

20 APDL Commands Every ANSYS Mechanical User Should Know

by **Eric Miller** 📅 August 29, 2013 ⌚ 8:35 am 💬 [Leave a comment](#) 📌 [The Focus](#)

🔖 [ANSYS Mechanical](#), [ANSYS Mechanical APDL](#), [APDL](#), [Command Objects](#), [Scripting](#)

One of the most powerful things about ANSYS Mechanical is the fact that it creates an input file that is sent to ANSYS Mechanical APDL (MAPDL) to solve. This is awesome because you as a user have complete and full access to the huge breadth and depth available in the MAPDL program. MAPDL is a good old-fashioned command driven program that takes in text commands one line at a time and executes them. So to access all those features, you just need to enter in the commands you want.

For many older users this is not a problem because we grew up using the text commands. But new users did not get the chance to be exposed to the power of APDL (ANSYS Parametric Design Language) so getting access to those advanced capabilities can be tough.

In fact, I was in a room next to one of our support engineers while they were showing a customer how to change the elements that the solver would solve (Mechanical defaults to the most common formulation, but you can change them to whatever still makes sense) and the user had to admit he had never really used or even seen APDL commands before.

So, as a way to get ANSYS Mechanical users out there started down the road of loving APDL commands, we got together and came up with a list of 20 APDL commands that every user should know. Well, actually, it is more than 20 because we grouped some

them together. We are not going to give too much detail on their usage, the APDL help is fantastic and it explains everything. In fact, if you use a copy of [PeDAL](#) you can get the help right there live as you type (yes, that was a plug for you to go out and plop down \$49 and [buy PeDAL](#)).

Also note that we are not getting in to how to script with APDL. It is a truly parametric command language in that you can replace most values in commands with parameters. It also has control logic, functions and other capabilities that you find in most scripting languages. We will focus on actual commands you use to do things in the program here. If you want to learn more about how to program with APDL, you can purchase a copy of our ["Introduction to the ANSYS Parametric Design Language"](#) book. (another plug)

Some APDL Basics

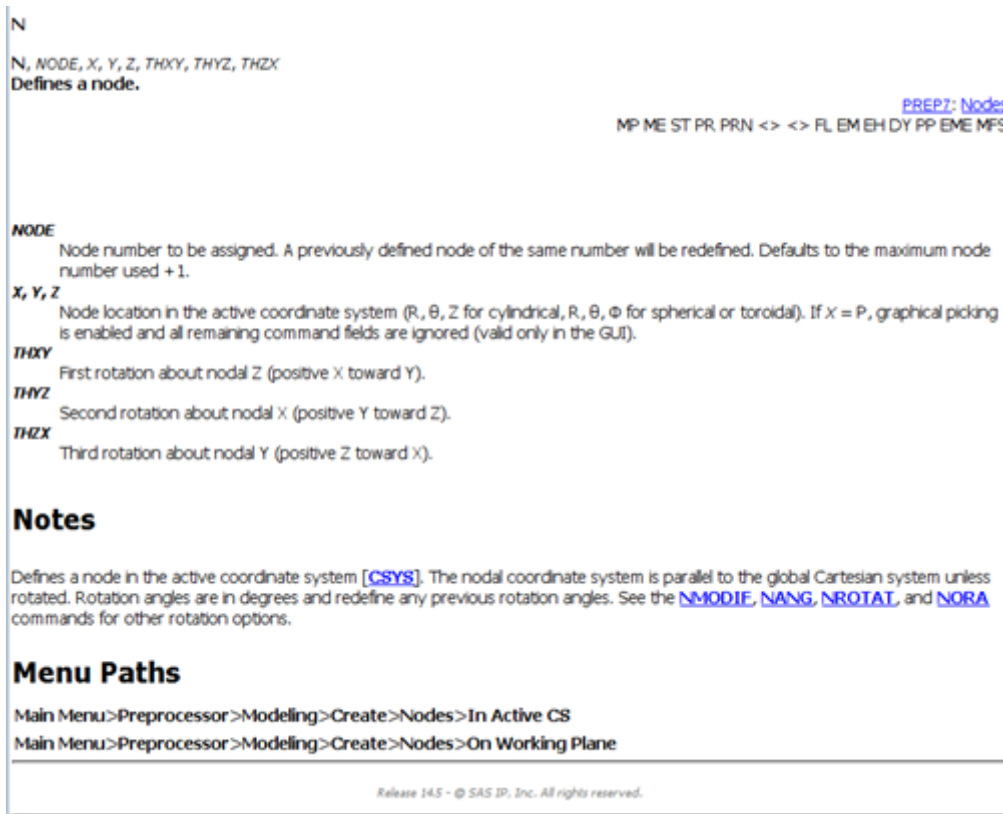
APDL was developed back in the day of punch cards. It was much easier to use than the other programs out there because the commands you entered didn't have to be formatted in columns. Instead arguments for commands are separated by commas. Therefore, instead of defining a Node in your model as:

```
345      12.456      17.4567      0.0034
```

(note that the location of that decimal point is critical). You create a line as:

```
N,345,12.456,17.4567, 0.0034
```

Trust me, that was a big deal. But what you need to know now is that all APDL commands start with a keyword and are followed by arguments. The arguments are explained in the Command Reference in the help. So the entry for creating a node looks like this:



The documentation is very consistent and you will quickly get the hang of how to get what you need out of it. The layout is explained in the help: // Command Reference // 3. Command Dictionary

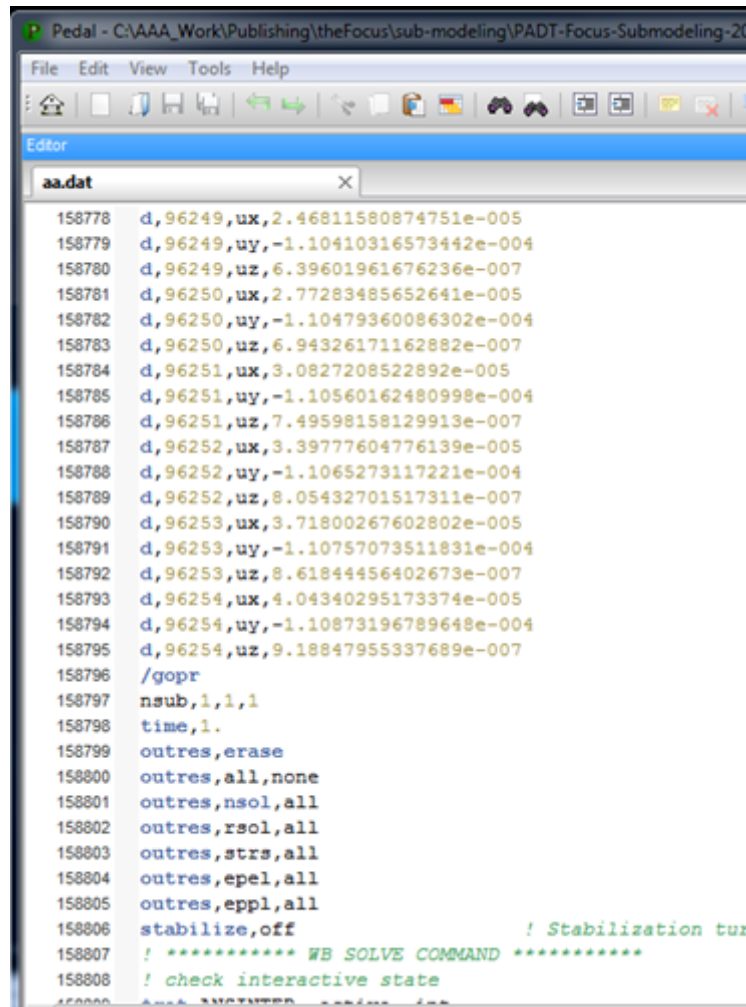
Another key thing to know about commands in MAPDL is that most entities you create (not loads and boundary conditions) have an ID number. You refer to entities by their ID number. This is a key concept that gets lost if you grew up using GUI's. So if you want to make a coordinate system and use it, you define an ID for it and then refer to that ID. Same thing goes for element definitions (Element Types), material properties, etc... Remember this, it hangs up a lot of newer users.

To use MAPDL commands you simply enter each command on a line in a command object that you place in your model tree. We did a seminar on this very subject about two years ago that you can watch here.

The idea of entity selection is fundamental to APDL. Above we point out that all entities have an ID. You can interact with each entity by specifying its ID. But when you have a lot of them, like nodes and elements, it would be a pain. So APDL deals with this by letting you select entities of a given type and making them "selected" or "unselected" Then when you execute commands, instead of specifying an ID, you can specify "ALL" and all of the selected entities are used for that command. Sometimes we refer to entities as being selected, and sometimes we refer to them as "active." The basic concept is that any entity in ANSYS Mechanical APDL can be one of two states: active/selected or

inactive/unselected. inactive/unselected entities are not used by whatever command you might be executing.

If you want to see all of the APDL command that ANSYS Mechanical writes out, simply select the setup branch of your model tree and choose Tools->Write Input File. You can view it in a text editor, or even better, in **PeDAL**.

A screenshot of the PeDAL text editor window. The title bar reads 'Pedal - C:\AAA_Work\Publishing\theFocus\sub-modeling\PADT-Focus-Submodeling-20'. The menu bar includes 'File', 'Edit', 'View', 'Tools', and 'Help'. The toolbar contains various icons for file operations and editing. The editor window shows a file named 'aa.dat' with a list of APDL commands. The commands are numbered from 158778 to 158808. The first 15 commands are displacement constraints (d,96249,ux,2.46811580874751e-005, etc.). The next 15 commands are element selection commands (/gopr, nsub,1,1,1, time,1., outres,erase, outres,all,none, outres,nsol,all, outres,rsol,all, outres, strs,all, outres,epel,all, outres,eppl,all). The final three commands are ! stabilize,off, ! ***** WB SOLVE COMMAND ***** and ! check interactive state. The text is color-coded: blue for commands, green for comments, and red for error messages.

One last important note before we go through our list of commands: the old GUI for MAPDL can be used to modify or create models as well as ANSYS Mechanical. Every action you take in the old GUI is converted into a command and stored in the jobname.log file. Many users will carry out the actions they want in an interactive session, then save the commands they need from the log file.

Wait, one more thing: Right now you need these commands. But at every release more and more of the solver is exposed in ANSYS Mechanical FUI and we end up using less and less APDL scripts. So before you write a script, make sure that ANSYS Mechanical can't already do what you want.

The Commands

1. !

An exclamation point is a comment in APDL. Any characters to the right of one are ignored by the program. Use them often and add great comments to help you and others remember what the heck you were trying to do.

2. /PREP7 – /SOLU – /POST1 – FINISH

The MAPDL program consists of a collection of 10 processors (there were more, but they have been undocumented.) Commands only work in some processors, and most only in one. If you enter in a preprocessor command when you are in the postprocessor, you will get an error.

When you create a command object in your ANSYS Mechanical model, it will be executed in either the Pre processor, the Solution processor, or in the Post processor. Depending on where in the model tree you insert the command object. If you need to go into another processor you can, you simply issue the proper command to change processors. JUST REMEMBER TO GO BACK TO THE PROCESSOR YOU STARTED IN when you are done with your commands.

/PREP7 – goes to the pre processor. Use this to change elements, create things, or modify your mesh in any way.

/SOLU – goes to the solution processor. Most of the time you will start there so you most often will use this command if you went into /PREP7 and need to get back. Modify loads, boundary conditions, and solver settings in this processor.

/POST1 – goes to the post processor. This is where you can play with your results, make your own plots, and do some very sophisticated post-processing.

FINISH – goes to the begin level. You will need to go there if you are going to play with file names.

3. TYPE – MAT – REAL – SECNUM

You only really need to know these commands if you will be making your own elements... but one of those things everyone should know because the assignment of element attributes is fundamental to the way APDL works.... so read on even if you don't need to make your own elements.

Every element in your model is assigned properties that define the element. When you define an element, instead of specifying all of its properties for each element, you can

definitions and give them numbers, then assign the number to each element. The simplest example are material properties. You define a set of material properties, give it a number, then assign that number to all the elements in your model that you want to solve with those properties.

But you do not specify the ID's when you create the elements, that would be a pain. Instead, you make the ID for each property type "active" and every element you create will be assigned the active ID's.

The commands are self explanatory: Type sets the Element Type, MAT sets the material ID, REAL set the real constant number, and SECNUM sets the active section number.

So, if you do the following:

```
type, 4
real, 2
mat, 34
secnum, 112
e, 1, 2, 3, 4, 11, 12, 13, 14
```

you get:

	ELEM	MAT	TYP	REL	ESY	SEC	NODES					
		1	34	4	2	0 112	1	2	3	4	11	12
13		14										
		2	3	4	4	0 200	101	102	103	104	111	112
113		114										

4. ET

The MAPDL solver supports hundreds of elements. ANSYS Mechanical picks the best element for whatever simulation you want to do from a general sense. But that may not be the best for your model. In such cases, you can redefine the element definition that ANSYS Mechanical used.

Note: The new element must have the same topology. You can't change a 4 noded shell into an 8 noded hex. But if the node ordering is the same (the topology) then you can make that change using the ET command.

5. EMODIF

If you define a real constant, element type, or material ID in APDL and you want to change a bunch of elements to those new ID's, you use EMODIF. This is the fastest way to change an elements definition.

6. MP – MPDATA – MPTEMP –TB – TBDATA – TBTEMP

Probably the most commonly needed APDL command for ANSYS Mechanical users are the basic material property commands. Linear properties are defined with MP command for a polynomial vs. temperature or MPDATA and MPTEMP for a piece-wise linear temperature response. Nonlinear material properties are defined with the TB, TBDATA, and TBTEMP commands.

It is always a good idea to stick your material definitions in a text file so you 1) have a record of what you used, and 2) can reuse the material model on other simulation jobs.

7. R – RMODIF

If you define an elements formulation with options on the ET command, and the material properties on the material commands, where do you specify other stuff like shell thickness, contact parameters, or hourglass stiffness? You put them in real constants. If you are new to the MAPDL solver the idea of Real constants is a bit hard to get used to.

The official explanation is:

“ Data required for the calculation of the element matrices and load vectors, but which cannot be determined by other means, are input as real constants. Typical real constants include hourglass stiffness, contact parameters, stranded coil parameters, and plane thicknesses.

It really is a place to put stuff that has no other place. R creates a real constant, and RMODIF can be used to change them.

8. NSEL – ESEL

As mentioned, selection logic is a huge part of how MAPDL works. You never want to work on each object you want to view, change, load, etc... Instead you want to place entities of a given type into an “active” group and then operate on all “active” entities. (you can group them and give them names as well, see CM-CMSEL-CMDELE below to learn about components)

When accessing MAPDL from ANSYS Mechanical you are most often working with either nodes or elements. NSEL and ESEL are used to manage what nodes and elements are active. These commands have a lot of options, so review the help.

9. NSLE – ESLN

You often select nodes and then need the elements attached to those nodes. Or you select elements and you need the nodes on those elements. NSLE and ESLN do that. NSLE selects all of the nodes on the currently active elements and ESLN does the opposite.

10. ALLSEL

A very common mistake for people writing little scripts in APDL for ANSYS Mechanical is they use selection logic to select things that they want to operate on, and then they don't remember to reselect all the nodes and elements. If you issue an NSEL and get say the nodes on the top of your part that you want to apply a load to. If you just stop there the solver will generate errors because those will be the only active nodes in the model.

ALLSEL fixes this. It simply makes everything active. It is a good idea to just stick it at the end of your scripts if you do any selecting.

11. CM – CMSEL

If you use ANSYS Mechanical you should be very familiar with the concept of Named Selections. These are groups of entities (nodes, elements, surfaces, edges, vertices) that you have put into a group so you can scope based on them rather than selecting each time. In ANSYS MAPDL these are called components and commands that work with them start with CM.

Any named selection you create for geometry in ANSYS Mechanical gets turned into a nodal component – all of the nodes that touch the geometry in the Named Selection get thrown into the component. You can also create your own node or element Named Selections and those also get created as components in MAPDL.

You can use CM to create your own components in your APDL scripts. You give it a name and operate away. You can also select components with the CMSEL command.

12. *GET

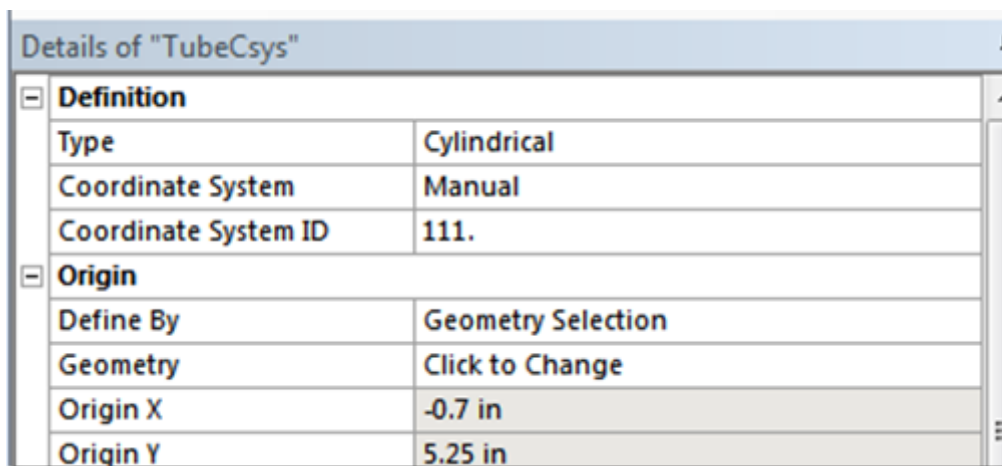
This is the single most awesomely useful command in APDL. It is a way to interrogate your model to find out all sorts of useful information: number of nodes, largest Z val

for node position, if a node is selected, loads on a node, result information, etc...

Check out the help on the command. If you ever find yourself writing a script and going "if I only knew blah de blah blah about my model..." then you probably need to use *get.

13. CSYS – LOCAL – RSYS

Coordinate systems are very important in ANSYS Mechanical and ANSYS MAPDL. In most cases you should create any coordinate systems you need in ANSYS Mechanical. They will be available to you in ANSYS MAPDL, but by default ANSYS Mechanical assigns a default ID. To use a coordinate system in MAPDL you should specify the coordinate system number in the details for a given coordinate system by changing "Coordinate System" from "Program Defined" to "Manual" and then specifying a number for "Coordinate System ID"



If you need to make a coordinate system in your APDL script, use the LOCAL command.

When you want to use a coordinate system, use CSYS to make a given coordinate system active.

Note: Coordinate system 0 is the global Cartesian system. If you change the active coordinate system make sure you set it back to the global system with CSYS,0

RSYS is like CSYS but for results. If you want to plot or list result information in a coordinate system other than the global Cartesian, use RSYS to make the coordinate system you want active.

14: NROTATE

One thing to be very aware of is that each node in a model has a rotation associated with it. By default, the UX, UY, and UZ degrees of freedom are oriented with the global Cartesian coordinate system. In ANSYS Mechanical, when you specify a load or a

boundary condition as normal or tangent to a surface, the program actually rotates all of those nodes so a degree of freedom is normal to that surface.

If you need to do that yourself because you want to apply a load or boundary condition in a certain direction besides the global Cartesian, use NROTATE. You basically select the nodes you want to rotate, specify the active coordinate system with CSYS, then issue NROTATE,ALL to rotate them.

Be careful though. You don't want to screw with any rotations that ANSYS Mechanical specified.

15. D

The most common boundary condition is displacement, even for temperature. To specify those in an ANSYS MAPDL script, use the D command. Most people use nodal selection or components to apply displacements to multiple nodes.

In its simplest form you apply a single value for displacement to one node in one degree of freedom. But you can specify multiple nodes, multiple degrees of freedom, and more powerfully, the value for deflection can be a table. Learn about tables [here](#).

16. F

The F command is the same as the D command, except it defines forces instead of displacement. Know, it, use it.

17. SF – SFE

If you need to apply a pressure load, you use either SF to apply to nodes or SFE to apply to elements. It works a lot like the D and F commands.

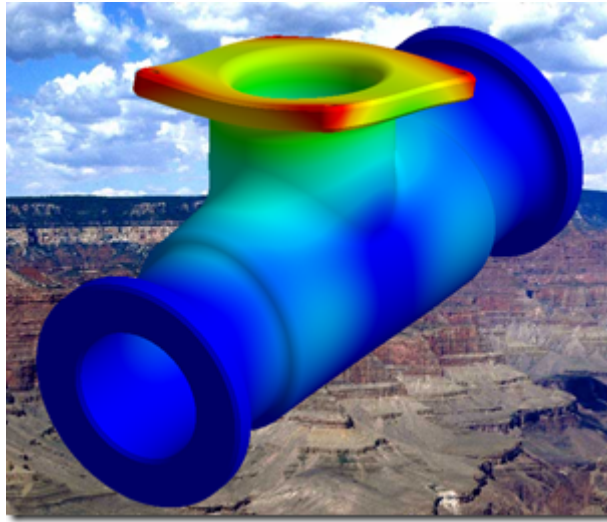
18. /OUTPUT

When the ANSYS MAPDL solver is solving away it writes bits and pieces of information to a file called jobname.out, where jobname is the name of your solver job. Sometimes you may want to write out specific information, say list the stresses for all the currently selected nodes, to a file. use /OUTPUT,filename to redirect output to a file. When you are done specify /OUTPUT with no options and it will go back to the standard output.

19. /SHOW

ANSYS MAPDL has some very sophisticated plotting capabilities. There are a ton of command and options used to setup and create a plot, but the most important is

/SHOW,png. This tells ANSYS MAPDL that all plots from now on will be written in PNG format to a file. Read all about how to use this command, and how to control your plots, [here](#).



20. ETABLE

The ANSYS MAPDL solver solves for a lot of values. The more complex the element you are using, the more the number of values you can store. But how do you get access to the more obscure ones? ETABLE. Issue 38 of The Focus from 2005 goes in to some of the things you can do with [ETABLE](#).

Where to go From Here

This is certainly not the definitive list. Ask 20 ANSYS MAPDL users what APDL commands all ANSYS Mechanical users should know, and you might get five or six in common. But based on the support calls we get and the scripts we write, this 20 are the most common that we use.

Command help is your friend here. Use it a lot.

The other thing you should do is open up ANSYS MAPDL interactively and play with these commands. See what happens when you execute them.

Share this:



Like this:

Loading...

Related

[Four Different Ways to Add Customization to ANSYS Mechanical](#)

May 22, 2019

In "The Focus"

[How to use APDL Command Snippets in your ANSYS Mechanical Model](#)

July 8, 2019

In "News"

[APDL Math–Access to the ANSYS Solver Matrices with APDL](#)

February 23, 2012

In "The Focus"

You must **log in** to post a comment.

< **PADT's Tempe Open House and AZ Tech Council Progress Forum – 2 Weeks Away**
Construction Started on PADT's new High Speed Fiber Connection >

Contact PADT
1-800-293-PADT 
info@padtinc.com 

SEARCH

Search



LINKS

[PADT Website](#)

[Contact PADT](#)

[PADTMarket.com](#)

[Manage PADT Subscriptions](#)

SUBSCRIBE TO OUR BLOG

Enter your email address to subscribe to this blog and receive notifications of new posts by email.

Join 1,804 other subscribers

Email Address

[Privacy](#) - [Terms](#)

Subscribe

UPCOMING EVENTS

01/19/2022 - 01/21/2022
[Arizona Photonics Days](#)

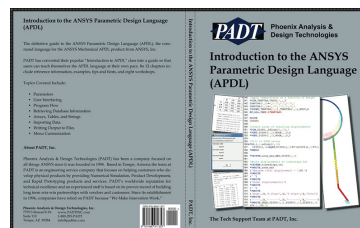
04/04/2022 - 04/07/2022
[37th Space Symposium](#)
[Arizona Space Industry Booth](#)

04/12/2022 - 04/14/2022
[D&M West](#) | [MD&M West](#)

3D PRINTING GLOSSARY



INTRODUCTION TO THE ANSYS PARAMETRIC DESIGN LANGUAGE



Learn APDL with this workshop based book written by PADT's Technical Support Team. The new and improved Second Edition contains additional chapters on APDL Math and APDL in ANSYS Mechanical.

[Privacy](#) - [Terms](#)

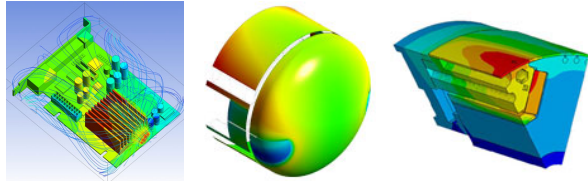
Now available on Kindle as well as paperback.

[More...](#)

SIMULATION SERVICES

PADT's simulation engineers are true experts in virtual prototyping. Trust the people you come to for ANSYS expertise to handle your simulation outsourcing needs.

Structural, Thermal, Fluid, Electromagnetic, and Systems.



LOOKING FOR ANSYS TRAINING?



Let the people who write “The Focus” train you and your team. Check out our [schedule](#) or [contact us](#) to set up a class at your place or something custom.

PODCAST: ALL THINGS ANSYS



ANSYS ACADEMIC PROGRAM



RSS SUBSCRIBE:



RECENT POSTS:

[Surprise – 2021 Turned out to Be a Lot Like 2020](#)

[All Things Ansys 102: Electronics Reliability Updates in Ansys 2021 R2](#)

[Simulating an Electro-Permanent Magnet \(EPM\) using Ansys Maxwell](#)

[Ansys Pro – Premium – Enterprise Electronics Licensing Adjustments](#)

[All Things Ansys 101: Additive & Structural Optimization Updates in Ansys 2021 R2](#)

CATEGORIES

[Additive Manufacturing](#)

[Ansys](#)

[ANSYS Discovery](#)

[ANSYS Energy Innovation Campaign](#)

[ANSYS R 18](#)

[Carbon](#)

[Education](#)

[Events](#)

[Flownex](#)

[Fun](#)

Getting To Know PADT

News

Nimbix

PADT Medical

PADT Startup Spotlight

Podcast

Product Development

Publications

Startups

Stratasys

Stratasys Marketing

The Focus

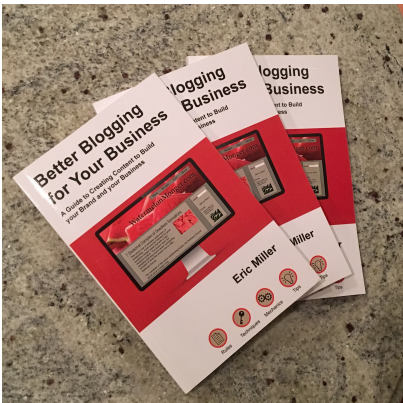
Uncategorized

Webinar

Contact PADT
1-800-293-PADT
info@padtinc.com



THINKING ABOUT A BLOG FOR YOUR COMPANY?



Where do you start? How do you keep it going? Where do I get ideas for posts? Should I use humor?

These questions and many others are answered in the book that was inspired by the success of PADT's blog:

Better Blogging for your Business

Available now as a Kindle book or softcover on Amazon.

PADT EMAIL

Manage how PADT Emails You

Want to receive Emails from PADT? Please select any of the basic topics below to tell us what you are interested in. We promise to keep it simple sharing news, opportunities, and useful information. You can come back and change your settings whenever you need to.

* Email

State/Province

Company

* Email Lists

- ☐ PADT Additive & Advanced Manufacturing Email List
- ☐ PADT General Information Email List
- ☐ PADT Product Development and Testing Email List
- ☐ PADT Simulation Email List

By submitting this form, you are consenting to receive marketing emails from: Phoenix Analysis and Design Technologies, 7755 S. Research Dr., Suite 110, Tempe, AZ, 85284, US, <http://www.padtinc.com>. You can revoke your consent to receive emails at any time by using the SafeUnsubscribe® link, found at the bottom of every email. **Emails are serviced by Constant Contact.**

Sign Up!

[Privacy](#) - [Terms](#)

Contact PADT
1-800-293-PADT ☎
info@padtinc.com ✉

SEARCH

Search

LINKS

[PADT Website](#)

[Contact PADT](#)

[PADTMarket.com](#)

[Manage PADT Subscriptions](#)

ANSYS STARTUP PROGRAM



LOOKING FOR ANSYS TRAINING?



Let the people who write "The Focus" train you and your team. Check out our [schedule](#) or [contact us](#) to set up a class at your place or something custom.

TRYING TO FIND A COMPUTER BUILT FOR SIMULATION?

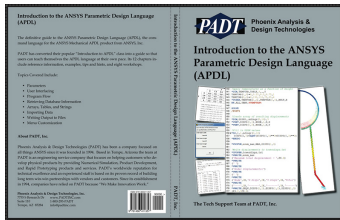


Workstations, Servers,
and Clusters
designed by
PADT
specifically for

simulation users.

[Learn More](#)

INTRODUCTION TO THE ANSYS PARAMETRIC DESIGN LANGUAGE



Learn APDL with this
workshop based book
written by PADT's
Technical Support Team.
The new and improved
Second Edition contains
additional chapters on
APDL Math and APDL in
ANSYS Mechanical.

Now available on Kindle
as well as paperback

[More...](#)

RECENT POSTS

[Surprise – 2021 Turned out to Be a Lot Like 2020](#)

[All Things Ansys 102: Electronics Reliability Updates in Ansys 2021 R2](#)

[Simulating an Electro-Permanent Magnet \(EPM\) using Ansys Maxwell](#)

[Ansys Pro – Premium – Enterprise Electronics Licensing Adjustments](#)

[All Things Ansys 101: Additive & Structural Optimization Updates in Ansys 2021 R2](#)

CATEGORIES

Additive Manufacturing

Ansys

ANSYS Discovery

ANSYS Energy Innovation Campaign

ANSYS R 18

Carbon

Education

Events

Flownex

Fun

Getting To Know PADT

News

Nimbix

PADT Medical

PADT Startup Spotlight

Podcast

Product Development

Publications

Startups

Stratasys

Stratasys Marketing

The Focus

Uncategorized

