

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

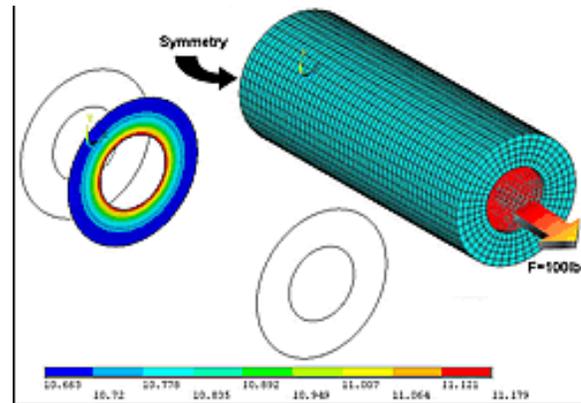
- [Surface Commands](#)
- [ANSYS Memory and Disk Space Management](#)
- [Benchmarking ANSYS](#)

On the Web

- [PADT's Rapid Manufacturing Solutions](#)
- [PADT's Enhanced Tech Support](#)

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



A Publication for ANSYS Users

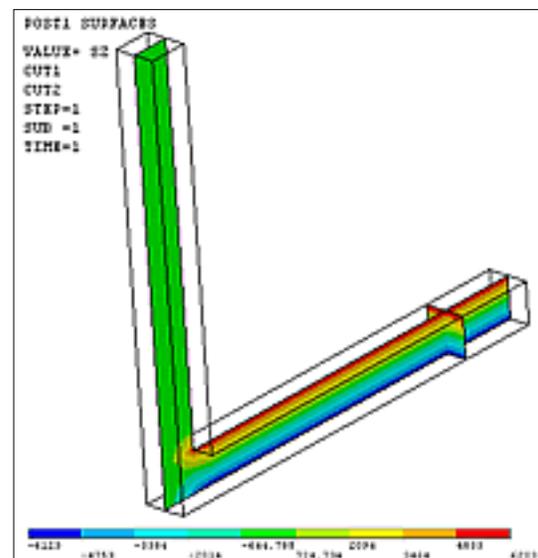
Surface Commands

by [Joe Woodward](#), PADT

Until now, to get a cross-sectional view of the results of a model, you could use the /TYPE command, or make sure that you had a cut through your model at the right position. The /TYPE command worked fine for getting plots, but you could not get information about the cross-section. In version 8.0, ANSYS has added a new set of commands for getting plots and information from cross-sections. These Surface commands are used to setup, plot, store, and resume information mapped onto the surface. Here is a brief description and example.

The SUCR command creates the surface onto which results will be mapped. Each surface is given an arbitrary name, which is used in subsequent commands like SUSEL that selects individual surfaces. Currently the surfaces can be flat or a sphere of specified radius. Flat surfaces use the working plane and spherical surfaces have their center at the origin of the working plane. There is an argument on the SUCR command for the level of refinement for the created surface. This refinement is a factor of how many interpolation points are created per element that the surface crosses.

The SUMAP command maps results from the database onto all of the currently selected surfaces. Each result set is given an arbitrary name so that several result sets can be stored for each surface. The results are mapped in the current results coordinate system and can be any of the quantities available through the PLNSOL command. Surface results are then plotted with the SUPL command, which specifies which surface and result set to plot. You can also choose whether to show just the surface or to include the outline of the selected elements. As shown in figure at right, more than one surface can be shown at a time, which, for me, beats using the /TYPE command.



The Focus



A Publication for ANSYS Users

But wait, it gets better&

ANSYS has stepped up to answer the users requests for usable cross-sections. When the surface is created with the SUCR command, data about the interpolation points are stored automatically. The coordinates of each point, the normal vector of the surface, and the discretized area at the point are all stored. As results are mapped to the surface, they are stored with this data and can be accessed by several other commands.

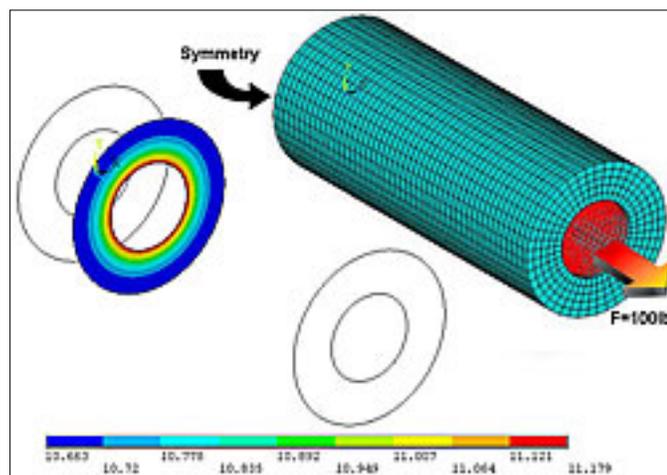
The data can be dumped to an array variable using the SUGET command, which by the way, initializes the array so that no *DIM command is needed. The data can also be accessed directly by the SUCALC, SUVECT, and SUEVAL commands. The SUCALC command creates a new mapped result on the surface by performing mathematical functions on two other mapped quantities, much like the *VOPER command does with arrays. This is shown in the example below.

The SUVECT command is used to perform vector calculation on the surface data. The cross product, for example, can be used to find the flow normal to the surface in a Flotran model.

The last command used to manipulate the data is SUEVAL. This command will integrate, average, or sum the given data over the surface. This can be used to find the force and moment through an arbitrary cross-section as I have done in the example below.

The amount of data, and the accuracy of the SUEVAL integration function, depends on both the refinement value given in the SUCR command and on the density of the mesh in the area of interest. On a sample problem comprised of a thick walled tube with internal pressure and a 100 lb axial force, the calculated cross-section force varied greatly with the number of elements through the thickness.

The results of the integration of the axial stress are listed in Table 1. Notice that there is a greater dependency on the mesh density than there is on the refinement level. However, the example at the end of this





A Publication for ANSYS Users

article, of an L-shaped beam under load, delivered very accurate force and moment calculations despite the mesh density. So it is always a good idea to refine the mesh density and run the surface calculations again to assure proper convergence of the integration operation.

# Elem. Thru Thickness	Surface Refinement			
	0	1	2	3
1	132.1145	133.0037	133.2203	133.2741
2	108.7705	108.9528	108.985	108.9891
3	103.8772	103.927	103.9353	103.9361
4	102.1385	102.1554	102.1585	102.1588
5	101.3703	101.3784	101.3798	101.38

Table 1. Calculated force value. The correct value is 100.

Once surfaces are setup, they can then be written to a file with the SUSAVE command. This stores all the surface geometry information and any result sets that have been mapped to the surfaces. This information can then be reloaded into the database using the SURESU command. You can resume the surface information without having the original model loaded. This will allow you to share cross-section plots and data without having to send the whole model.

I have been quite impressed with the functionality and forethought that ANSYS put into this set of commands. They should allow you to have greater plotting and model checking flexibility. Play with them and explore their potential. It is definitely worth the short amount of time that it takes to learn them.

```
##### L-Shaped Beam Example #####
/prep7
b=1    !beam section width
h=1    !beam section height
nelem=2 !Number of elements through the thickness
mp,ex,1,30e6
mp,nuxy,1,.3
wpcs,1,0
block,-b/2,b/2,-h/2,h/2,0,10    !build horizontal beam
block,-b/2,b/2,-h/2,h/2,10,10+h !build corner section
block,-b/2,b/2,h/2,10,10,10+h    !build vertical section
nummrg,kp
```



A Publication for ANSYS Users

```

et,1,95          !High order solid hex elements
esize,min(b,h)/nelem
vsweep,all

asel,s,loc,y,10,,1    !Area at the top of vertical section
f,all,fz,100/ndinqr(0,13) !100 lb force in the Z direction

csys,0
asel,s,loc,z,0,,1    !Area the end of horizontal section
d,all,uz,0          !Symmetry conditions
nselect,r,loc,x,-b/2
d,all,ux,0
d,ndnext(0),uy,0

save
/solu
eqslv,pcg
alls
solve

/post1
wpc,1,0            !Position the work plane
wpoff,,2

sucr,cut1,cplane,3 !Create a planar surface CUT1 using
                    !refinement of 3
sumap,sz,s,z       !Map Z-direction stress and name it SZ
supl,cut1,sz,1     !Plot result set SZ on surface CUT1
sueval,f1,sz,intg  !Get axial force by integrating axial stress
sueval,a1,da,sum   !Get cross-section area by summin
                    !the dA geometry data

sucalc,szadj,sz,exp,,1,-f1/a1 !Create pure moment results
                               !by subtracting axial stress
suget,cut1,szadj,data,1       !Dumps the szadj result set
                               !from surface cut1 to an array
                               !called data . You can used the
                               !*Vxx set of commands to find
                               !where szadj is zero to find

```



A Publication for ANSYS Users

```

!the neutral axis.
fyoff=0          !In my case, this should be
                 !the neutral axis.
sucalc,yc,gcy,exp,,1,-fyoff !Use the EXP function to add
                 !constants to the results.
sucalc,mom1,sz,mult,yc
sueval,zmom,mom1,intg      !Integrates (yc*sz) to get moment

sucalc,ysquare,yc,exp,,2
sueval,I1,ysquare,intg     !Integrates yc^2 to get the
                 !moment of inertia

suget,cut1,yc,cut1data,1   !Dumps global Y coord. to array
*VABS,0,1
*vscfun,c,max,cut1data(1,8) !Finds the max fiber distance.
                 !This would need to be adjusted
                 !if section is not on the origin.
poa=f1/a1              !Hand calculation for P/A
mcoi=zmom*c/I1        !Hand calculation for Mc/I

```

The Focus



A Publication for ANSYS Users

ANSYS Memory and Disk Space Management

by [Ted Harris](#), PADT

The way that ANSYS utilizes memory has been enhanced over the last few releases. In the old days, we specified workspace and database memory allocations (or accepted the defaults) and those are the amounts of memory ANSYS used for the entire session.

Starting around release 5.5, ANSYS started to dynamically allocate memory under certain circumstances. Normally this was during model solution, for certain solvers. ANSYS still does dynamic memory allocation, but now there are more options in implementing this. We'll describe the different options and how they work.



First, some terminology.

Total Workspace. This is the total amount of memory that ANSYS can use, whether it be for the nodes and elements, loads, results database, as well as memory needed to perform Boolean operations or solution calculations.

Database Space. This is a subset of the total workspace that has been designated for items contained in the ANSYS database. This includes nodes, elements, loads, geometry, and also includes results data once we issue the SET command in the General Postprocessor.

Database. This is the in-memory data that describes your model's nodes, elements, geometry, loads, material properties, etc. This is what gets written to the hard disk when you perform the SAVE operation.

Database File or .db File. This is the file that resides on your hard disk after you perform the SAVE operation. It is a "snapshot" of the state of your model when you performed the save.

Scratch Space. Sometimes called heap space. This is the amount of memory that is the difference between the total workspace and the database space. This is the memory available for calculations, such as Boolean operations in the preprocessor, as well as solution calculations.



A Publication for ANSYS Users

Page File. In regards to running ANSYS, this is the ANSYS-created jobname.page file in the working directory on your hard disk. If it has nonzero size, it is being used as virtual memory to increase the database space memory over what has been specified. Note that there may also be operating system usage of a page file (see *system page file*), which is a different concept that happens to share the same terminology.

System Page File. This is the computer operating system's virtual memory. It is a portion of your hard disk that has been set aside for the operating system to use as supplemental memory above and beyond the physical amount of RAM in your computer.

There are two scenarios we'll look for utilizing memory in ANSYS.

Scenario 1. Explicitly specifying workspace and database space.

This is done either on the command line using `-m` and `-db`, or from the ANSYS launcher by specifying values in the Total Workspace and Database fields. The default values for ANSYS 7.1 and 8.0 are shown in Table 1.

Version	Database Space	Total Workspace
ANSYS 7.1 Windows	32 MB	64 MB
ANSYS 7.1 Unix	64 MB	128 MB
ANSYS 8.0 Windows	256 MB	512 MB
ANSYS 8.0 Unix	256 MB	512 MB

Table 1. ANSYS default memory allocations.

We can specify our own settings to change the values from the defaults. Note that since the database space will always be a subset of the total workspace, the value specified for the database must be less than that specified for the total workspace. As a rule of thumb, we normally recommend making the total workspace at least twice the value specified for the database.

So how does a user know when to use the default settings and when to specify explicit values? Two main considerations come into play. First, are we solving, or are we pre- or post-processing? If we are just solving, then we can live with a small value for the database space, and therefore maximize the amount of memory available as scratch space. It's scratch space that will be most used for solving.

On the other hand, if we're doing mostly pre- or post-processing, we'll be accessing the database space a lot, so we'll want to make sure we specify enough database space that ANSYS won't create its jobname.page file. The .page file tends to really slow down our

The Focus



A Publication for ANSYS Users

plotting. If you're working interactively and you see the message,

Note: ANSYS page file being use.

This is your cue to save what you're doing, get out of ANSYS and get back in with a larger specification for the database space. How do you know how large of an amount to specify? You want the amount of memory allocated for the database space to be larger than the hard disk space that will be taken up when you save the database. In other words, if your .db file takes up 120 MB on your hard disk, you'll want to specify more than 120 MB for the database space. If you don't, ANSYS will create its jobname.page file and your plotting will get very slow.

Further, if you are doing postprocessing, your results will also be stored in the in-memory database. The results typically take up the same amount of room in memory as the "preprocessing" database. This means that when you're postprocessing you should make the database space allocation at least twice the amount of space that the .db file takes up on the hard disk. Otherwise, the .page file will be created and your plotting will bog down.



One thing to be aware of is that although ANSYS cannot increase the amount of memory allocated to the database space, it *can* and *does* dynamically allocate the total workspace, if necessary (unless one uses the `-f` option discussed below). This increased allocation may happen during Boolean operations or during a solution. If not enough memory is available to the solver, for example, ANSYS will attempt to allocate more, including the memory available in our RAM as well as system virtual memory. ANSYS cannot dynamically allocate forever, though, as it is limited by the amount of contiguous memory available, as well as a system limitation. The amount of memory available depends on how much RAM we have, how much virtual memory is specified on our system, and how much memory is being utilized by the operating system and other applications. The system limitation for ANSYS memory allocation on most Windows systems is 1,700 MB (and only 1,400 MB are available via the `-m` option). This means that no matter how much RAM and virtual memory you have, ANSYS cannot allocate more 1.7 GB of total workspace. The exception to this limit is with Windows 2000 Server and Windows XP when there is 4 GB of physical RAM on the computer. When this is the case, there is a 3 GB operating system switch that can be set that will allow ANSYS to address 3 GB for the database. Table 2 shows ANSYS total workspace limits for various operating systems.



A Publication for ANSYS Users

Hardware Platform	Maximum -m Value
Compaq Alpha	16,383 MB
HP PA8000 32-bit	1,880 MB
HP PA8000 64-bit	16,383 MB
HP Itanium IA64	16,383 MB
IBM RS/6000 32-bit	2,047 MB
IBM RS/6000 64-bit	16,383 MB
SGI 32-bit	2,047 MB
SGI 64-bit	16,383 MB
Sun UltraSPARC 32-bit	2,047 MB
Sun UltraSPARC 64-bit	16,383 MB
Intel IA-32 Linux	2,040 MB
Intel IA-32 Windows 2000	1,700 MB
Intel IA-32 Windows NT 4.0	1,700 MB

Table 2. ANSYS memory limits for various operating systems.

Scenario 2. Using the default memory model.

Starting with version 6.1, ANSYS has implemented the default memory model, which attempts to take some of the difficulty out of memory allocation, and it attempts to allow more memory to be utilized for scratch space by reducing the dependency on the availability of contiguous blocks of memory. The default memory model starts with the default settings (see Table 1) and will dynamically allocate total workspace as needed.

Memory that we explicitly specify for the workspace (-m) must be in one contiguous block. Whereas if we allow ANSYS to dynamically allocate it, the memory does not have to be in one contiguous block. Also, with the default memory model we're not tying up memory allocated to the workspace that may be too big. This means we'll have more memory available to other applications or the ANSYS help system.

The big drawback of the default memory model in versions 6.1 through 7.1 is that the amount of memory allocated to the database space remains fixed at 32 MB. That means that if you need more than that for pre- and post-processing a model, ANSYS will create its jobname.page file and your plotting will slow down dramatically. For this reason, we at PADT recommend the default memory model for solving, but not for pre- and post-processing, unless you're working with a fairly small model for which the database space never exceeds 32 MB.



A Publication for ANSYS Users

What if I don't want ANSYS to dynamically allocate memory at all?

By default, no matter which method we use to specify memory, ANSYS will attempt to dynamically allocate total workspace if it needs it. If for some reason you don't want ANSYS to do this, the feature can be turned off with the "fixed memory option". This can be activated by specifying `-f` on the command line, or by `MEM_GRO` to zero in your configuration file. More information on the `-f` option and memory in general can be found in the ANSYS online help's Basic Analysis Procedures Guide, Section 19.

Are there circumstances when ANSYS won't dynamically allocate memory at all?

As of version 7.1, ANSYS doesn't support dynamic memory allocation on 64-bit Intel systems or Linux systems. Also, there is a documented feature of `-f nogrow` on startup that intends to disable the dynamic memory manager. However, from version 6.1 to 8.0 on Windows (and maybe UNIX) a program bug prevents the `-f nogrow` from functioning. The plain `-f` command performs as documented.

How else can I control dynamic memory allocation?

There are a few knobs to turn within the ANSYS configuration file. The configuration file can contain some settings that ANSYS will use every time you start up the program. The list of all the available options is shown in Section 19.4 of the Basic Analysis Procedures Guide. The settings that affect memory and memory allocation are:

- `VIRTM_MB` allows changes to the default total workspace allocation
- `NUM_VPAG` maximum number of database pages in memory
- `SIZ_VPAG` size of each database page
- `NUM_DPAG` number of database pages on disk (.page file)
- `MEM_GROW` starting size of dynamic memory allocation.

More details on these parameters are given in Section 19.4 of the Basic Analysis Procedures Guide.



A Publication for ANSYS Users

How much memory do I need for my run?

It's problem-dependent, but as a rule of thumb, plan on a workspace of 1 GB per million degrees of freedom in the model. For example, a shell model with six degrees of freedom at each node and 100,000 nodes would have 600,000 degrees of freedom and would require 600 MB of workspace. Different solvers require different amounts of memory, so this rule can vary. The PCG solver, for example, normally needs more memory than the sparse solver.

Besides memory, how much disk space do I need for my run?

Again, it's problem-dependent, but as a rule of thumb plan on 10 GB of disk space per million degrees of freedom. That means our 100,000 node shell model would need 6 GB of space on the hard disk to solve. Different solvers require different amounts of disk space, so again the rule can vary. The PCG solver normally requires less disk space than the sparse solver.

When ANSYS runs out of disk space, it typically produces an error such as

```
**** FATAL ***  
Random write error on unit 7, possible full disk  
I/O status error = 9
```

Sometimes the hard disk may be getting full due to scratch files being created during solution, yet after the job has terminated the disk may not be full. This is because some of the scratch files get deleted as the job terminates. If you suspect that your hard disk is filling up, obviously you can try freeing up space. Another option is to use the /ASSIGN command to assign large files to be written to a directory on another hard disk, if possible.

Sometimes this error can occur even if the hard disk is not full. Typically this means that you have reached a maximum file size for your operating system. The /CONFIG,FSPLIT command may be needed to split the file to avoid this limitation. Files size limits vary by operating system and file system, but on Windows 2000, for example, the maximum file size will be at least 4 GB. Detailed information for Windows systems can be found on the Microsoft web site.



A Publication for ANSYS Users

How can I find out how much memory and disk space my model needs?

In addition to the guidelines given above, you can have ANSYS predict resource requirements with the Run Time Statistics. Run-Time Stats are available in the ANSYS Main Menu, or with the RALL command. Keep in mind the statistics provided are estimates and may or may not accurately predict the amount of memory and disk space needed. In most cases, however, the run time stats can give you a good idea of the resources needed to solve your model.

If you have further questions, check the ANSYS documentation. You can also contact PADT's technical support group if you are a PADT Tech Support customer.

The Focus



A Publication for ANSYS Users

Benchmarking ANSYS

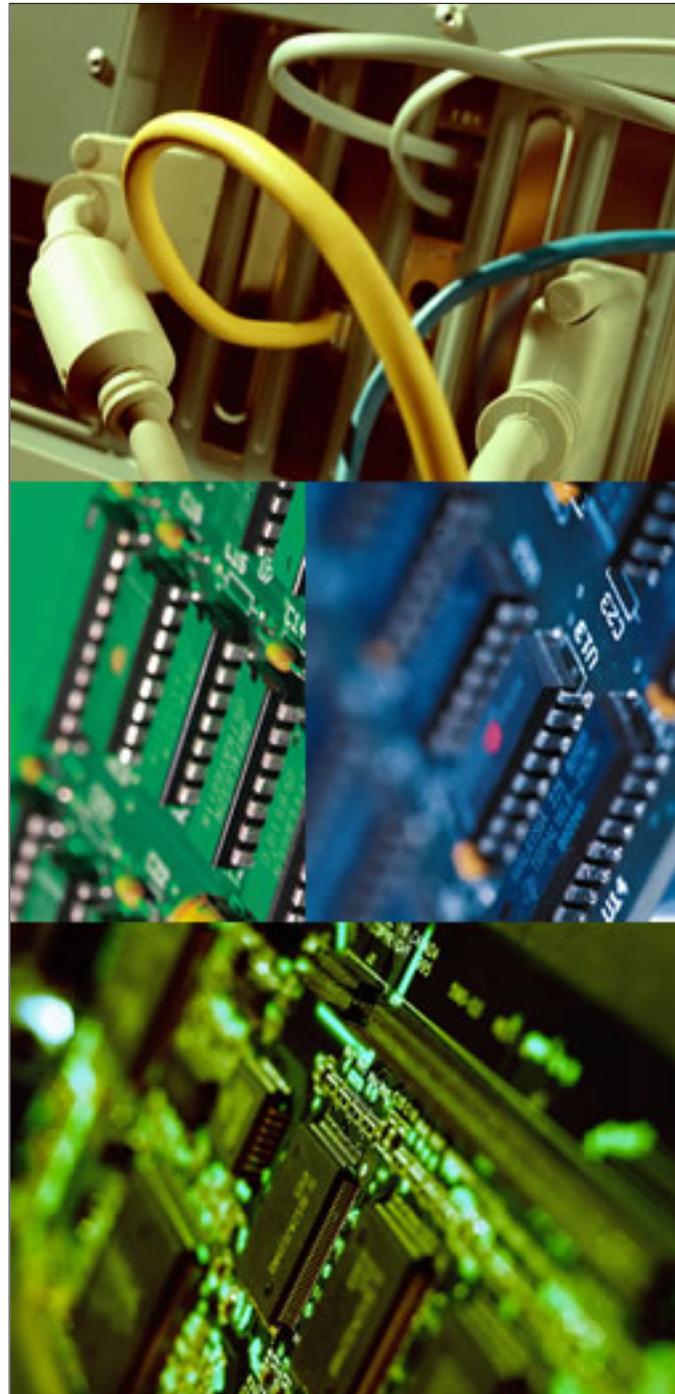
by [Carlos Shultz](#), PADT

How fast does ANSYS run on your machine? Is it time to dump the old name-brand mainframe and get a disposable wintel computer? Is Linux ready for primetime? Does the adage still hold true that you can have any combination of *low price*, *fast speed*, and *stability* in a computer, but not all three?

You won't know the answers to these questions until you run some benchmarks, and then compare the prices and performances of various machines.

A good place to start looking for benchmark results are the many websites that have information regarding benchmarking of hardware. Sites like [Tom's Hardware](#), [Arstechnica](#), [Anandtech](#), [MaximumPC](#), and [Spec](#). If your company is locked into a specific vendor, you should try that vendor's site to see which of their machines they recommend for engineering workstations.

One problem associated with generic benchmarks is that they sometimes fail to predict how





A Publication for ANSYS Users

ANSYS will run with the hardware. A good example of this was the comparison between a GeForce4 Ti 4200 128 MB (\$225) and a GeForce2 MX400 64 MB video card (\$55), both of which were run through identical ANSYS video benchmarks. If you had purchased the more expensive card based upon the OpenGL testing on [Tom's Hardware](#), you would have been expecting 200% more frames per second than the cheaper card. However, when using ANSYS, the GeForce4 card had only a 25% higher frame-rate than the GeForce2 card.

One resource that specifically addresses benchmarking ANSYS are the benchmarks provided by [ANSYS Inc.](#) on their site. In addition, PADT has developed five of its own [benchmarks](#) to test hard drives, CPU, and video subsystems. PADT's benchmarks rely upon the user community to populate the databases, so be sure to try them out and see how your own machine rates.



A Publication for ANSYS Users

Upcoming Training at PADT

A complete schedule of the training courses for the next three months is shown below. [Learn more](#) about how the **Training Services** offered by PADT can save you time and money. Or, feel free to drop an e-mail to our training coordinator, [Ted Harris](#)!

Month	Start	End	Course	Description	Location
Dec '03	12/15	12/16	801	Introduction to APDL	Irvine, CA
	12/17	12/19	201	Basic Structural Nonlinearities	Tempe, AZ
Jan '04	1/8	1/9	104	ANSYS Workbench Simulation - Intro	Irvine, CA
	1/12	1/14	101	Introduction to ANSYS, Part I	Tempe, AZ
Feb '04	1/22	1/23	801	Introduction to APDL	Irvine, CA
	1/26	1/28	201	Basic Structural Nonlinearities	Irvine, CA
	2/2	2/4	101	Introduction to ANSYS, Part I	Irvine, CA
	2/5	2/6	202	Advanced Structural Nonlinearities	Tempe, AZ
	2/9	2/11	902	Multiphysics Simulation for MEMS	Albuquerque, NM
	2/17	2/18	104	ANSYS Workbench Simulation - Intro	Tempe, AZ
	2/19	2/20	301	Heat Transfer	Irvine, CA
Mar '04	2/26	2/27	102	Introduction to ANSYS, Part II	Irvine, CA
	3/1	3/3	101	Introduction to ANSYS, Part I	Tempe, AZ
	3/8	3/9	203	Dynamics	Tempe, AZ

Whether it s one of our regularly-scheduled classes at our training facility, or a customized class tailored to your specifications and location, you can be assured that the training you receive will have immediate, positive results on your capabilities in design, analysis, and product and process improvement.





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.