The Focus

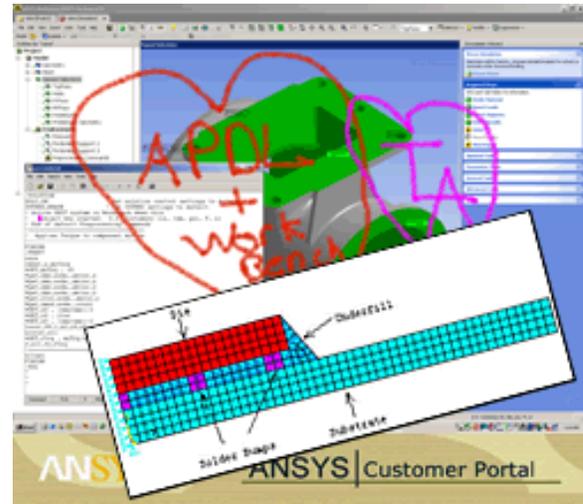*A Publication for ANSYS Users*

Contents

Feature Articles

- [Leveraging the Advantages of Workbench: Using APDL with Workbench](#)
- [Using Element Birth and Death](#)
- [The ANSYS Customer Portal](#)

On the Web

- [Programming with Tcl/Tk in ANSYS](#)
- [PADT's Power Generation Solutions](#)

Resources

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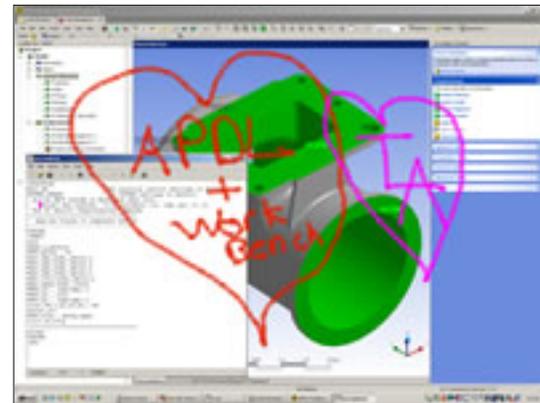
The Focus

Leveraging the Advantages of Workbench:
Using APDL with Workbench

by Eric Miller, PADT

Overview

Over the past several releases, ANSYS Inc. has been rolling out more and more of their Workbench technology. There is already a lot of marketing and technical information available on the various aspects of Workbench, but PADT is still often asked: Why should I use Workbench? or How can Workbench help me? We attempt to answer these questions in this ongoing series of articles, which describe how to use these new tools to your advantage.

Once users start to get an understanding of what Workbench can do, the next comment is usually: That s great, but it can t do this or that. I wish I could use APDL with Workbench. The development team at ANSYS Inc. came to this same conclusion, and so they have provided some nice tools for using APDL with the Workbench in a very powerful way.

Have Your Cake and Eat It, Too

The technologies that make up the Workbench offer some significant advantages over more traditional FE pre/post processors, but they may lack some critical capabilities that users would like to have access to. ANSYS has a huge wealth of capabilities but does not offer the advantages of Workbench. Once again APDL (ANSYS Parametric Design Language) steps into the breach to solve the problem.

This is possible because the way that the Workbench works under the hood. In order to talk to the solvers in ANSYS, the program actually creates a batch APDL script that ANSYS solves, and then converts the answers into a format

The Focus

A Publication for ANSYS Users

Workbench can understand. Users have access to this file in either of two ways:

1. Go into Control Panel > Preferences > Solution and set Save ANSYS Files to Yes . A file called jobname.inp will be saved in your solution directory.

2. Click on the Solution branch of your model then select Write ANSYS Input File& from the Tools menu.

To understand what is and is not in this file, take a simple Workbench model, save the file, and view it in the editor. As you can see, there is no geometry, just nodes, elements, loads, and solver commands. If you use the second option a /eof is placed in the file just before the solve commands. Note how the code asks for all sorts of useful bits of information for Workbench, as well as how that information and the results are written to Workbench with the /xml command. Also note that Named Selections from the Workbench are converted into Components in ANSYS. This is a very powerful aspect of how the two programs work together, and it enables a large amount of sophisticated automation.

Some users simply use the Workbench as much as they can, then write out this file as a starting point for an APDL-based analysis. More experienced users will also use this file to debug a run that is not working correctly. This is fine for many circumstances but if geometry modifications or parameter studies are needed, Workbench offers a better alternative.
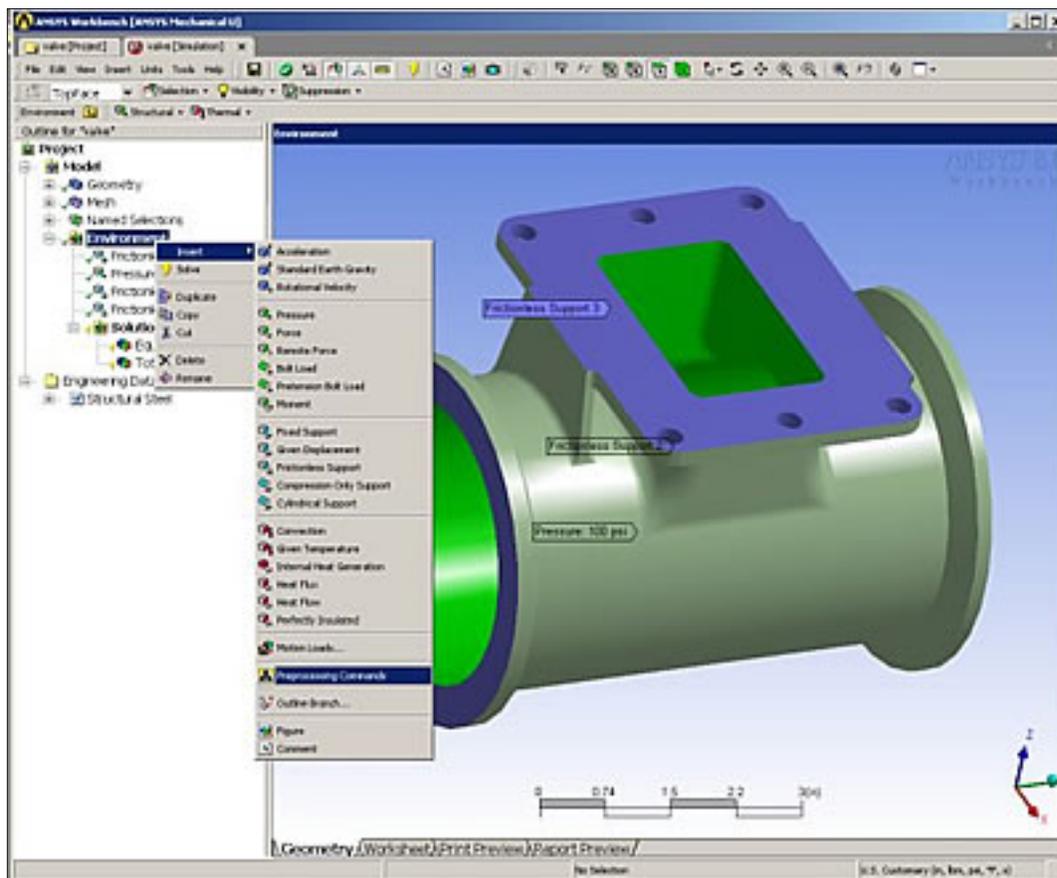
The Focus

PADT

A Publication for ANSYS Users

Figure 1. Inserting preprocessing commands object.

A Snippet Is a Fine Thing

To embed APDL into your studies, users can create objects called Preprocessing
Commands and Postprocessing Commands in the GUI. Right-click on the
appropriate model branch and insert the object and now APDL is a part of your
Workbench model. You now can insert APDL code, called snippets, to carry out
more complex tasks in batch during the solve. You enter your commands by
clicking on the Worksheet tab of the graphics window while the object is
selected. Figure 1 shows the menu pick to insert the Preprocessing Commands
object, and Figure 2 shows a simple APDL macro for applying a torque on the
end of the sample part. Almost all APDL snippets will be similar to   but often
more complex than   this example.

```
/SOLUTION
SOLC,ON      ! Set solution control settings to default
OUTRES,ERASE ! Set OUTRES settings to default
! Active UNIT system in Workbench when this object was
```

The Focus

PADT

A Publication for ANSYS Users

```
! created: U.S. Customary (in, lbm, psi, F, s)
! End of default Preprocessing commands
!======================================
! Applies Torque to component myFace
!----------------------------------
FINISH
/PREP7
csys
cmsel,s,myface
*SET,myTrq , 25
*get,xmx,node,,mxloc,x
*get,xmn,node,,mnloc,x
*get,zmx,node,,mxloc,z
*get,zmn,node,,mnloc,z
*get,yloc,node,,mnloc,y
```

Figure 2. Sample APDL snippet, torque on a face

As the example shows, it is easy to grab nodes or elements if you pre-name them as Named Selections . MYFACE was used in this example to identify the end nodes. This naming will carry through even if the model is remeshed, so you should make your snippets general enough that they do not assume a fixed number of entities.

Two things in the example should be noted. First, Workbench puts you in /SOLUTION, so if you need to go into /PREP7, make sure you return to /SOLUTION when your code is finished. Second, make sure you do an ALLSEL at the end of the Preprocessing Commands . Forgetting these two commands is a very common source of error.

Once these commands are saved in your model, they become part of the normal solution process. PADT and others have used this capability to set up complex loads, time-dependent runs, non-linear material properties, and advanced dynamic analyses such as PSD. One of the more productive applications is to automate the setup and postprocessing of complex analysis that are to be conducted frequently. Just as a macro can be used in ANSYS to automate the process, APDL can be embedded in a Workbench template for the same benefits.

If you are not up to writing your own code, you can create some basic snippets with the Preprocessor Command Builder. This is a limited version of the ANSYS GUI that lets you specify non-linear materials and complex solutions using the material and solution wizards from ANSYS.

Not Just for the Front End

Although most people see value in APDL snippets for preprocessing, you can also access advanced postprocessing the same way. Standard POST1 and POST26 commands work in the postprocessing object. Users can save in standard ANSYS formats such as .jpg and .png by using the proper /SHOW commands.

In addition, you can right-click on the Postprocessing Command and choose Postprocessing Command Builder. This brings up the Result Viewer from ANSYS, which gives you full access to POST1 and POST26 commands through a simplified GUI, and saves your actions as commands in the postprocessing object. This is the preferred way to do more advanced viewing of your results from within the Workbench, as well as a way to build up postprocessing snippets.
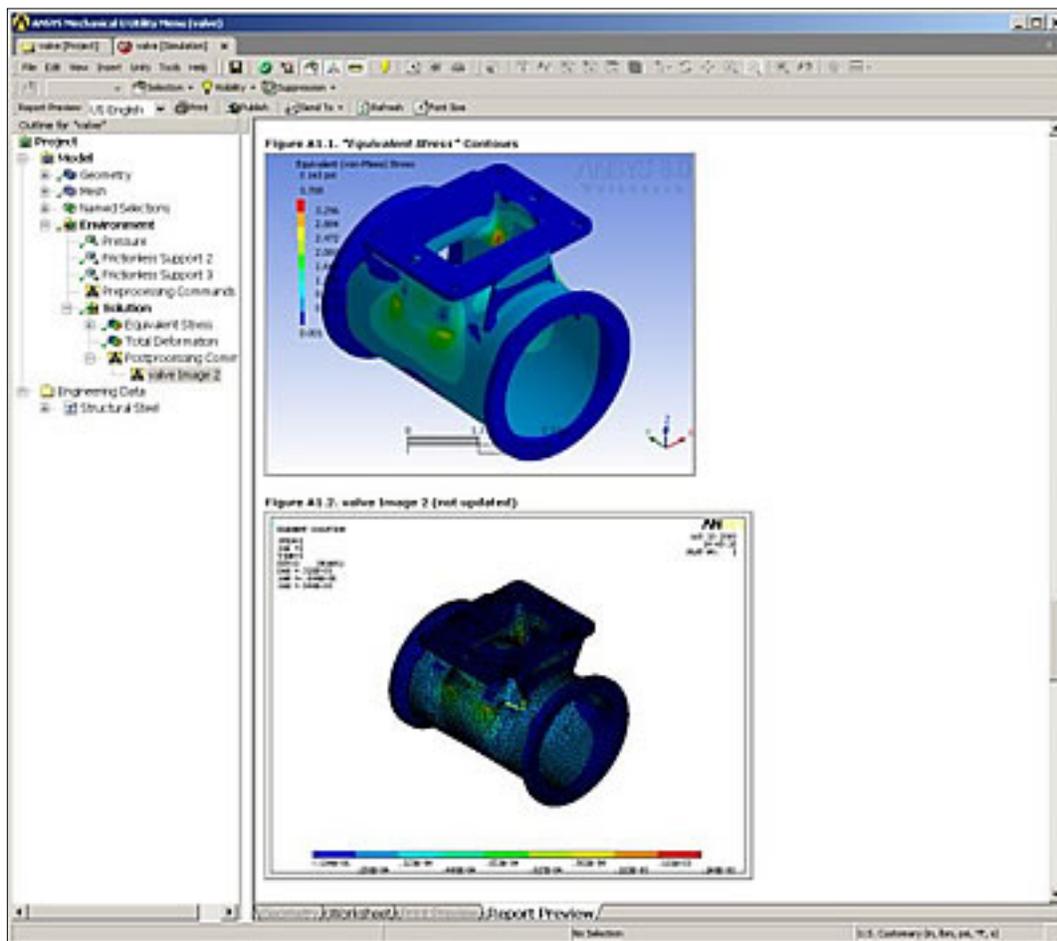

Figure 3. Workbench and ANSYS plots peacefully coexisting in a Workbench report.

The Focus

If you are using the automatic report generation capabilities in the Workbench, you can still use APDL to generate the plots. Figure 3 shows the example model with Workbench and ANSYS plots side by side. These types of plots require that you use standard APDL as well as embedding the Tcl/Tk ~eui, ansys::report::* objects into the snippets. Figure 4 shows the code used to produce the plot in Figure 3. The best way to understand these commands is to let ANSYS put them in your snippet by using the Report Image Capture capability in the Result Viewer .

```
/POST1
SET,FIRST
/PLOPTS,INFO,3
/CONTOUR,ALL,18
/PNUM,MAT,1
/NUMBER,1
/REPLOT,RESIZE
/view,1,1,1,1
/type,1,4
/vup,1,z
/repl
~eui,'::ansys::report::sizeGraphics down'
/REPLOT,RESIZE
~eui,'::ansys::report::setOptions reverseVideo 1'
PLESOL,EPTO,1
~eui,'ansys::report::imagecapture {valve Image 2} '
~eui,'::ansys::report::finished'
/REPLOT,RESIZE
/REPLOT,RESIZE
/REPLOT,RESIZE
! /UIS,ABORT,1
~eui,'::ansys::report::finished'
```

Figure 4. Example postprocessing commands.

Lessons Learned for Effective Snippiting

The most important piece of advice for using APDL in the Workbench is one that you will hear often from PADT: *keep it simple*! It is very easy to confuse yourself,

A Publication for ANSYS Users

Workbench, or ANSYS, if things get too complex. Experience has shown that when things start to get confusing, it is better to save the model as an APDL input file and to carry out the complex tasks in ANSYS directly. However, if things are kept simple and general, significant productivity gains can be achieved using APDL snippets.

The only thing that is really frustrating about creating these snippets is that there is no immediate feedback. To get around this, PADT tends to write the APDL as macros in an ANSYS session that starts with reading in the Workbench model. This allows the user to iteratively create the macro. Once completed, it can then be pasted into the Preprocessing Commands object in the Workbench.

A last piece of advice is to lean heavily on Named Selections/Components. It does not take long to conclude that it is very useful to have this capability. PADT has found that by using components, very complex macros can be written and applied in a general manner.

Full Power of ANSYS from Workbench

As the Workbench evolves more and more, functionality that was only available in ANSYS will be made available with the new tools. However, if some functionality is currently missing, it doesn t necessarily mean that Workbench cannot be used. By embedding powerful and flexible APDL snippets into the Workbench model, users can access the full power of ANSYS while still keeping the significant advantages of the Workbench.

As has been stated in all of the previous articles in this series: users should try out this functionality for themselves to understand the true power. To help with this particular capability, we have posted a zipped version of the model   with the mesh and results removed for size   on our web site. Just click the above link to download it. The file also contains a good example of using DesignModeler (see article in last issue of *The Focus*).

This is cool stuff that can be very useful. Give it a try!

The Focus

Using Element Birth and Death

by [Carlos Shultz](), PADT

Element birth and death, using the ealive and ekill commands, allows the user to deactivate and reactivate elements throughout an analysis.

A typical use of this feature is to model assembly procedures, adding materials to the model at various loadsteps. Figure 1 and Figure 2 show a typical before and after view of live elements in a chip package assembly simulation. The additional materials come in without a strain history. Birth and death are also used to simulate more complex physics, like phase change, annealing, and material failure.
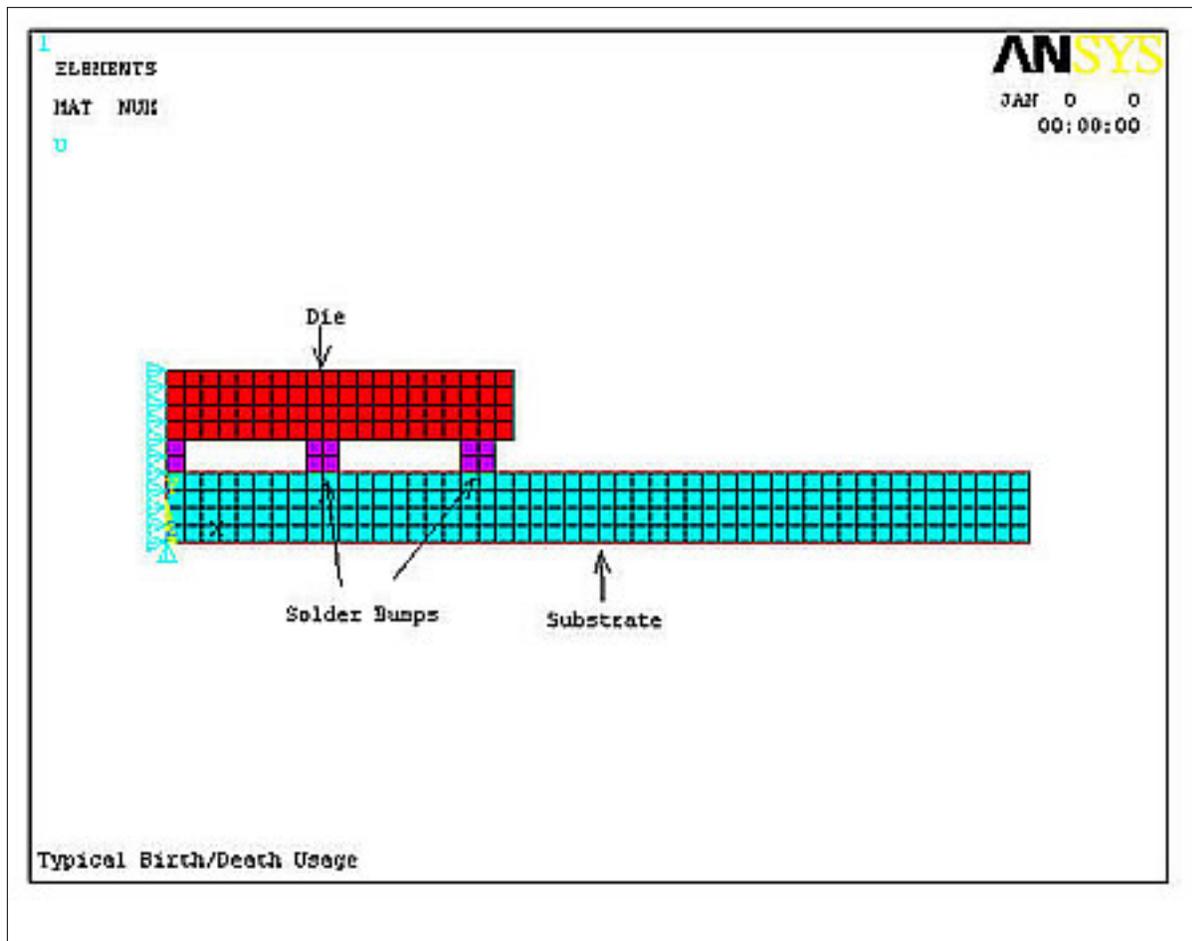

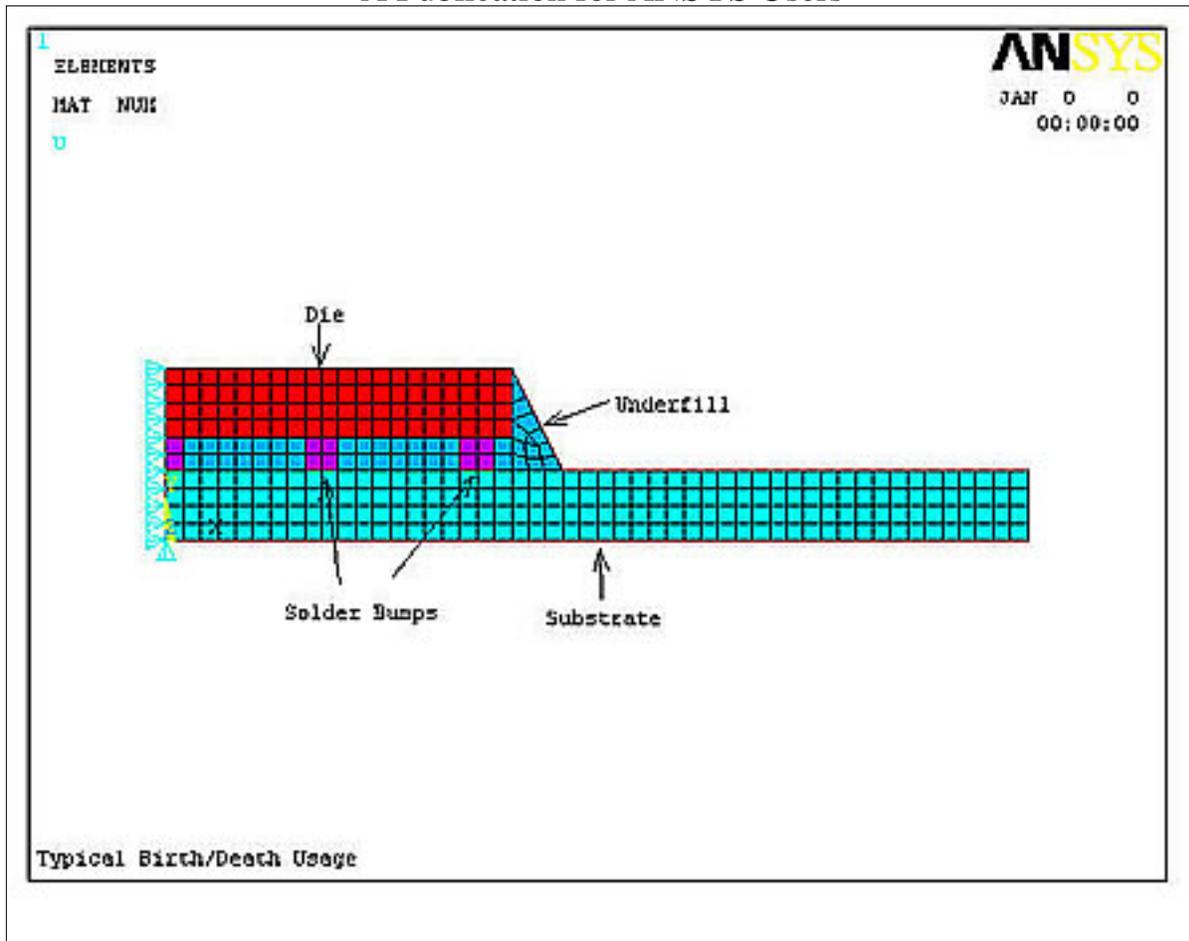Figure 1. Die attached to substrate during loadstep 1.

The Focus

Figure 2. Underfill activated using ealive during loadstep 2

Make sure nlgeom is on to ensure that the elements are located properly. If the user intends to add material without thermal strain, set the material s reference temperature equal to the temperature of the activation loadstep. Applying constraints to the unused nodes in the model (nodes attached to only deactivated elements) will prevent unwanted deflections of the elements when they are later made alive.

The deactivation of elements is done by reducing their contributions to stiffness matrices significantly and removing their contributions to load vectors completely. An element's strain is set to zero as soon as that element is killed.

Select the elements that are alive at any given time using Esel,s,live, or killed by inverting the set Esel,s,live $ Esel,inverse. However, this doesn t relieve the user of having to keep track of birth/death usage, because ANSYS does not store the status in the .rst file.

Make sure to unselect killed elements and unused nodes and to turn powergraphics off when plotting nodal results. Otherwise, the results will improperly average at the boundaries between the live and dead elements.

A Publication for ANSYS Users

The ANSYS Customer Portal

by Ted Harris, PADT

If you are a technical support (TECS) paying, professional (non-academic) ANSYS customer, you can sign up to have access to the ANSYS Customer Portal on the ANSYS web site. Among the items you can access on the Customer Portal are online Class 3 error reports and QA notices, service packs, beta testing information, and the ANSYS Knowledge Base.

To establish a login, you ll need to know your customer number, which you can access from the ANSYS GUI by clicking on Help > About ANSYS. This works for ANSYS 7.0 and later. If you can t find your customer number, contact the ANSYS Support Coordinator (ASC) at your site, or contact your own ANSYS support provider.

Once you re ready to create your login, go to the ANSYS web site at www.ansys.com. Next, click on the MY ANSYS link near the top of the page. To create your login the first time, click on Simply follow this link to register. You ll be asked for the typical name, address, company, etc. information as well as your customer number to complete your request. If the login is not created after entering your information, there should be a link to click on for assistance.

Logins for university customers may be restricted, so it s best to contact your ANSYS Support Distributor for your status.

Once your login is established, you ll be allowed to access the customer portal from the ANSYS web site by clicking on the MY ANSYS link and by following the appropriate links to log in.

As a TECS-paying customer you can now access a variety of information on the Customer Portal. One important link is to the ANSYS Knowledge Base, which has a variety of technical questions, answers, and solutions for common and sometimes not-so-common problems. You ll also be able to search the Class 3 Error reports and QA notices. In addition, ANSYS service packs are downloadable from the customer portal. There are also some example input files for MEMS applications, ANSYS/LS-DYNA, etc., including some with Powerpoint explanations. There are other links as well, such as to the Technotes informational bulletins.

The ANSYS Customer Portal is one of the benefits customers receive by continuing to pay annual technical support. Establishing a login is a way to increase the value you receive as an ANSYS customer. The information on the Customer Portal is augmented regularly, so it s a good idea to log in often to see what s new.

The Focus

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
Oct '03	10/23	10/24	203	Dynamics	Irvine, CA
	10/27	10/29	902	Multiphysics Simulation for MEMS	Tempe, AZ
Nov '03	11/3	11/5	101	Introduction to ANSYS, Part I	Tempe, AZ
	11/6	11/7	301	Heat Transfer	Irvine, CA
	11/12	11/14	401	Introduction to Electromagnetics	Irvine, CA
	11/17	11/18	102	Introduction to ANSYS, Part II	Tempe, AZ
Dec '03	12/3	12/5	101	Introduction to ANSYS, Part I	Irvine, CA
	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

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.