

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

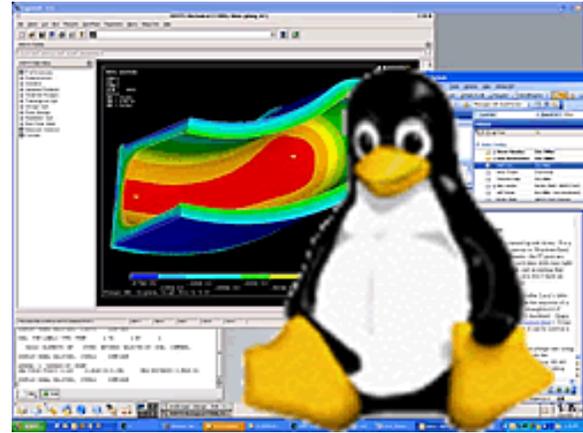
- [Linux & ANSYS: Lessons Learned](#)
- [Backup Tool](#)
- [Design Modeler FAQ](#)

On the Web

- [APDL Customization course notes now available for purchase](#)
- [ANSYS and MathCAD](#)
- [ANSYS Acquires Century Dynamics](#)

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

Linux & ANSYS: Lessons Learned

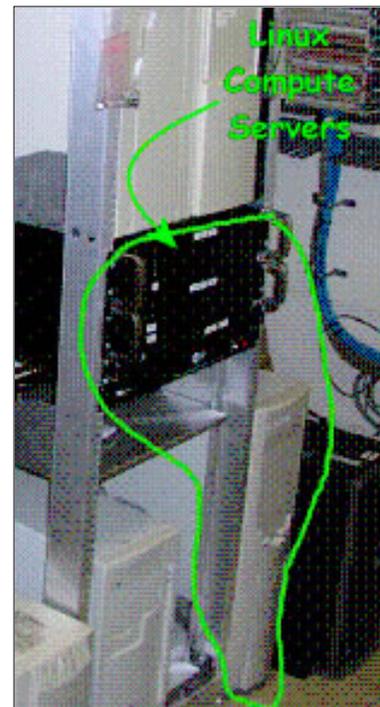
by Eric Miller, PADT

Every couple of years, the computing picture for analysts gets turned upside down. For a long time now the industry has been moving from Unix workstations to Windows/Intel desktop machines. The wintel price/performance has been fantastic, the IT guys are happier, and all of that productivity software that you spend so much time with runs in the same spot. We have been happy with a stable and known environment. However, accepting the fact that unless you work for a big company that can buy some Unix servers, you just don't have an easy way to get some extra horsepower other than getting a new box.

Then along comes this Finnish guy that may or may not have been named after Lucy's little brother. With not much of a life and a very large brain, he popped out the majority of a complete and free version of Unix that anyone can use, breaking the stranglehold of (expensive) proprietary Unix OS's that ran on (expensive) proprietary hardware. It has taken almost 10 years, but Linux has now matured to the point where it can be used as a platform for analysis computing.

► Learn the [real story](#) behind Linux.

PADT has played with Linux for about 10 years, but we took the big plunge into regularly using it for ANSYS about two years ago. We have been more than pleased with the results, and with the introduction of affordable 64-bit hardware brought on by the AMD Opteron, we are very big proponents of this paradigm. In fact, we no longer upgrade our analyst's desktop machines just so they can run bigger models. Instead, we invest in servers.



Having made a few mistakes along the way, we wanted to share our lessons learned with the ANSYS user community.

1. *Find someone who remembers how to run ANSYS on a compute server.*
Running remotely on a server is nothing new. In fact, it was the most common paradigm for many years. It is still used at many large companies that can afford the expensive proprietary hardware. We recommend finding someone who has been down this road because they already know the do's and don'ts of using compute servers. Such experience can save a lot of time and money with fumbling around if that person is available to do a little hand holding now and again.
2. *Find someone who knows what `ls -lF` / `grep` √ does.*
Hopefully, this is the same person who has compute server experience. All Unix flavors, including Linux, offer a very powerful operating environment with hundreds of

The Focus



A Publication for ANSYS Users

little tools that can make the user more productive. The problem is that they're command line based, so if you grew up on GUIs and don't have someone to teach you the power of awk, you will not get the full value out of the efficiency of this environment.

3. *Get the correct OS.*

This is the most important lesson that we learned. Being cheap, and a little lazy, we tried to load various versions of Linux that were "close" to what ANSYS, Inc. recommends. Every time we ran into problems. The reality is that all Linux builds are not alike, and differ especially in some of the high-end math and computation libraries that ANSYS will need to use. To make sure you are on the right OS, take a look in the "UNIX Installation and Configuration Guide" for the version of ANSYS you are running.

4. *Pay for the OS.*

The first thing that happens when you try and load the "correct OS" is that you find it's often not the free version of that OS. Most Linux vendors offer a free "low-end" version and a "Server" or "Professional" version that costs money. And guess what, those versions contain the high-performance libraries that ANSYS compiles against. We spent hours trying to make free versions work. Suck it up and pay the \$100 to \$200. It is money well spent.

5. *Get Cygwin.*

One of the big advantages of Unix/Linux is that it is built from the ground up to be used over a network. The command line interface, called a shell, can be accessed by any authorized machine and is very powerful. To take advantage of this power from your wintel workstation, you can use a terminal on your desktop. All Microsoft Windows OS's come with one called Hyperterm that works fine. But, if you want to be really efficient and get full access, what PADT recommends is that you load a Unix shell called [Cygwin](#) on your wintel machine. Once properly installed, you can bring up a Linux shell and use tools like rcp, rsh, rlogin, telnet, ssh, as well as shell scripting. Oh yeah, and because it's open source, it's free!

6. *Getting Cygwin loaded successfully.*

We have found that although cygwin is robust and free, almost everyone who installs it has some problems. The following steps you through the install process:

a. *Download and run setup.exe.*

When you click on Install cygwin now, you actually download a little program called setup.exe, which is an install wizard unlike any other. Be sure to put it somewhere handy because you may use it many times

b. *Choose a download source.*

The first step in the wizard after the title form allows you to choose a source. If you have a Cygwin-happy company, someone may have set up a local copy, but in most cases you will choose Install from Internet.

c. *Select Root Install Directory.*

Clicking Next> gets you to this page. It is where you define where the Cygwin

The Focus



A Publication for ANSYS Users

files go. Don't bury the directory down in Program Files or somewhere clever. Stick it at the top level of a drive, preferably c:\cygwin if you have room. Also, choose Install For: All Users since you never know when your buddy may want to use your box. The last option, Default Text File Type should be "Unix". DOS still puts extra characters in text files, which will make those files on your wintel machine unusable over on Linux.

d. *Select a Local Package Directory.*

This is a temporary place to store all the files you need to install. Pick some temp directory with a lot of space (300-500 MB).

e. *Select you Internet Connection.*

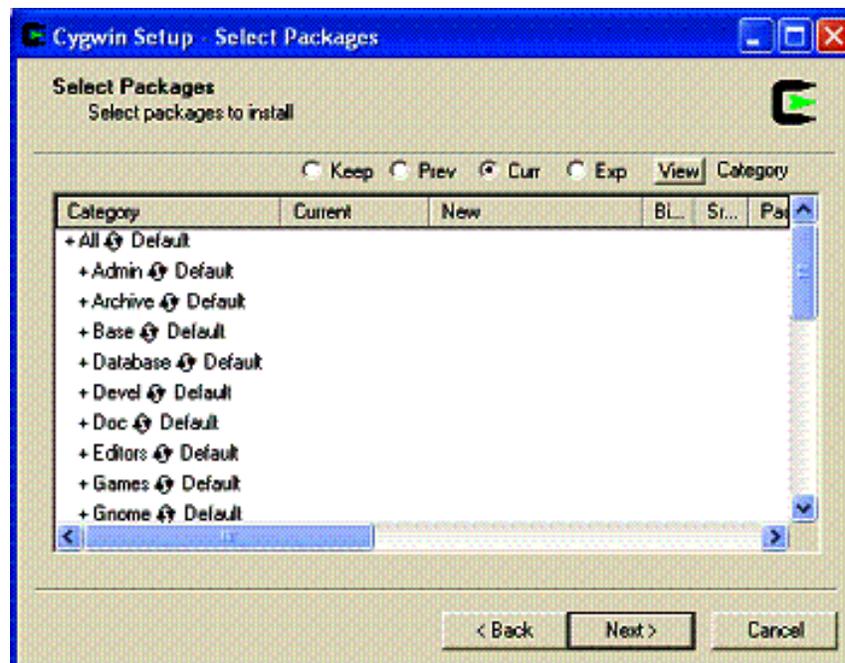
These days, just use Direct Connection. If you have problems with that, see your IT people.

f. *Choose a Download Site.*

This is a tough one. If you pick the wrong site it can take forever to download and may get interrupted. Pick a site in your country. I've had a lot of luck with kde-cygwin.sourceforge.net and mirrors.kernal.org. Avoid the .gov and .edu sites; they tend to go down or are slow.

g. *Select Packages.*

This is where most people give up. What a stupid interface! It's taken me almost a year of helping people install Cygwin for me to understand it. Basically what you are presented with is a list of all the different modules you can install. You click on the little cycle button to tell the wizard if you want a given branch to be installed. The options are: Default, Install, Reinstall, Uninstall.



You will never need half of them. However, the default download is missing

The Focus



A Publication for ANSYS Users

some major things, like telnet and vi. So what I recommend is that you go to the top of this list, at All, and choose install. Be patient, click once and let it sit for a while to update. Once it updates, everything will be selected and you will be good to go for anything you want to do.

If you don't have the disk space, then step through the first level of the tree and make sure you choose Install (not default) for: Archive, Base, Editors, Net, System, Utils, and X11.

h. *Progress.*

Once you have made your choices and hit next, the program will load the software. This can take a while, depending on download speeds. Go do something else for a while...

i. *Use it.*

Once things are done, you can go to your MS Windows start menu and choose cygwin->cygwin bash shell to get a Unix prompt on your wintel machine.

7. *Get an X-windows desktop.*

You can certainly run batch on your Linux machines just using a telnet window or a bash telnet. But to get full access to your machines, you should use X-windows. To get it running just open a cygwin bash shell and type startx. It will by default bring up a simple window manager called twm. If you want a more robust solution we recommend that you load the kde desktop, which is an option on your Cygwin download.

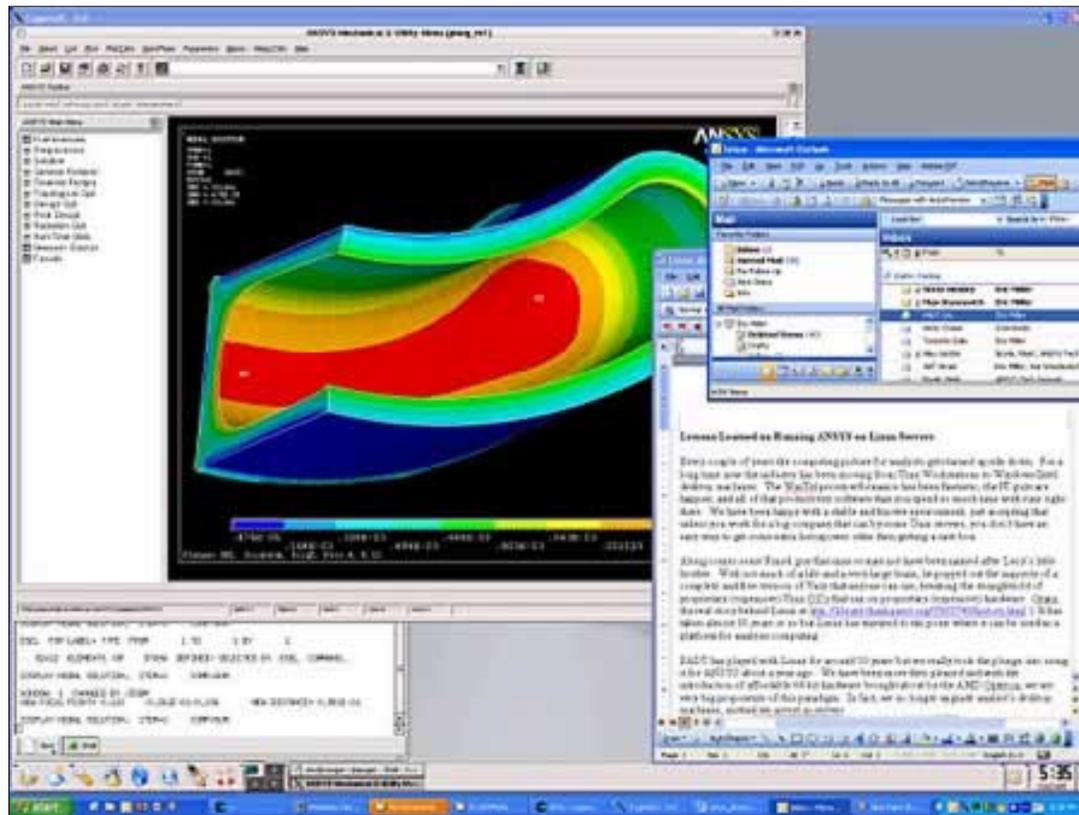
One important point about X-windows: When you execute a program on a remote machine you may get a message like "Can't open display". This is because the remote machine needs to know where to send the graphics info. To specify this, issue the command:

```
export DISPLAY=mymachine:0.0
```

The Focus



A Publication for ANSYS Users



8. *Secure your servers and wintel machines behind a good firewall then open up security.*
All of the idiot hackers out there have really made more work of using networked computers. There are some nice things you can do with Linux that get turned off by default because of security. Work with your IT person to make sure all you machines are behind a firewall and then turn off all the security walls so that you can use commands like rcp instead of ftp. If you don't know what rcp is then find that guy who knows what `ls -lF | grep V` is and ask him to set up the `hosts.equiv` so you can rsh and rcp.
9. *Learn VI, EMACS, or get a good text editor.*
Some people like to do all their pre- and post-processing on their desktop wintel machine and just run batch remotely. This is okay, but to really crank you need to get on the Linux machine and edit your scripts and APDL. To do this you can use the wimpy text editors that come with X-windows, but I recommend you learn VI, EMACS, or NEDIT. A little time spent learning one of these text editors can go a long way.
10. *Take the time to set ANSYS up correctly.*
There are a few things you need to do to make your ANSYS Linux experience enjoyable.
Read the ANSYS installation manual and do everything it says. This will make everyone's life a lot easier. This also tells you how to actually run ANSYS.

The Focus



A Publication for ANSYS Users

Also, make sure you set your path to point to the ANSYS directory. You do this by adding

```
Export PATH=$PATH:/ansys_inc/vnn/ansys/bin
```

to your .bashrc file in your home directory. *vnn* will be v90 for version 9.0, v81 for version 8.1, etc.

Next, you will find that when you start ANSYS the fonts may be a little small. I have never figured out why the default fonts are so little, but they are easy to change. Start ANSYS and go to the MenuCtrls menu and choose Font Selection. Go through this and pick some decent sized fonts. With my monitor, I use 12pt Helvetica for most things.

Lastly, grab the lower left hand corner and drag it in so that your ANSYS window only uses about two thirds of your window. This will leave room for text editors and the output window.

11. *Segregate the CFD guys.*

If you have CFD people on staff, don't let them know about your machines. CFD programs, and CFX in particular, run great in parallel and they are always adding a million more elements to get better convergence. I am sort of joking, but in some ways I'm not. If they start running 3-week jobs on your server, you are not going to see much of a benefit.

12. *Stay consistent on hardware.*

If you are going to get more than a couple of machines, we have found that it is nice to try and make them all the same. Then you don't have to worry about how much RAM one has or what type of disk space, etc.

13. *Build your own hardware.*

Unfortunately, people are charging near Unix prices for Linux systems. And if you are buying a "server" this is especially true. We recommend that you build your own hardware to get what you want at the best price.

14. *Tips for running batch.*

If you grew up on intel machines, you are used to launching and running ANSYS from a GUI. The Linux port allows this and it works well, but sometimes you may just want to run batch. In that case you need to do the following:

- a. Create a batch script that contains the APDL commands you want to run. By convention this is usually called `jobname.inp`.
- b. Go to your run directory.
- c. Run the command: `nohup ansys90 -b -j jobname -I jobname.inp -o jobname.out &`

Here is a breakdown of the command:

- `nohup` tells Linux to not stop the job if you log out (or if your remote session/PC crashes)
- `ansys90` is the version of ANSYS you are running

The Focus



A Publication for ANSYS Users

- -b tells ANSYS to run in batch mode
 - -j jobname specifies the jobname (duh! substitute your own jobname)
 - -i jobname.inp specifies the input script
 - -o jobname.out specifies the file to write to
 - & tells Linux to “run in the background”, and allows you to get your prompt back
- d. If you want to see what is going on, use `tail -f jobname.out` to scroll the output file to the screen.

If you really want to get fancy, you can create scripts that copy files from your desktop, run ANSYS, and copy files back. Here is an example for a job I am actually running while I write this article.

```
rsh cougar mkdir tmprun
rcp plung_rn1.db cougar:tmprun
rcp plung_rn1.inp cougar:tmprun
rsh cougar "cd tmprun ; nohup ansys90 -b -j plung_rn1 -I plung_rn1.inp -o plung_rn1.out &"
```

15. *Don't get discouraged.*

Change is hard and things may not work the first time you try it. Work it through and you will find a very cost-effective way to add significant horsepower to your network.

The Focus



A Publication for ANSYS Users

Backup Tool

by Rod Scholl, PADT

The day will come when your hard drive locks up, and within 5 minutes of admitting the loss of data, someone's going to say, "Did you have a backup?" As though by simply asking the question, it's implied that *everyone* backs up data. I cringe for the poor soul who, the day this happens, has to say, "Um, no ... not really. Sorry." You can feel the audible hush in the room that follows. Probably some shaking heads, and clucking tongues from people who also don't back up their data.

I haven't done the math on the odds of a hard-drive failure vs. the cost of losing a year's worth of FEA, but I'm not much of a betting type. Especially after the guy in the cube next to me had it happen, and lost everything. And with FEA in particular, this is a big concern because the bulk of what we produce stays electronic. So other than to preach where I probably have no business, I did a survey of backup tools out there looking for one suitable for FEA users. The unique feature needed is the ability to screen what is backed up by file size *and* type. Many backup tools have a global size limit, but not the ability to assign different sizes to different file types. Surprisingly, only one tool out of the thirty I checked had this ability: [AISBackup](#).

It's a no-brainer that you won't want your .osav files in most cases, and probably not your .tri's, etc., no matter what their size. But what about the .rst's and .db's? Many are less than 15 MB, but quite a few are 1.5 GB. And what about my 30 MB reports? Those images really add up. Obviously, I can't afford to lose those from a <15 MB criteria. And geometry files of the CAD? And .zips of other archives? I can't lose those, either.

For me, I have criteria that if an .db/.rst is less than 15 MB, then back it up. Most of the other ANSYS file types I just trash, thinking I can recover from a .db if needed. Perhaps see [previous recommendations](#) in *The Focus* on archiving. If I'm working with the GUI much, I likely have an unmeshed .db and meshed .db, and the unmeshed one will always make the archive with a <15 MB criteria. The meshed one may or may not, depending on the size, or if it's 2-D or 3-D. If I've APDL scripted much, my .db is rarely greater than 10 MB anyway.

With this filter logic, and excluding program files and some other junk, I store my current 120 GB of unarchived ANSYS files, email, CAD, etc. into a mere 9 GB. I

The Focus



A Publication for ANSYS Users

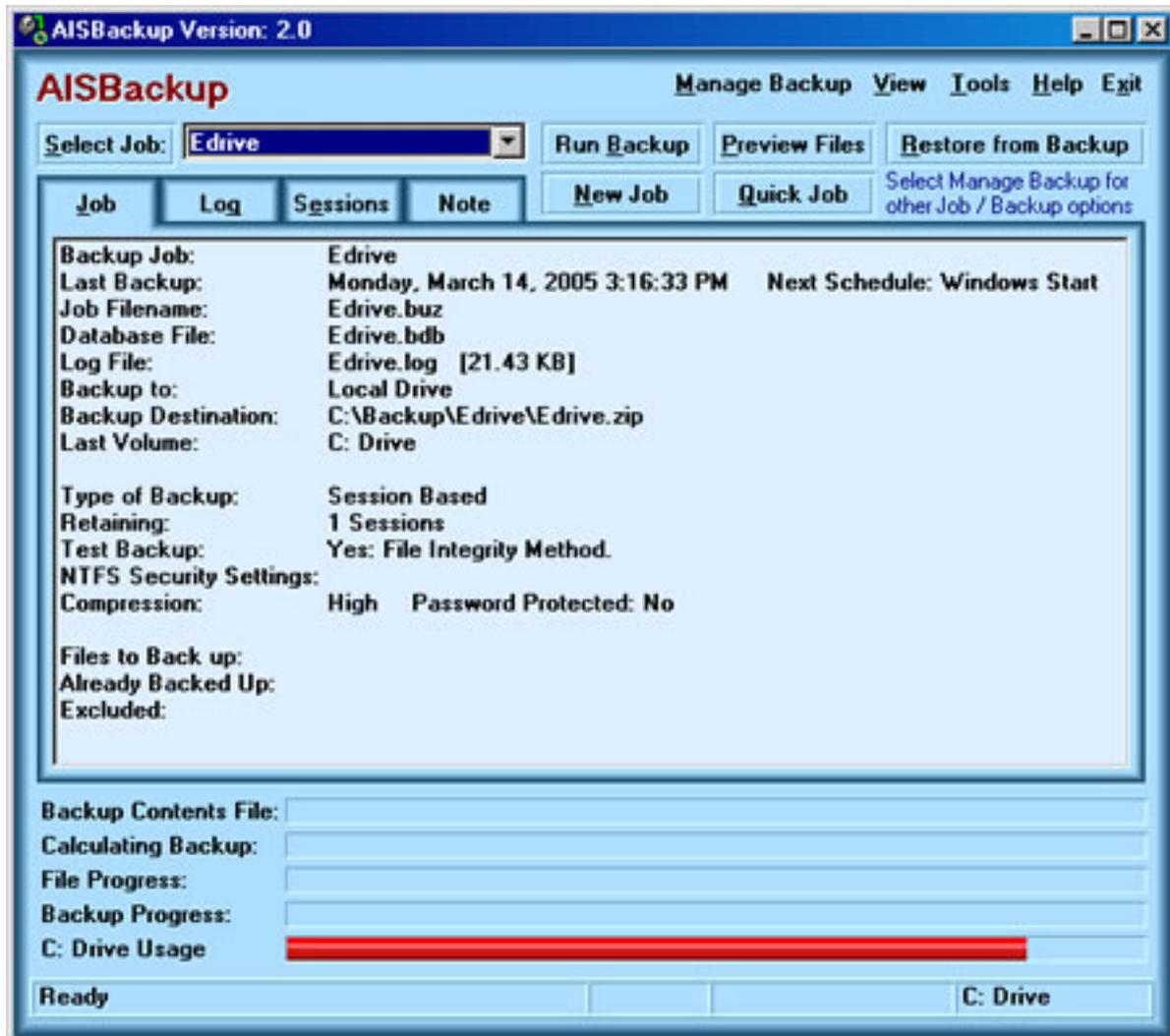
back up each drive to another (C to E, E to G, G to C). Then every 3 weeks, I burn the whole 9 GB to DVD-Rs. All of this takes place in the background, so you won't need to lift a finger after the initial setup. For me, this means theft, fire, or flood causes a maximum data loss of 3 weeks, whereas a single harddrive crash is less than a day.

I scanned about thirty backup tools for the ability to filter with a different size on each file type. I found only one, AISBackup. Fortunately, it also is a robust, full-featured archival tool as well. It really gives you control over many options, right down to the font you print the report in, FTP settings, email alerts, archive file splitting, etc. Also, the archives are simply .zip files, so you may even go back and dig up a file you accidentally deleted now and then (I use it as a Recycle Bin for my Recycle Bin). AISBackup takes a little learning because of its rich features, but there are tutorials and thorough documentation. Of course, I just bluffed and tested till I got it working. Wouldn't want to take the fun out of it by reading the "cheats"!

The Focus



A Publication for ANSYS Users



Screen Shot

[Aquarius I. S. Consultancy, Ltd](http://www.aisc.co.uk) has kindly offered a coupon below to *Focus* subscribers (you must be on the direct e-mail list). If you simply send an e-mail to customerservices@aiscl.co.uk with the subject "Coupon", you will receive a 15% discount through June 30, 2005. The e-mail reply will include instructions for purchasing at the discounted price. Future version upgrades are included in the price.

Now that you've read this article, if your hard drive crashes without a backup, you'll kick yourself even harder!

The Focus



A Publication for ANSYS Users

Design Modeler Frequently Asked Questions

by Eric Miller, PADT

For several releases now, ANSYS Inc. has offered a powerful solid modeling tool as part of their Workbench suite of products. Design Modeler, usually referred to as DM, is a unique solution in the marketplace and often generates a large number of questions about its features, integration, and positioning as a product. After a little discussion amongst sales and support people here at PADT, we have come up with the following Frequently Asked Questions:

1. **What is DM anyway?**

DM is a solid modeling tool that is integrated into ANSYS Inc.'s new Workbench product environment. It is aimed at providing modern geometry tools to people doing CAE simulation. Based on the Parasolid modeling kernel, it is a very robust tool that follows the "feature based" and "parametric" paradigm used in CAD packages like Pro/E, SolidEdge, and SolidWorks. It can be used to create geometry from scratch or to modify and defeature geometry created elsewhere.

2. **Is DM a CAD package?**

No. It does not have drafting or a true assembly environment. In addition, it allows you to create geometric definitions that most modern CAD tools would consider invalid. It is a solid modeling tool for analysts.

3. **Is DM the same as PREP7 modeling with a new GUI?**

No. DM is a totally different approach to solid modeling. PREP7 is based on bottom up and Boolean paradigms for constructive solid modeling. DM is feature based and variational. How you use it is different, and we have found it much more efficient. Also, DM is based on the industry standard Parasolid geometry kernel, while PREP7 uses two older and no longer supported geometry kernels.

4. **What does DM do that my CAD package can not?**

The primary difference between DM and a CAD package, besides missing detailing and assembly functions, is that it supports operations on your model that CAD packages do not allow. For instance, you can take two separate volumes and "glue" them together so that they share a single face between them. Useful for FE meshing, but a nightmare for manufacturing!

The Focus



A Publication for ANSYS Users

Another example is that you can create patch faces on surfaces for loading. Further, you can create solids surrounding your structures and fill in voids in your structures to provide fluid geometry for CFD meshing. DM also has a subset of commands oriented towards modeling beams, shells, and points that are unique to the needs of analysts.

5. **Do I have to have DM to use Workbench Simulation?**

In most cases the answer is no. Many tasks are made easier by DM, but if you have a good CAD tool, you should be able to provide geometry to the FE pre/post processor in Workbench (Simulation).

6. **Why do I have to pay extra for DM? Shouldn't my TECS payments cover this product? Why are the Bean Counters / Marketing Guys / Suits / [Insert_name_of_non-technical_group_here] trying to rip me off?**

Let's be honest. This is the most asked question about DM. The truth is that this was not a decision made by a bunch of greedy ex-frat boys sitting around a table at the country club. Highly technical people who can drive an FE program quite well looked at the capabilities being offered, the need in the user community, and the cost of developing and maintaining this tool, and decided to make it a separately priced item. There are three primary reasons for this:

- a. DM is robust because it is built upon best-in-class software libraries like Parasolid and D-Cubed. These software libraries are not cheap and have a significant license fee attached to them.
- b. DM is not an enhancement of Prep7 solid modeling. It does a whole lot more and a totally different manner. Saying you should get it as an upgrade is like saying you should get CFD analysis for free if you have a mechanical license.
- c. Not all users want or need DM. If ANSYS Inc. included it with every seat, and raised the price of ANSYS products accordingly, then the majority of users would complain that they don't want it and don't need it.

7. **Is this product packaging strategy a trend at ANSYS Inc.?**

No. DM is unique because of the reasons mentioned above. It is not a strategy to take functions in ANSYS, break them off as separate modules under Workbench, and then ask for extra money to give users access.

8. **Can I script DM?**

Yes. DM supports scripting for both 2D profiles and creating the full 3D

The Focus



A Publication for ANSYS Users

model. However, in most cases, you don't need to write scripts like you do in ANSYS. Feature-based parametric solid modeling is inherently parametric and in essence write a script behind the scenes. In most cases you simply need to change parameters and regenerate.

9. **Can I do parametric studies with geometry from DM?**

Heck yeah! This is one of the most powerful features in DM, and Workbench in general. DM models are fully parametric and associative not only to Simulation, but also back to the original CAD system if you started the geometry there. Parameters flow up and down through applications.

10. **If I have a CAD model already, why do I need DM?**

You may not need DM. DM comes in handy to slice up the model, to defeature it, merge surfaces, or to glue volumes together.

11. **Can I use DM with ANSYS PREP7?**

Yes, DM outputs an ANSYS ANF file that reads right into PREP7.

12. **What file formats does DM read?**

ACIS (*.sat)
CATIA V4 (*.model, *.dlv)
CATIA V5 (*.CATPart, *.CATProduct)
Inventor (*.ipt, *.iam)
IGES (*.igs, *.iges)
Mechanical Desktop (*.dwg)
Parasolid (*.x_t)
Pro/E (*.prt, *.asm)
SolidEdge (*.par, *.asm, *.psm, *.pwd)
SolidWorks (*.sldprt, *.sldasm)
UG (*.prt)

13. **What file formats does DM write?**

Parasolid (*.x_t)
DesignModeler (*.agdb)
IGES (*.iges)
ANSYS PREP7 (*.anf)

14. **Does DM support 2D?**

Yes. The 9.0 release fully supports 2D geometry. It just has to sit on the XY plane at Z=0.

15. **How does DM support assemblies?**

It depends on how you define "assemblies". DM supports the assembly of multiple objects that may or may not share geometry where they touch. See

The Focus



A Publication for ANSYS Users

the part/body question below.

16. What is this part/body stuff?

This is a concept unique to geometry used in FE meshing. You can have more than one volume connected to each other that share geometry at their interface. DM refers to a collection of geometry that shares geometry at the interface as a part and refers to each volume as a body.

17. Where do I learn more?

Contact your technical support provider and start asking questions. We love to talk about this stuff.

18. How can I try DM out?

Contact your sales person. They can get you a demo very easily. The truth is, you need to try it out to understand how useful this tool is.

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
Mar '05	3/28	3/30	902	Multiphysics Simulation for MEMS	Tempe, AZ
Apr '05	4/4	4/6	104	ANSYS Workbench Simulation - Intro	Tempe, AZ
	4/11	4/13	401	Introduction to Electromagnetics	Tempe, AZ
	4/14	4/15	604	Introduction to CFX	Tempe, AZ
	4/19	4/20	201	Basic Structural Nonlinearities	Tempe, AZ
	4/21	4/22	204	Advanced Contact and Bolt Pretension	Tempe, AZ
	4/25	4/27	152	AI*Environment	Tempe, AZ
May '05	5/2	5/4	101	Introduction to ANSYS, Part I	Irvine, CA
	5/5	5/6	102	Introduction to ANSYS, Part II	Irvine, CA
	5/9	5/10	100	Engineering with Finite Element Analysis	Tempe, AZ
	5/12	5/13	803	Tcl/Tk for ANSYS	Tempe, AZ
	5/16	5/17	604	Introduction to CFX	Irvine, CA
	5/19	5/20	203	Dynamics	Irvine, CA
	5/23	5/23	106	ANSYS Workbench DesignXplorer	Tempe, AZ
Jun '05	6/13	6/15	101	Introduction to ANSYS, Part I	Tempe, AZ
	6/20	6/21	201	Basic Structural Nonlinearities	Albuquerque, NM
	6/22	6/23	204	Advanced Contact and Bolt Pretension	Albuquerque, NM

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.



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.