

The Focus



A Publication for ANSYS Users

Contents

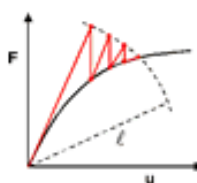
Feature Articles

- [Reasons to Upgrade to 7.0 / 7.1](#)
- [Buckling and the Arc-Length Method](#)
- [Getting the Most from Your ANSYS License](#)



On the Web

- [PADT Web Resources](#)
- [ANSYS.NET: Sheldon's Site](#)



Resources

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

ANSYS
Support Distributor

© 2002, by Phoenix Analysis & Design Technologies, Inc. All rights reserved.

The Focus



A Publication for ANSYS Users

Reasons to Upgrade to 7.0 / 7.1

by [Rod Scholl](#)



Before you launch a counter-strike...

I am not foolish enough to make a global recommendation to upgrade to 7.0. However, I am willing to highlight some features that may make the upgrade worthwhile.

I also won't go into the bugs and problems in 7.0, but if you keep up with XANSYS traffic, you've probably heard of many. I found it funny that the command K,p (perhaps the most basic command in ANSYS) fails!

Even still, I use 7.0 daily with little difficulty. I expect the majority of bugs (which are primarily usability, and not accuracy bugs) will be fixed by 7.1.

What's New

You can find the full descriptions and list at the start of the help files for 7.0, or on the ANSYS web site. This is an arbitrarily truncated listing skewed towards my structural/thermal analysis thinking. The highlighted topics will be discussed in this article.

● Structural

- [FEMXplorer](#) Response Surface based on CADOE technology
- Point / Edge Contact Element 175 (better than elements 26, 48, 49)
- [Faster Contact Convergence / Improvements](#)
- [MPC Rigid Link / Beam Element](#)
- New Hyperelastic Material Models
- Cyclic Symmetry Improvements / Traveling Wave animations!
- [Rotational Loading](#) for Individual Components

● Thermal

- [Radiosity Speed Enhancement](#) / Storage of View Factor Matrix
- [Layered Thermal Shell](#)

● Usability

- Component Selection Ability
- [Geometry Healer](#) available through the GUI

The Focus



A Publication for ANSYS Users

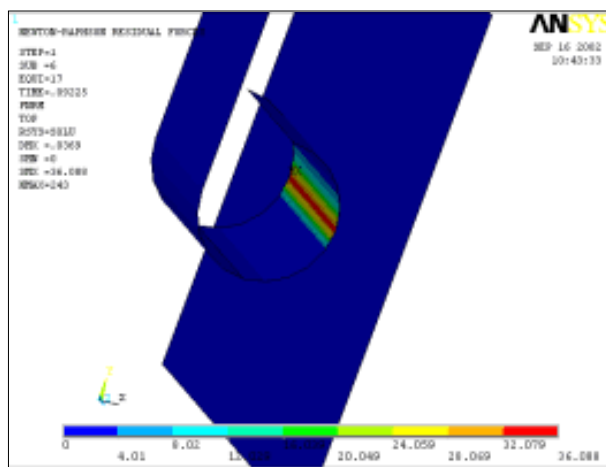
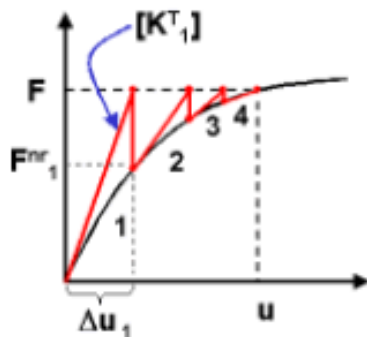
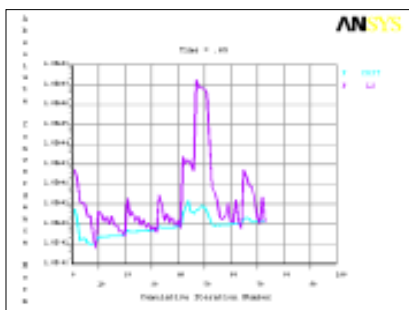
- **Other new features**
 - DDS Solver now support CP s and CE s
 - [Plotting Non-Linear Convergence Residuals](#)
- **Installation, licensing, and configuration**
 - Catia V5 connection product
 - Installation directory structure makes a little more sense
 - On windows, now some systems can access more than 2GB
- **ANSYS commands**
 - CYCGEN and CYCSOL are no longer supported

Geometry Healer

The Automatic Geometry Healing Module (AGH) is now accessible through the menus under File > Import > Heal. Previously you could only access it by typing ~HEAL. Learn more about the healer from a [previous article](#) in *The Focus*.

Plotting Non-Linear Convergence Residuals

You can now plot the region of high residuals during a non-linear run using NRRES. This will be a huge benefit when trying to determine what region of your model is having the most trouble converging.



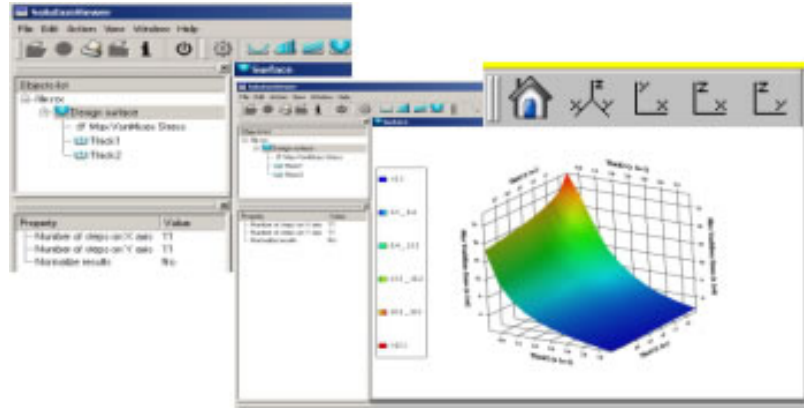
The Focus



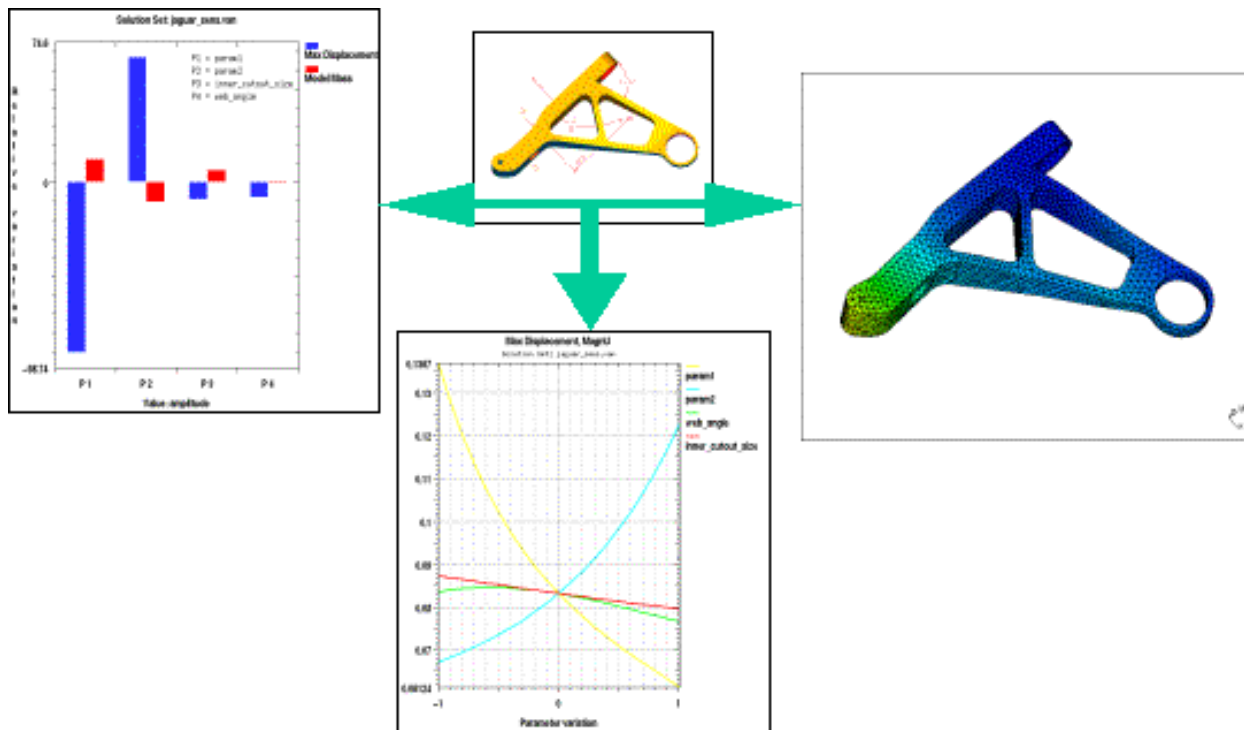
A Publication for ANSYS Users

FEMXplorer Response Surface Based on CADOE Technology

The Good News: This amazing new technology developed by [CADOE](#) allows the generation of a response surface to design variables with a SINGLE solve.



Naturally, this is fantastic for optimization, and cost/benefit analysis. This is the future for linear analysis. By fitting a Taylor series on changes to the stiffness matrix as a result to changes in input variables, the parameter terrain is perfectly knowable for all linear runs with a single matrix inversion. Throw away the DOE!!!



The Bad News: They have not included changes to geometry yet, unless it is captured in a real constant. It is coming soon, so you might want to start the learning curve now.

The Focus



A Publication for ANSYS Users

Also, this approach is not valid for non-linear runs. Perhaps someday I don't know if someone has a strategy for stiffness matrixes that don't vary linearly with input parameters, but I can't think of anything off the top of my head. Whatever that's worth, given that I didn't think of using this approach either&

Contact Improvements

The improvements to contact are terrific. You can plot the number of times an element changes status in a solution (CNOS) to detect chattering or determine the region of difficult convergence. Perhaps you might change your FKN for that region only&

Also take advantage of Key Option 10 = 1. A recent [case study](#) in *The Focus* shows it to be an order of magnitude faster in getting contact convergence. There are several other improvements worth reading about in the help files next time you are struggling with contact convergence.

MPC Rigid Link / Beam Element

Now one can add a Nastran-like rigid link between nodes. This is superior to a CE/CP because one can still apply BC's to the linked nodes. Also, the deformed shape is captured for a Large Deflection (NLGEOM,on) analysis.

Multiple Rotation Loads

Now one can apply different rotation values to different entities by assigning different rotations to different components. Also the axes of revolution can be defined in any coordinate direction!

Radiosity Speed Enhancement

The radiosity solver is more robust. Also, one can store the view factors in a binary file which is faster. One can still elect the ASCII format, so they can be queried.

Layered Thermal Shell

The new thermal element supports through thickness variation in temperature (up to 32 layers). This should greatly streamline modeling circuit boards, and other such structures. Learn more in the [recent article](#) in *The Focus*.

The Focus



A Publication for ANSYS Users

Making the Switch to 7.0

As with any release, no one should make the change unless the improvement to usability and new technology saves time/money. I usually find (even for the buggier versions) that if there is some new technology I can use, it outweighs the 10 minutes it takes to learn workarounds to odd GUI behavior, broken/missing buttons, etc. This of course does not preclude me from belly-aching about them.

Good luck, and if you want to learn more about 7.0 new features, there s this guy I know name Manuel, he can probably help you out!



Buckling and the Arc-Length Method

by [Jeff Plastic Strain](#), ANSYS Supporter and User

One major characteristic of nonlinear buckling, as opposed to eigenvalue buckling, is that nonlinear buckling phenomenon includes a region of instability in the post-buckling region whereas eigenvalue buckling only involves linear, pre-buckling behavior up to the bifurcation (critical loading) point. See Figure 1 for an illustration.

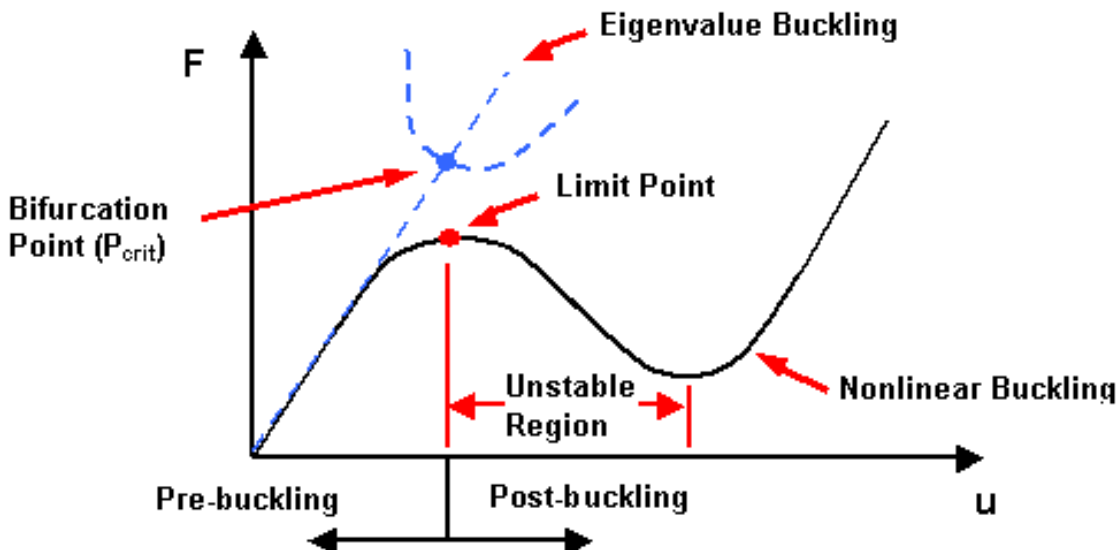


Figure 1. Nonlinear vs. Eigenvalue Buckling Behavior

The unstable region above is also known as the snap through region, where the structure snaps through from one stable region to another. To illustrate, consider the shallow arch loaded as shown in Figure 2.

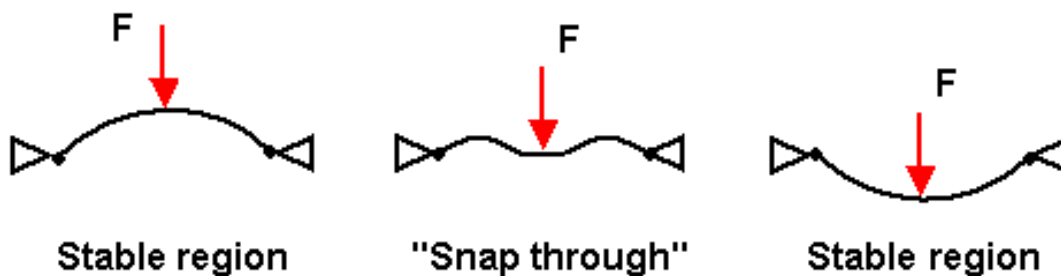


Figure 2. Snap Through Buckling

For most nonlinear analyses, the Newton-Raphson method is used to converge the solution at each time step along the force deflection curve. The Newton-Raphson method works by iterating the equation $[KT]\{u\} = \{F_a\} - \{F_{nr}\}$, where $\{F_a\}$ is the applied load vector and $\{F_{nr}\}$ is the internal load vector, until the residual, $\{F_a\} - \{F_{nr}\}$, falls within a certain convergence criterion. The Newton-Raphson method increments the load a finite amount at each substep and keeps that load fixed throughout the equilibrium iterations.

Because of this, it cannot converge if the tangent stiffness (the slope of the force-deflection curve at

The Focus



A Publication for ANSYS Users

any point) is zero. See Figure 3.



Figure 3. Newton-Raphson Method

To avoid this problem, one should use the arc-length method for solving nonlinear post-buckling. To handle zero and negative tangent stiffnesses, the arc-length multiplies the incremental load by a load factor, λ , where λ is between -1 and 1. This addition introduces an extra unknown, altering the equilibrium equation slightly to $[KT]\{u\} = \lambda\{F_a\} - \{F_{nr}\}$. To deal with this, the arc-length method imposes another constraint, stating that

$$\sqrt{\Delta u_n^2 + \lambda^2} = \ell$$

throughout a given time step, where ℓ is the arc-length radius. Figure 4 illustrates this process.

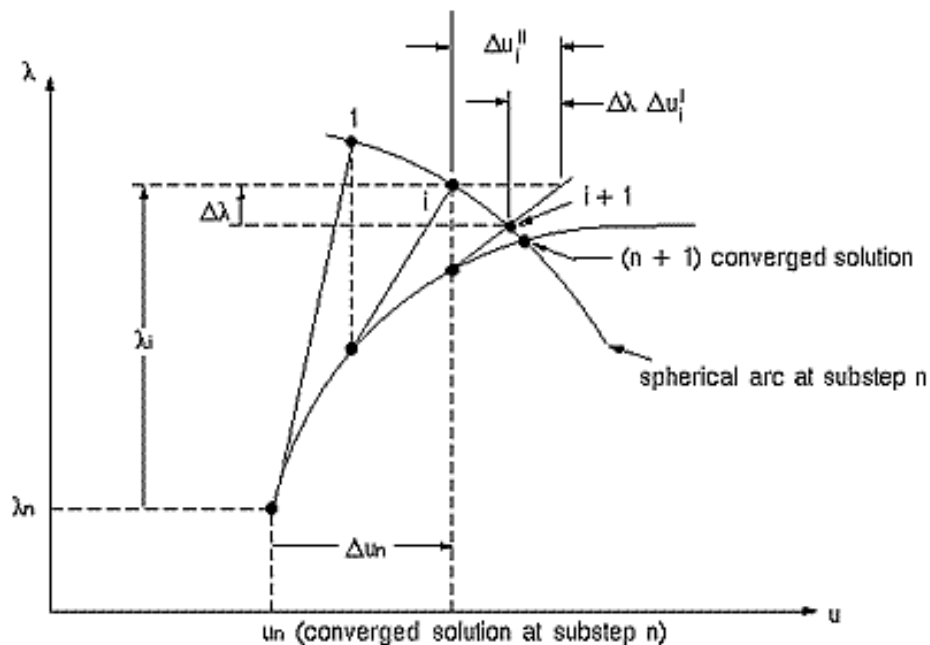


Figure 4. Arc-Length Methodology

The Focus



A Publication for ANSYS Users

The arc-length method therefore allows the load and displacement to vary throughout the time step as shown in Figure 5.

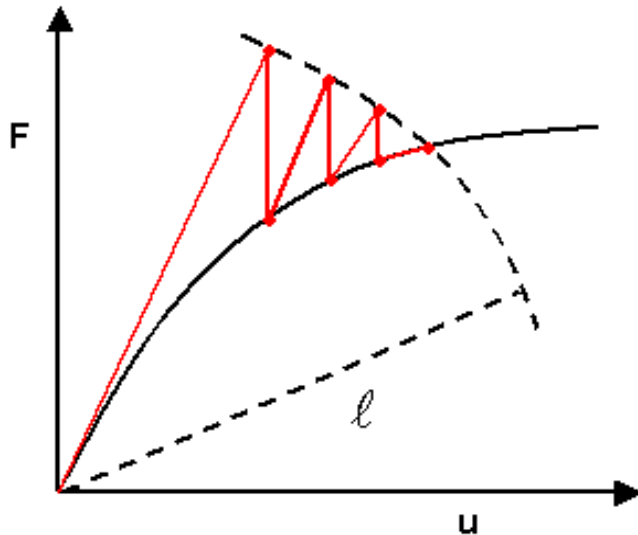


Figure 5. Arc-Length Convergence Behavior

To use the arc-length method in ANSYS, execute the command `ARCLEN,ON,MAXARC,MINARC` where *MAXARC* is the maximum allowable multiplier of the reference arc-length radius and *MINARC* is the minimum allowable multiplier. *MAXARC* defaults to 10 and *MINARC* defaults to 0.001. The reference arc-length radius is calculated as the total load divided by the number of substeps specified by the `NSUBST` command. In ANSYS 7.0, the menu pick for activating the arc-length method is Solution > Analysis Type > Sol'n Controls > Advanced NL.

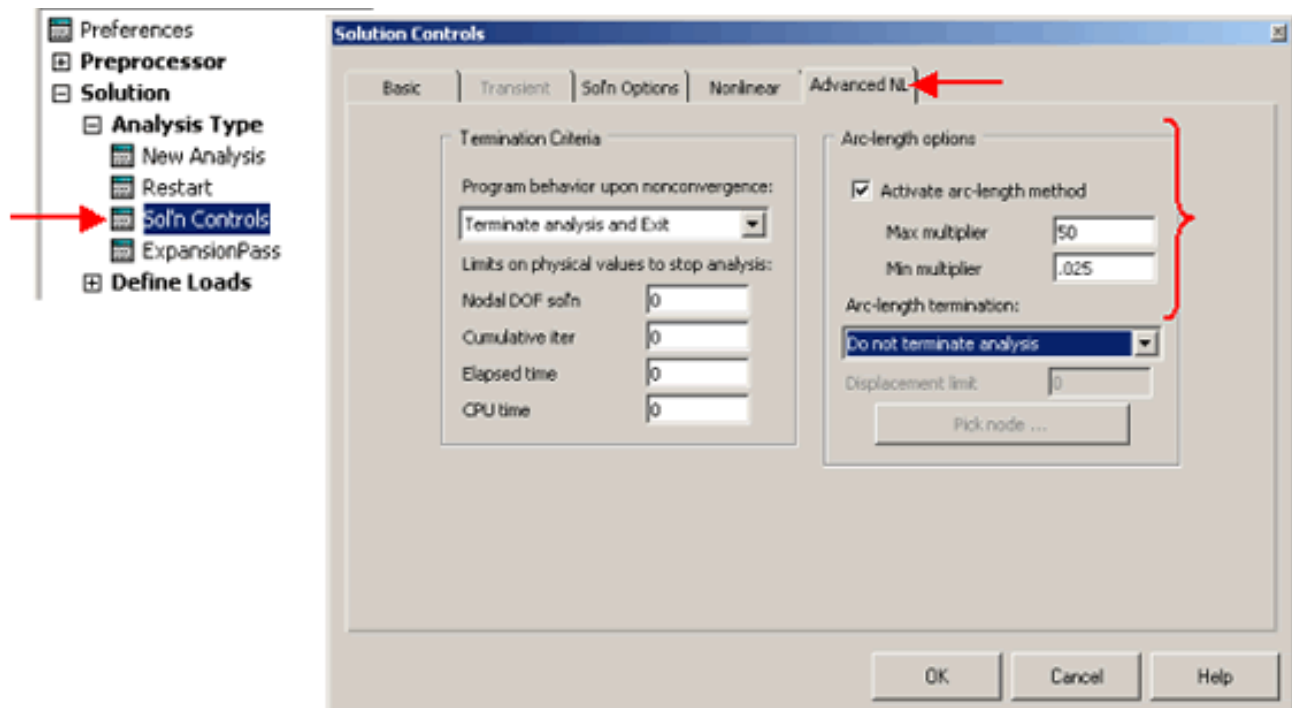


Figure 6. Arc-Length Method Menu Picks

The Focus



A Publication for ANSYS Users

ANSYS varies the arc-length radius to fall between within the range specified by MINARC and MAXARC based on the solution behavior and arc-length radius of the previous substep. This is a sort of auto-timestepping, which means that settings specified by the commands AUTOTS, LNSRCH (line search), and PRED (DOF solution predictor) are ignored when the arc-length method is activated.

The Focus



A Publication for ANSYS Users

Getting the Most from Your ANSYS License

by [Ted Harris](#), Lead Engineer Tech Support and Training

Many people may think of training as just for people new to ANSYS. Indeed we do train a lot of new users, but we also offer a full set of classes for those who are perhaps intermediate or advanced users. Our goal is to help you fully utilize your ANSYS licenses to help you create the best possible products and processes. Please allow us to introduce our training offerings, for all levels of ANSYS expertise.



We think training is great because we get to know the students and the problems they are working on, and we love helping them to become more proficient and productive in getting their jobs done. It allows our students to get farther and faster up the learning curve in two or three days of compressed, relevant learning than they could accomplish on their own. Further, the students get to know us and can then put faces to the names in cases when we provide them with subsequent engineering services or ANSYS technical support. Our students are overwhelmingly positive about their training experiences at PADT, as the following comments from our surveys attest:

- The instructor was knowledgeable, patient, and easy to understand. Exercises were also relevant to understanding the code.
- Excellent course, highly recommended. Will definitely be back for more advanced training.
- I liked the workshops and the lecture was very well done. Good teacher.

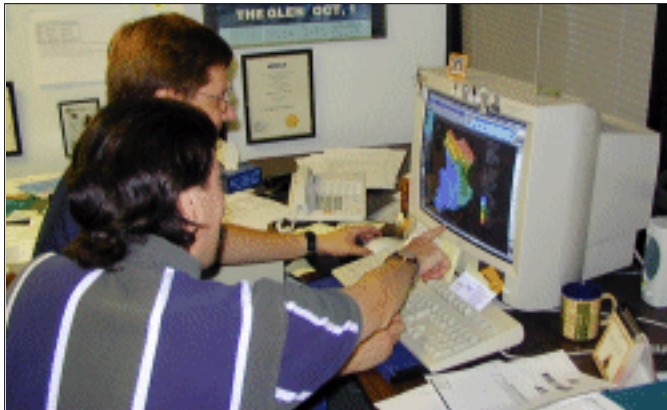
The Focus



A Publication for ANSYS Users

- The class was well run. Good computers, good site, good instructor, good balance between lectures and workshops.
- Great class, good hands-on, great instructor who is willing to discuss beyond the class scope.
- The training was great. It was money well spent that will pay dividends to my company.
- Great class. I can't wait to use this.
- Taught by very experienced and knowledgeable instructors with a lot of industrial background.
- The instructor asked questions about specifics [of what we do at our company] and customized the discussion to help us at our jobs.

Although we have regularly scheduled classes in Arizona and California throughout the year, PADT can also offer custom classes, with tailored content and schedules to meet your needs. We perform training at our facilities or at customer sites. [Details](#) on all our standard course offerings can be seen on our web site, but short introductions to each of our 2003 offerings are shown below as well.



For People Starting Out With ANSYS

Engineering with Finite Element Analysis. This class is independent of the finite element tool used, but covers the how to of using FEA. Among the topics covered are how to represent your real world problem in finite elements, how to simulate bolt preloads, press fits, etc., how to interpret results and make design decisions, and how to evaluate structures to ASME code requirements.

The Focus



A Publication for ANSYS Users

Introduction to ANSYS, Part I. This is the most popular class for new users. It covers how to get into ANSYS and move around with the GUI, how to import and create geometry, how to mesh and apply loads, how to solve linear structural and thermal problems, and how to post-process. It also gives an introduction to scalar parameters, toolbar abbreviations, and the automatic report generator, among other topics.

Introduction to ANSYS, Part II. This is a continuation of the Intro. Part I class. Its material includes array parameters, DOF coupling and constraint equations, beam elements, use of the element table for postprocessing, basic nonlinear modeling, and introduction to macro writing. Also included is the use of submodeling to build a refined local model using the DOF and temperature results from a larger, coarse model.

Introduction to ANSYS for MEMS. This is a compressed intro class that focuses on Micro-Electrical Mechanical Systems applications.

PADT also offers a one-day introduction to [Designspace](#), the lower-cost, designer level analysis package from ANSYS, Inc.

Intermediate/Advanced Classes

The remainder of PADT's course offerings is for intermediate to advanced ANSYS users. Even the most seasoned ANSYS user can learn something useful in one of our classes. Often those of us who have been using the tool for 15 years or more discover things we didn't know as we go through some of the classes and answer questions from the students. Most of the classes below require some prior ANSYS experience, such as attending an Intro. course or having on-the-job experience. We'll be happy to answer any questions you may have about any the classes.

Basic Structural Nonlinearities, Advanced Contact, and Bolt Pretension. Includes material nonlinearities, rate-independent plasticity, and contact.

Advanced Structural Nonlinearities. Includes 18X elements, rate-dependent plasticity, creep, viscoplasticity, viscoelasticity, hyperelasticity, concrete, buckling, and element birth and death.

Dynamics. Includes modal, transient, harmonic response, spectrum, random vibration, prestressed modal, and cyclic symmetry analyses. Also includes damping.

The Focus



A Publication for ANSYS Users

Heat Transfer. Includes conduction, convection, radiation, pipe elements, steady-state and transient analyses, and phase change.

Introduction to Electromagnetics. Includes 2D and 3D static, AC, and transient simulations of electromechanical devices such as armatures/stators. Includes coupled-field simulations.

High Frequency Electromagnetics. New! Includes resonance, propagating, radiation, and scattering in 3D as well as 2D modal analysis for waveguides/transmission lines. Also includes Q factor and S parameter calculations, and near and far electromagnetic fields.

Multiphysics Simulation for MEMS. New! Includes MEMS units, geometric nonlinearities, initial stress, contact, and prestressed modal analyses. Covers electrostatic analyses including capacitance, Trefftz domain for field effects in open domains, sequential and direct methods of electrostatic/structural coupling, transient analyses, reduced order modeling, piezoelectric analysis, thermal/electric coupling, thermal/structural coupling, and CFD for reduced order damping modeling.

ANSYS/LS-DYNA. Covers this explicit code for handling large deformation, nonlinear materials, and high frequency response, with or without contact.

Introduction to CFD with FLOTRAN. Includes laminar and turbulent fluid flow, convection and conjugate heat transfer, and fluid/structure coupling.

CFD with FLOTRAN, Part II. Includes more theory, analyzing non-Newtonian fluids, distributed resistances, turbulence, compressible flow, multiple species, and free surfaces.

Design Optimization and Probabilistic Design. Includes the concept of the parametric analysis file, design optimization usage, and probabilistic design to assess the effect of variable inputs on results.

Introduction to APDL. Includes macro programming, customized ANSYS subroutines, GUI customization, parameters and *GET command, interfacing with users. Also includes arrays, tables and vector operations, and user-programmable features.

Advanced APDL and Customization. Includes vectors, strings, and undocumented commands, user interface design language, introduction to Tcl/Tk GUI language, binary file I/O, and creating tilde commands (DLL s).



A Publication for ANSYS Users

Tcl/Tk for ANSYS. Includes background on Tcl/Tk GUI scripting language, creating GUI s, adding features to and modifying the ANSYS GUI.

For More Information

To see our detailed [class listing](#) and [schedule](#), or to register for a class, please see our [web site](#). You can also [e-mail](#) or call the author at 480-813-4884 for more information or to discuss customized training possibilities. We look forward to working with you to help you get the most out of your ANSYS investment.

The Focus



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.