,0,1                                f,11,fy,100
n,11,0,0,1                             finish
n,12,1,.123,1                          /solu
n,13,1,1,1                             solve
n,14,0,1,1
```

When you go into /AUX2 (by typing /AUX2) the output window lists the commands available. You can get to /AUX2 through the menu File>List>Binary Files, but for this type of work typing can be better. The commands are:

Table 1. /AUX2 Commands.

FILE,fname,fext,fdir	Use to define the file to dump
----------------------	--------------------------------



A Publication for ANSYS Users

Specify what you want to dump.

FORM, lab

- Lab = RECORD will print just the header info for each record
- Lab = TEN will print the same info as RECORD plus the first 10 words in the record
- Lab = LONG will print everything

STAT

Prints out the current /AUX2 option settings
Tells /AUX2 to dump (print to screen) the file. NSTART and NSTOP specify which records to dump. If you set NSTART to 1 or HEAD, you just get the header. If you set it to ALL, you get everything.

DUMP, NSTART, NSTOP

HBMAT

Specifies that you dump a matrix file (not covered here)

FINISH

Leave AUX2 and go back to the begin level

If we enter the following into /AUX2:

FILE, , RST

FORM, LONG

DUMP, ALL

we get [dump1.txt](#). If you get out [FDRESU.INC](#), which is included in your ANSYS installation, you can decode the file. As an example, we will find the Y location and Y displacement on node 12.

1. Think of everything in this file in one giant single column array of numbers. Each chunk of data is stored as one or more binary real numbers. DP numbers and strings are created by converting the number into a real number for storage. The location of each chunk is called its LOC. Chunks of data are grouped together into something called a *record*. Each record knows its length, which is the number of items (real numbers) in the record. Each item is either a value we care about or a reference to the LOC of something we may be looking for.
2. In the file [fdresu.inc](#), find the information you desire and its corresponding label. For example, if we wanted to know the number of DOFs solved, we can see it is the 5th item in the results header. Now, moving back over to our [dump1.txt](#) file, we index the 5th item in Record 2, to see the number is 3. In the event that the information you want has a label that begins with “ptr”, then you know that the number listed in Record 2 is going to be a pointer — it is going to point to another record which in turn will help you extract the information you want.
3. So, proceeding with extracting the displacement of node 12, first skip the standard header (record 1) and go to the RST header (record 2).
4. Node numbers are compressed and may not be in order, so we need to find the “Node Equivalence Table record”, ptrNOD, as described in [fdresu.inc](#). This is the 15th item in the RST File header. Note the handy counter on the far right of the file (< 5, < 10, etc.). So item 15 in record 2 is 152. Because our label begins with ptr- we realize this 152 number is a pointer to Location 152.
5. Looking down the file we see that this is Record 4. A quick look shows that node 12 is the 6th node in the list. Store this in your memory.
6. Now we need to find the nodes. Go back up to the 2nd record and find ptrGEO. The model



A Publication for ANSYS Users

information is stored in its own chunk of the file and it has its own header that points to where the model information is held. ptrGEO is going to tell which record is the header to this chunk of geometry information. [It is called geometry because this file was created long before ANSYS supported “real geometry” like lines and surfaces.] So ptrGEO is item 16 in the record, which is LOC 7176 for our example.

7. Scrolling down we see that Record 9 is what we are looking for. As expected this is another header. To extract information from this header, we will again reference our include file. Note that we are looking further down in the include file this time, not in the .rst header, but now an index of the GEO header which is what ptrGEO pointed to. In the include file we see that the node location is stored in ptrNODL, item 27. This equals 7376. (If you look at the include file you will see a ptrNOD and probably think that is the pointer you want. Wrong! That is the “old” pointer that ANSYS no longer uses. Since ANSYS supports very large node numbers, they need a 64 bit number to give the location of the nodes, so this value is now in ptrNODL.) Still with us? Kind of like a treasure map.
8. Scroll down and you will see that record 12 is the start of the nodal coordinates (LOC 7376). There is a record for each node. Looking in the include file for the description of the nodal coordinate record (NOD) we see it is N, X, Y, Z, THXY, THYZ, THZX. So for node 12, our 6th node (remember step 4?) we count down 6 records and then over 3 for the Y value: 0.123! Now you know why we used a weird y value. Zero would have been boring.
9. To get the Y deflection, we need to get the solution data for the load step of interest. For our test, this is the first load step. So back to the header, we want to find the location of the data steps index header, ptrDSI. This is item 11 which is LOC 167.
10. Jumping down we see LOC 167 is record 6. There is only one data set on the file so the first item is the pointer to the header: LOC 7538.
11. In the include file you will find the description of this header under “Solution data header”. This is a very complex header that supports all types of physics and such. For what we need we can skip over most of it and look for the pointer to the nodal solution: ptrNSL. This is the 11th item. Looking at our dump we see that it points to LOC 609. Wait, that is back above! Somewhere in the middle of the solution data set pointers! Ahhh!!! Don’t panic. The LOC pointers in the solution data are relative to the start of the data set. $7538 + 609 = 8147$. Scroll down and you will find that it is record 25.
12. We know a SOLID45 has 3 DOFs, so for node number 12, the 6th node, we go 3 items for each node prior, plus two more to get to the y value: $3 * 5 + 2 = 17$. The value is 0.933316E-04. A quick PRDISP in POST1 confirms this.

As you can see, this process is not very difficult; it just has lots of little steps. You can also see that it lends itself to programming. Using the dump facility is great for understanding the structure and debugging, but you would never use this hand method for large result files.



A Publication for ANSYS Users

Putting your Result File on a Diet

The most common thing people want to do with a result file is trim it down. When a run is done you may not want, or cannot store, all of the data for all of the steps. The thing to do at that point is edit the file with /AUX3. You can also change the values of the solution sets so they make more sense after deletion. Table 3 lists the /AUX3 commands:

Table 2: /AUX3 Commands.

<code>FILE, fname, fext, fdir</code>	Use to define the file modify
<code>LIST, lab</code>	Lists the header info, similar to the set command. If you specify lab=2, it will give titles. This is also used as your STAT command to see what changes you have made.
<code>DELETE, SET, nstart, nend</code>	Marks set nstart through nend for deleting. You can give a range or specify multiple sets one at a time.
<code>UNDELETE, SET, nstart, nend</code>	Removes the delete mark on the specified sets. UNDELETE, ALL deletes all deletes.
<code>MODIFY, Set, Lstep, Iter, Cumit, Time, kTitle</code>	Allows you to specify new index values and a title for the given Set number. To change the title you set kTitle to 1 and it changes it to the value specified in the last /title command.
<code>COMPRESS</code>	This processes any deletes you have asked for. The modify commands take effect immediately.

To show how this works, we have a new single element model in tstaux3.mac. We can then make some changes with:

Listing 2. Change Title on Step 3.

```
! Change Title on Step 3
/aux3
file,,rst
/title,New Title for Data Set 3
modify,3,,,,1
list,2
finish
```

Listing 3. Delete Step 3.

```
! Delete Step 3
/aux3
file,,rst
delete,set,3,3
compress
list
```



A Publication for ANSYS Users

Listing 4. Change Time on Step 2.

! Change Time on Step 2

/aux3

file,.,rst

modify,2,,,,100

list

finish

Paste these guys in and you will see how they work. If you go into POST1, you will be able to work with these files just like normal. One suggestion: Use the /COPY command or the OS to make a copy of your RST file before you start playing with it. Also, there was a known problem in 9.0 with some of these commands that is fixed in 10.0.

When an /AUX2 or /AUX3 Just Doesn't Cut It

Many users have needs that /AUX2 and /AUX3 just are not enough for. In these cases you need to break out the FORTRAN manual and write some programs to read and write the RST files. We did an article on this last year, "[Reading and Writing Binary Files](#)". It covers the basics of what you need to know.

Don't get too scared. There are some useful examples that come in the Custom directory of the ANSYS install that you can start with.

When it is all said and done, and you consider the era in which it was created, the binary results files created by ANSYS are pretty cool. They pack a lot of data in there and allow you to jump around pretty easily. When you think of the complexity and variety of data that it needs to store, it is impressive.

The Focus



A Publication for ANSYS Users

Hydrostatic and Deviatoric Stresses: Professor Jargon's Term of the Month

by Professor F. E. Jargon, Jargon Technical School

Prof. Jargon returns again to discuss common finite element terms and their meaning and relevance to the finite element analyst.

Ah, it's been a busy month, students. I'm about to take a break so I can get out and enjoy this fine Phoenix summer, but before I do, I thought I would treat you to another bit of information regarding finite element methods. This goes back a bit to your Def Bods class (*Introduction to Deformable Bodies*, for those of you who don't use cool abbreviations like I do).

Consider a block lying in 3-D space. For a given load, the faces of this block experience tensile stresses normal to the faces and shear stresses tangent to the faces. See Figure 1 to get an idea of what I'm talking about.



Professor F. E. Jargon

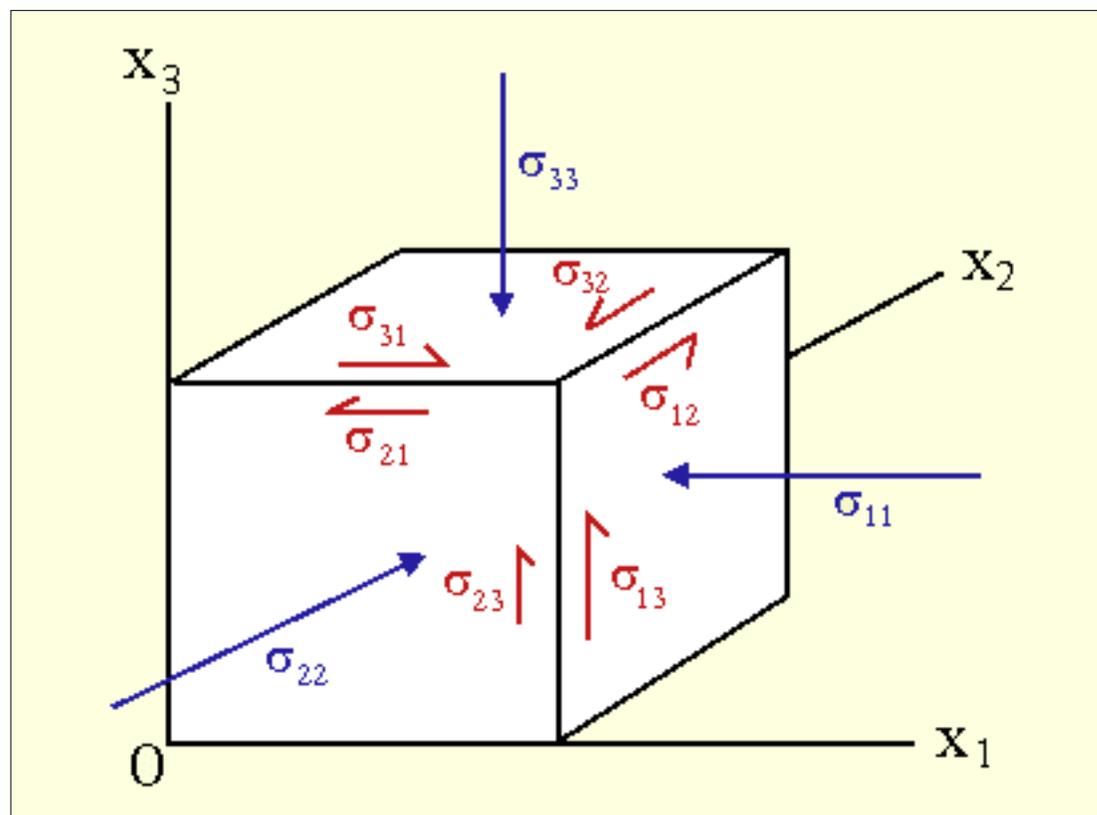


Figure 1. General stresses on a control volume.

The Focus



A Publication for ANSYS Users

These stresses can be expressed as a matrix known as a stress tensor:

$$\sigma_{ij} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix}$$

The tensor can be broken down into two components: stresses that act on the block to change the volume and stresses that act to distort the volume. The former stresses are referred to as *hydrostatic* stresses and the latter as *deviatoric* stresses.

Since the hydrostatic stresses maintain the original proportions of the volume, it makes sense that they would be equal. Thus the hydrostatic stresses are simply the mean of the principal stresses.

$$\sigma_m = \frac{\sigma_1 + \sigma_2 + \sigma_3}{3}$$

The deviatoric stresses consist simply of the hydrostatic stresses subtracted from the original stress tensor. The resulting matrix includes tensile stresses that elongate the volume as well as shear stresses that cause angular distortion—in other words, stresses that cause the volume to deviate from its original proportions.

For a simple example, consider the cube in Figure 2, subject to an evenly distributed compressive load of 210 lb in the y-direction. Assume each face has an area of 1 in², even though I didn't do a very good job of drawing it that way. Since the loading is uniaxial, we can leave Professor Mohr out of this.

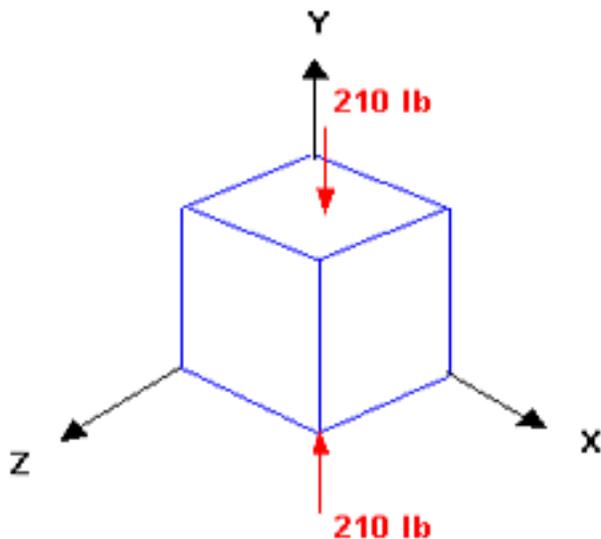


Figure 2. Block subject to 210-lb compressive load.

It is apparent that the cube will compress in the y-direction. It is also apparent that the cube will expand in the x- and z-directions due to Poisson's effect.



A Publication for ANSYS Users

The hydrostatic stresses on each face may be calculated as:

$$\sigma_m = \frac{-210 + 0 + 0}{3} = -70 \text{ psi}$$

Thus the hydrostatic stresses here are represented as 70 psi stresses on each face that act to compress the volume.

Deviatoric stresses are simply

$$\sigma_d = \begin{bmatrix} 0 + 70 & 0 & 0 \\ 0 & -210 + 70 & 0 \\ 0 & 0 & 0 + 70 \end{bmatrix} = \begin{bmatrix} 70 \text{ psi} & 0 & 0 \\ 0 & -140 \text{ psi} & 0 \\ 0 & 0 & 70 \text{ psi} \end{bmatrix}$$

Thus the contraction/expansion of the volume is represented as a 140 psi compressive stress in the y-direction and 70 psi tensile stress in the x and z-directions.

Well I'm off for my own personal festival that I affectionately refer to Carcinoma 2005. Take care until next time when I have more jargon to throw your way.

Ever run across a word in the ANSYS world that you were unsure about but were afraid to ask about? [Send us an e-mail](#) and we will forward it to Professor Jargon. All submissions will be kept strictly anonymous.



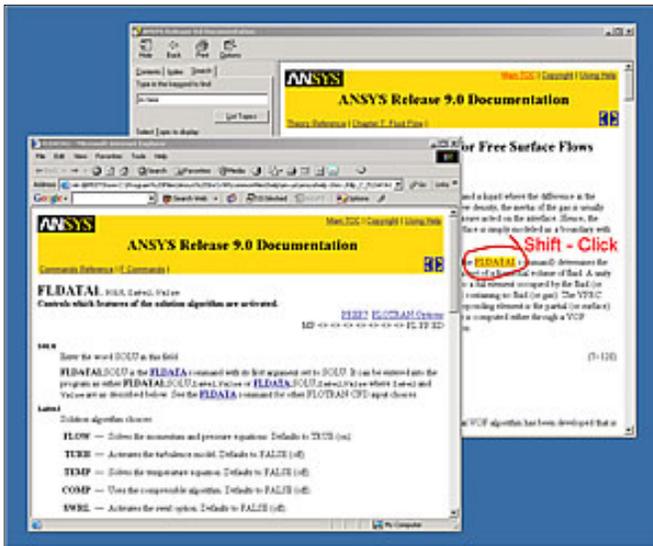
A Publication for ANSYS Users

Search Operators in Help

by Rod Scholl, PADT

How 'bout four basic tips for using Help?

1. Many of us were recently enlightened when Joe Woodward pointed out that the ANSYS Help program would take logic operators. I've tried + 's and - 's before to no avail — and I think we all lament the missing ability to add quotes (") to a string — but I never thought to try AND 's OR 's and NOT 's. These operators function as expected in the search field.
2. Ever try Shift-Click? Often I am navigating in Help, want to follow a link — but don't want to lose where I'm currently at. There's only so much one can do with the back button. Shift-Click on a link, opens the link in a new window.



3. Ever wish you could send a link to someone, rather than specifying the guide and then the location such as 8.7.3.1? Not the cleanest, but when sending a Help link to a peer, try

Options> Internet Options...

Then on the homepage area, click "Use Current" to display the URL of where you are browsing. It should look something like this:

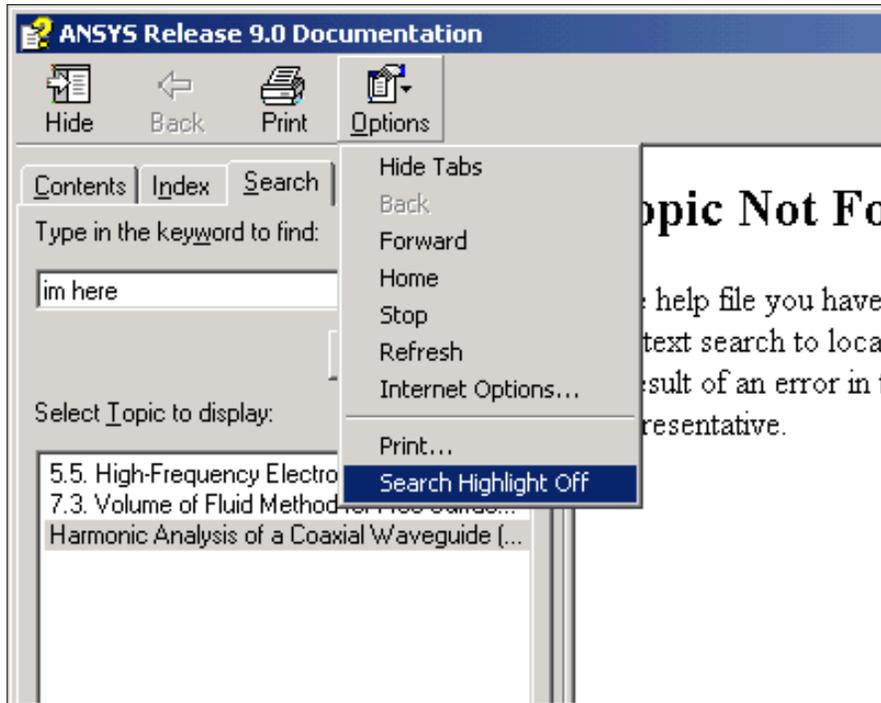
mk:@MSITStore:C:\Program%20Files\Ansys%20Inc\v90\commonfiles\help\en-us\ansyshelp.chm::/Hlp_C_TOFFST.html

If they use Internet Explorer, and install to the default location, it might just bring them to the section you want to refer to. Of course, you may have to go back and reset your homepage to <http://tracking.my.stocks.instead.of.working.com>.

3. Did you also know that you can turn the search highlighting off?



A Publication for ANSYS Users



I find this annoying when I'm e-mailing snippets from Help, or capturing descriptions for documentation.

Okay... all a bit thin, but the Booleans in the search field are nice.



A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.