# Starting ANSYS Products From the Command Line

by **Eric Miller**     February 29, 2012     10:04 am     💬 **Leave a comment**     🔖 **The Focus**

 Sometimes you just get tired of clicking on icons.  Sometimes you just need to feel the control and power of launching your applications from the command line.  You type it in, you hit the enter key, and sometimes you can actually hear the disk spin up or the fan run faster to cool the processor as the program you asked for, the program you took time to type out, leaps to life in front of you. Satisfaction.

OK, maybe not.  More often you are scripting some batch solves.  Or maybe you are using the graphical user interface in Workbench but you need to set options for the solvers you are running from within workbench.  Because most of the solvers in the ANSYS family of products predate such new-fangled concepts as GUI's, and because they are often run remotely on other machines, they have command line interfaces. And that gives the knowledgeable user more power and control.

## General Concepts for Launching from the Command Line

Although the number of options available changes from application to applications, there a few common things you should know.

## Paths

The first and most important concept is to be aware of the path.  This is where most errors happen. One of the big changes over the years is that as software gets more complicated, the executable program or script that you use to launch a solver is now buried deep down inside a directory structure.  Since we never run in that actual directory we need to tell the operating system where the executable is. You can do this by including the full directory path in your command line argument, or by adding it to your path.

On Linux follow the directions in the help:

```
// Installation and Licensing Documentation // Linux Installation Guide // 5. Post-
Installation Instructions
```

for each of the products you want to run. Generally, you need to set the PATH environmental variable in your .cshrc, .login, or .profile.

On Windows it is not documented, the assumption being that you will be clicking on icons and not typing into a command window.  So a little detective work is needed. Use a file explorer and the Linux documentation on launching to locate the executable for solvers you want to use:

```
// Installation and Licensing Documentation // Linux Installation Guide // 5. Post-
Installation Instructions // 5.3. Launching ANSYS, Inc. Products
```

The /ansys_inc part is usually replaced with c:\Program Files\ANSYS Inc.  The rest of the path is pretty much the same, swapping forward slashes with backward slashes.  Use these paths in your command line or add to your path by:

- From the desktop, right-click **My Computer** and click **Properties**.
- In the System Properties window, click on the **Advanced** tab.
- In the Advanced section, click the **Environment Variables** button.
- Finally, in the Environment Variables window (as shown below), highlight the **Path** variable in the Systems Variable section and click the **Edit** button. Add or modify the path lines with the paths you wish the computer to access. Each different directory is separated with a semicolon as shown below.
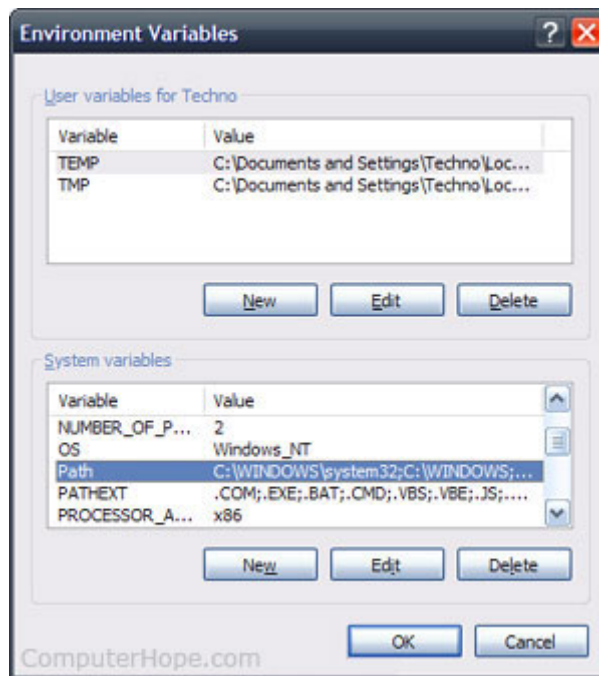
Important note for Windows: If you are typing the path in on the command line, you need to put it in double quotes. The convention on Windows is to specify directories with spaces in the name. But the convention is not to have a command line parser that recognizes this. So you will get an error if you type:

```
C:\Program Files\ANSYS Inc\v140\ansys\bin\winx64\ansys140.exe
```

But if you put it in quotes, it works fine:

```
"C:\Program Files\ANSYS Inc\v140\ansys\bin\winx64\ansys140.exe"
```

## Versions Numbering

If you look at the example for launching MAPDL above you will notice that 140 is used in the directory path and in the name of the executable. This will change with version: v130, v145, etc… Just be aware of that if you are reading this blog posting in 3 years and we are on release 16.5. you would use:

```
"C:\Program Files\ANSYS Inc\v165\ansys\bin\winx64\ansys165.exe"
```

## Where do you Launch From?

You of course need a command line to launch a solver. This is usually a window that lets you type operating system commands: called a Command Prompt in Windows or a shell on Linux. On Linux it can be an xterm window, a console window, or some other terminal window you have opened.

But you can also launch from a script, and that script can be launched from a command prompt or shell, or it can be launched by an icon. All that needs to happen is that the script needs to be executed with the environmental variables required for the command prompt/script or the GUI. If you don't know how to make that happen, contact your IT support or someone who understands your operating system and how it runs processes.

# ANSYS Mechanical APDL

The solver with the most options and capabilities from the command line is Mechanical APDL. So we will start there. It is important to know these even if you use Mechanical most of the time. That is because you can set these, and better control your solves, under Tools->Options->Mechanical APDL. Here is what that dialog looks like:
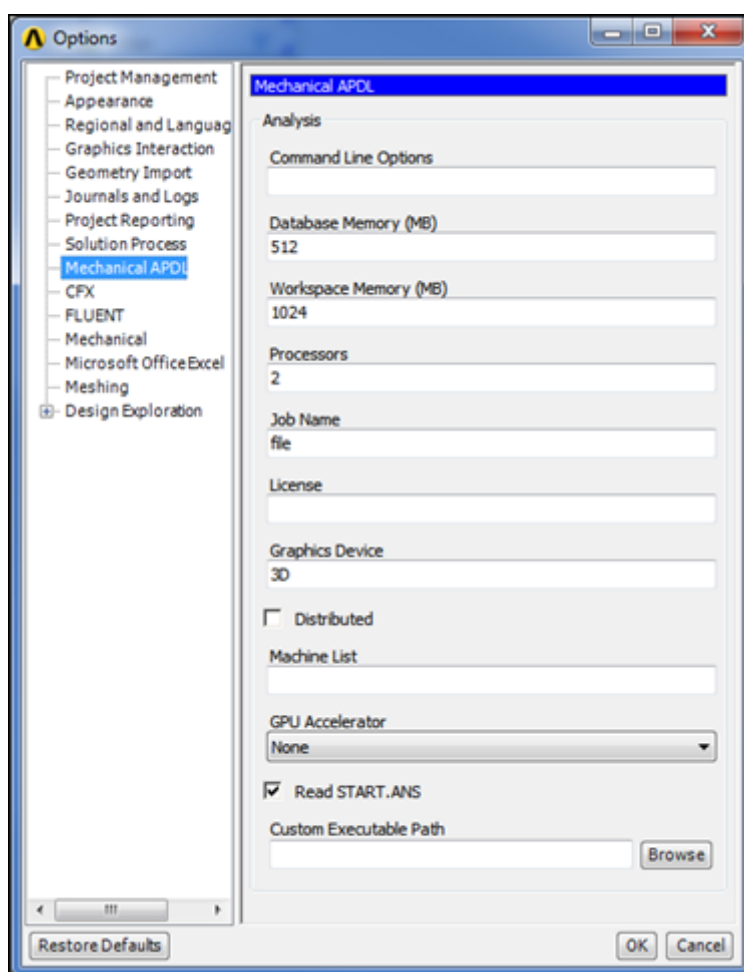


The most common settings have their own widgets, and the others can all be accessed by using "- string" command line style arguments in the first text widget aptly named: Command Line Options.

Here are the options, grouped for your studying pleasure:

| Option | Type | Description |
|---|---|---|

| -ansexe | Customization | In the ANSYS Workbench environment, activates a custom ANSYS executable. |
|---|---|---|
| -custom | Customization | Calls a custom ANSYS executable. See help on running custom executables for more |
| -acc *device* | HPC | Enables the use of GPU compute accelerator. As this is written, nvidia is the only option. But as other cards become available look for this to have other options. Check the help. |
| -dis | HPC | Enables Distributed ANSYS. See the Parallel Processing Guide for more information. |
| --*machines* | HPC | Specifies the machines on which to run a Distributed ANSYS analysis. See Starting Distributed ANSYS in the Parallel Processing Guide for more information. |
| -mpi | HPC | Specifies the type of MPI to use. See the Parallel Processing Guide for more information. |
| -mpifile | HPC | Specifies an existing MPI file (appfile) to be used in a Distributed ANSYS run. See Using MPI appfiles in the Parallel Processing Guide for more information. |
| -np *value* | HPC | Specifies the number of processors to use when running Distributed ANSYS or Shared-memory ANSYS. |
| -d *device* | Interface | Specifies the type of graphics device. This option applies only to interactive mode. For Linux systems, graphics device choices are X11, X11C, or 3D. For Windows systems, graphics device options are WIN32 or WIN32C, or 3D. |
| -g | Interface | Launches the ANSYS program with the Graphical User |

| | | Interface (GUI) on. Linux only. On windows do a /SHOW and /MENU,ON to get the GUI up. |
|---|---|---|
| -l *language* | Interface | Specifies a language file to use other than US English. |
| -b [list \| nolist] | Launch | Activates the ANSYS program in batch mode. The options -b list or -b by itself cause the input listing to be included in the output. The -b nolist option causes the input listing not to be included |
| -i *inputname* | Launch | Specifies the name of the file to read input into ANSYS for batch processing. On Linux, the preferred method to indicate an input file is <. Requried with -b option. |
| -j *Jobname* | Launch | Specifies the initial jobname, a name assigned to all files generated by the program for a specific model. If you omit the -j option, the jobname is assumed to be file. |
| -o *outputname* | Launch | Specifies the name of the file to store the output from a batch execution of ANSYS. On Linux, the preferred method to indicate an output file is >. |
| -p *productname* | Launch | Defines which ANSYS product will run during the session (ANSYS Multiphysics, ANSYS Structural, etc.). This is how you pull a different licence from the default. Very handing if you have multiple licenses to choose from. |
| -s [read \| noread] | Launch | Specifies whether the program reads the start140.ans file at start-up. If you omit the -s option, ANSYS reads the start140.ans file in interactive mode and not in batch mode. |
| -dir | Luanch | Defines the initial working directory. Remember t double quotes if you have spaces in your director |

| | | path name. Using the -dir option overrides the ANSYS140_WORKING_DIRECTORY environment variable. |
|---|---|---|
| -db *value* | Memory | Defines the portion of workspace (memory) to be used as the initial allocation for the database. This and -m are the two most important options. If you ever find that ANSYS is writing a *.PAGE file, up this number. |
| -m *value* | Memory | Idefines the total memory to reserve for the program. It is always better to reserve it up front rather than letting ANSYS grab as it needs. |
| -schost *host name* | MFX | Specifies the host machine on which the coupling service is running (to which the co-simulation participant/solver must connect) in a System Coupling analysis. |
| -scname *name of the solver* | MFX | Specifies the unique name used by the co-simulation participant to identify itself to the coupling service in a System Coupling analysis. For Linux systems, you need to escape the quotes or escape the space to have the name recognized with a space: |
| -scport *port number* | MFX | Specifies the port on the host machine upon which the coupling service is listening for connections from co-simulation participants in a System Coupling analysis. |
| -mfm | MFX | Specifies the master field name in an ANSYS Multi-field solver – MFX analysis. See Starting and Stopping an MFX Analysis in the Coupled-Field Analysis Guide for more information. |
| -ser *value* | MFX | Specifies the communication port number between ANSYS and CFX in an MFX analysis. |
| | | |

| -dvt | Other | Enables ANSYS DesignXplorer advanced task (add-on). |
|---|---|---|
| -dyn | Other | Enables LS-DYNA. |
| -v | Other | Returns the ANSYS release number, update number, copyright date, customer number, and license manager version number. Does not actually run ANSYS MAPDL. |
| –name value | Params | Defines ANSYS parameters at program start-up. The parameter name must be at least two characters long. These get passed into ANSYS and are used by any APDL scripts you run. |

The ones that everyone should know about are: –p, –m, –db.  We find that not using these to define what license to use (-p) or to control how memory is pre-allocated (-m, –db) generate the most tech support questions.   The next most important is the –np command. Use this to define more processors if you have HPC licenses.

## Cheating – Use the Launcher

Sometimes the options can get long and confusing.  So what I do is I use the "ANSYS Mechanical APDL Launcher" and fill in all the forms. then go to tools->View Display Command Line and I can see all the options.
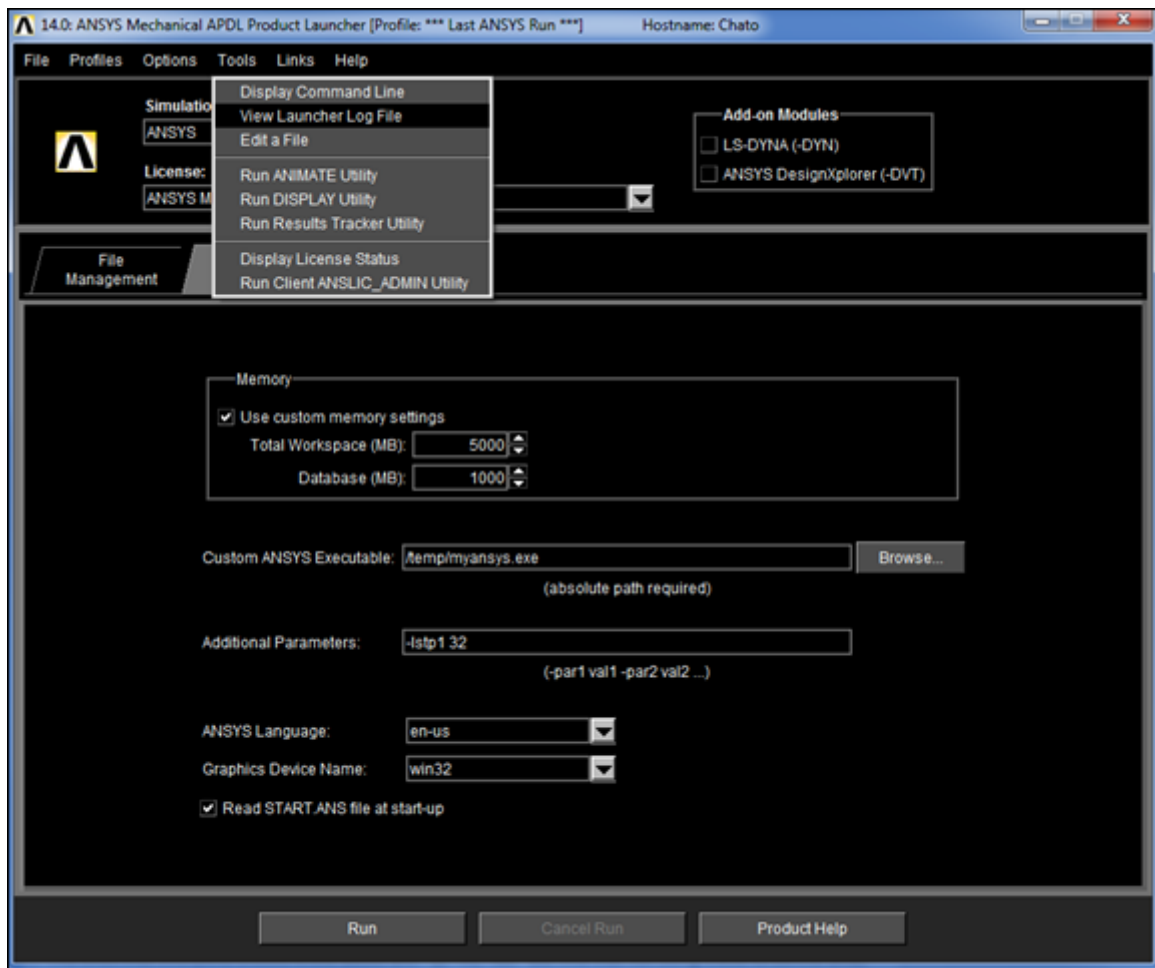
Here is a a fancy command line that got generated that way:

```
"C:\Program Files\ANSYS Inc\v140\ANSYS\bin\winx64\ansys140.exe"  -g -p ane3flds -np 2
-acc nvidia -dir "c:\temp" -j "grgewrt1" -s read -m 5000 -db 1000 -l en-us -lstp1 32
-t -d win32 -custom "/temp/myansys.exe"
```

It runs interactive (-g), uses a multiphysics license (-p ane3flds), grabs two processors (-np 2), uses an NVIDEA GPU (-acc nvidia (I don't have one…)), runs in my temp directory (-dir), uses a job name of grgewrt1 (-j), reads the start.ans file (-s), grabs 5000mb and 1000mb for memory (-m, –db), uses English, passes in a parameter called lstp1 and sets it to 32 (-), uses the win32 graphics driver (-d) and runs my custom ANSYS executable (-custom).

I have no idea what –t is.  Some undocumented option I guess…

## ANSYS Workbench

ANSYS Workbench also has some command line arguments. not as rich as what is available in MAPDL, but still powerful. It allows you to run Mechanical in Batch or Interactive mode, supply python commands as needed.  The key thing to remember is that the workbench interface is not Mechanical or FLUENT. It is the infrastructure tha

other programs run on. Scripting in workbench allows you to control material properties, parameters, and how systems are created and executed.

Here are the options:

| Argument | Operation |
|---|---|
| -B | Run ANSYS Workbench in batch mode. In this mode, the user interface is not displayed and a console window is opened. The functionality of the console window is the same as the ANSYS Workbench Command Window. |
| -R *<ANSYS Workbench script file>* | Replay the specified ANSYS Workbench script file on start-up. If specified in conjunction with –B, ANSYS Workbench will start in batch mode, execute the specified script, and shut down at the completion of script execution. |
| -I | Run ANSYS Workbench in interactive mode. This is typically the default, but if specified in conjunction with –B, both the user interface and console window are opened. |
| -X | Run ANSYS Workbench interactively and then exit upon completion of script execution. Typically used in conjunction with –R. |
| -F *<ANSYS Workbench project file>* | Load the specified ANSYS Workbench project file on start-up. |
| -E *<command>* | Execute the specified ANSYS Workbench scripting command on start-up. You can issue multiple commands, separated with a semicolon (;), or specify this argument multiple times and the commands will be executed in order. |

The big deal in this list is the –B argument.  This allows you to run workbench, and applications controlled by the project page, in batch mode.  You will usually use the –R argument to specify the Iron Python script you want to run.  In most cases you will often want to throw in a –X to make sure it exists when the script is done.

# Other ANSYS Products

Here is where it gets boring.  The other products just don't have all those options. At least not documented.  So you simply find the executable and run it.  Here is the list for Linux. Use it to find the location on Windows.

| Product | Command |
|---|---|
| Mechanical APDL | /ansys_inc/v140/ansys/bin/ansys140 |
| ANSYS Workbench | /ansys_inc/v140/Framework/bin/*<platform>*/runwb2 |
| ANSYS CFX | /ansys_inc/v140/CFX/bin/cfx5 |
| ANSYS FLUENT | /ansys_inc/v140/fluent/bin/fluent |
| ANSYS ICEM CFD | /ansys_inc/v140/icemcfd/*<platform>*/bin/icemcfd |
| ANSYS POLYFLOW | /ansys_inc/v140/polyflow/bin/polyman |
| ANSYS CFD-Post | /ansys_inc/v140/CFD-Post/bin/cfdpost |
| ANSYS Icepak | /ansys_inc/v140/Icepak/bin/icepak |
| ANSYS TurboGrid | /ansys_inc/v140/TurboGrid/bin/cfxtg |
| ANSYS AUTODYN | /ansys_inc/v140/autodyn/bin/autodyn140 |

**Share this:**

Privacy - Terms

**Like this:**

Loading...

**Related**

[Workbench and Excel, Part 2: Driving Workbench From Excel with Python](#)
July 13, 2011
In "The Focus"

[Four Different Ways to Add Customization to ANSYS Mechanical](#)
May 22, 2019
In "The Focus"

[Overcoming Convergence Difficulties in ANSYS Workbench Mechanical, Part II: Quick Usage of Mechanical APDL to Plot Distorted Elements](#)
October 18, 2012
In "The Focus"

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

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

Privacy · Terms

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

Join 1,804 other subscribers

Email Address

Subscribe

## UPCOMMING EVENTS

01/19/2022 - 01/21/2022
Arizona Photonics Days

04/04/2022 - 04/07/2022
37th Space Symposium
Arizona Space Industry Booth
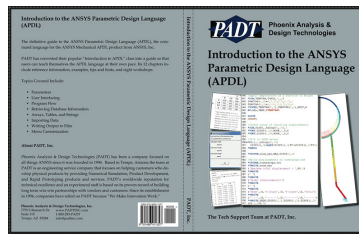
04/12/2022 - 04/14/2022
D&M West | MD&M West

## 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.

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.**
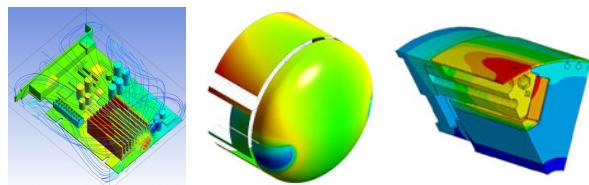


## LOOKING FOR ANSYS TRAINING?



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 ✉

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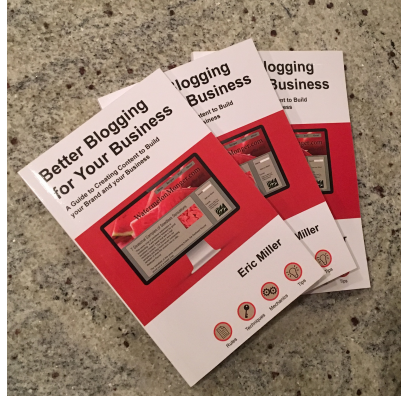
**Where do you start? How do you keep it going? Where do I get ideas for posts? Should I use humor?**

**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!**

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.
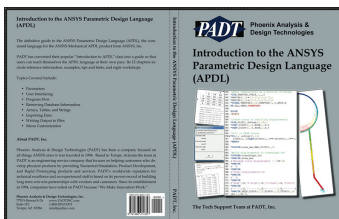
## TRYING TO FIND A COMPUTER BUILT FOR SIMULATION?



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

## 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

Connect with PADT

Privacy - Terms