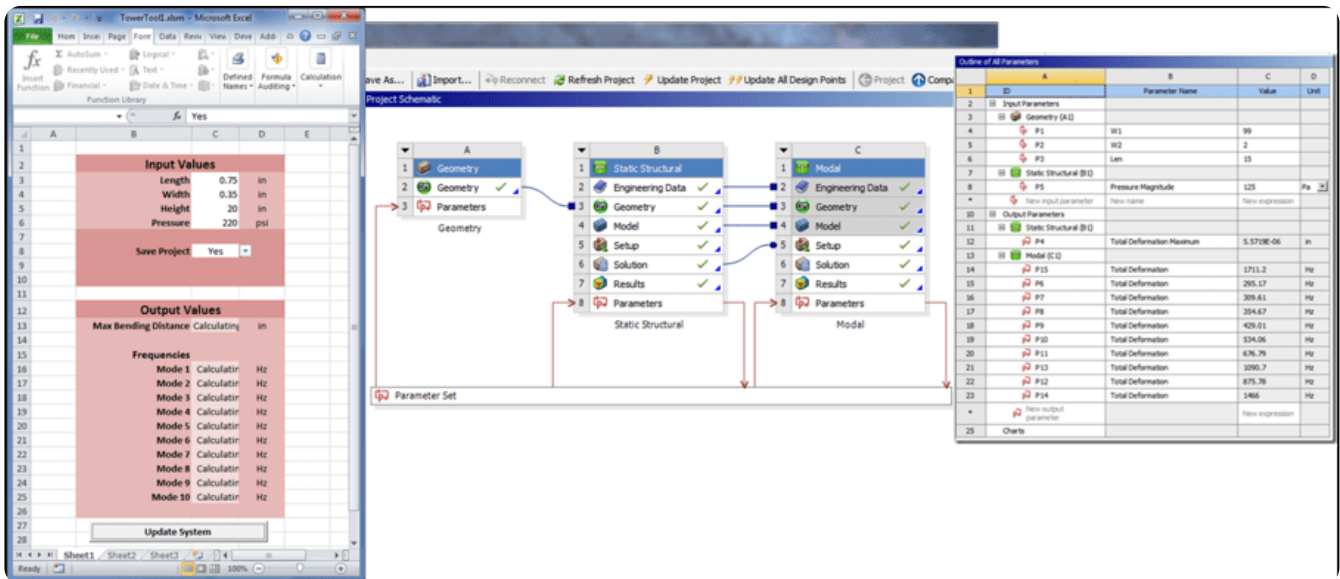


PADT, Inc. – The Blog

WE MAKE INNOVATION WORK



Four Different Ways to Add Customization to ANSYS Mechanical

by **Eric Miller** May 22, 2019 9:05 am [Leave a comment](#) [The Focus](#)

ACT, ANSYS, ANSYS ACT, ANSYS Mechanical, ANSYS Mechanical APDL, APDL, Customization, programming

ANSYS Mechanical is a very powerful tool right out of the box. Long gone are the days when an FEA tool was just a solver, and users had to write code to create input files and interpret the results. Most of the time you never have to write anything to effectively use ANSYS Mechanical. But, users can realize significant gains in productivity and access greater functionality through customization. And it is easy to do.

Before we talk about the four options, we need to remember how the tool, ANSYS Mechanical, is actually structured. The interface that users interact with is a version

ANSYS Workbench called ANSYS Mechanical. The interface allows users to connect to geometry, build and modify their model, set up their solution, submit a solve, and review results. The solve itself is done in ANSYS Mechanical APDL. This is the original ANSYS Multiphysics program.

When you press the solve button ANSYS Mechanical writes out commands in the languages used by ANSYS Mechanical APDL, called the ANSYS Parametric Design Language, or APDL. Yes, that is where ANSYS Mechanical APDL got its name. We like to call it MAPDL for short. (Side note: years ago we started a campaign to call it map-dul. It didn't work.) Once the file is written, MAPDL is started, the file is read in, the solve happens, and all of the requested output files are written. Then ANSYS Mechanical reads those files and shows results to the user.

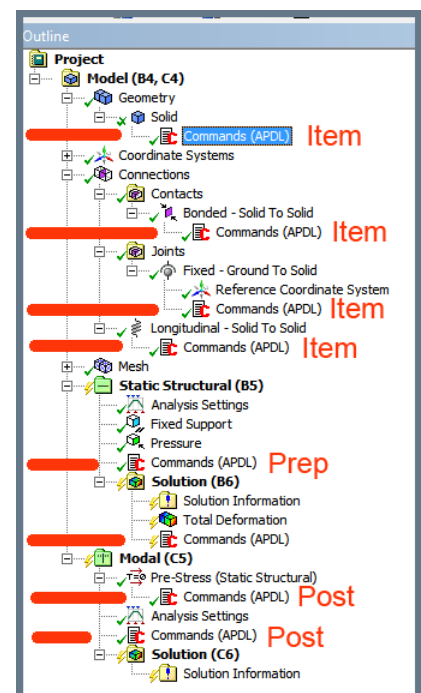
Customization Tool 1: Command Snippets for Controlling the Solver

Not every capability that is found in ANSYS Mechanical APDL is exposed in the interface for ANSYS Mechanical. That is not a problem because users can use the APDL language in ANSYS Mechanical to access the full capability of the solver. These small pieces of code are called Snippets and they are added to the tree for your ANSYS Mechanical model. When the solver file is written, ANSYS Mechanical inserts your snippets into the command stream. Simple and elegant.

PADT has a seminar from back in 2011 that lays it all out. You can find the PowerPoint Presentation [here](#). We do have plans to update this webinar soon.

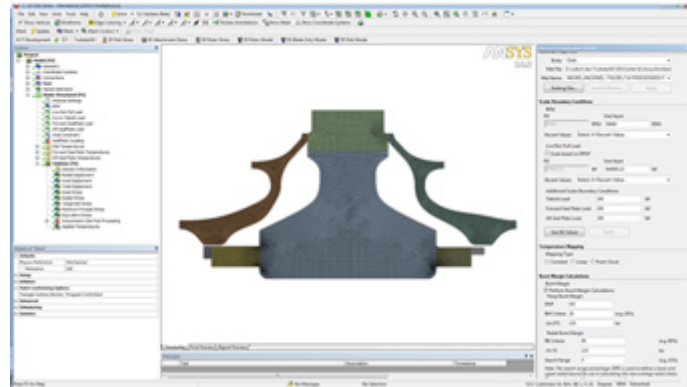
This approach is used when you want to access capabilities in the solver that are not supported in the interface but you want to get to those features and keep track of them from inside your ANSYS Mechanical Model.

If you are not familiar with APDL, find a more “seasoned” user to help you. Or you can teach yourself APDL programming with [PADT's Guide to APDL](#).



Customization Tool 2: ANSYS Customization Toolkit (ACT) for Controlling the User Interface and Accessing the Model

As mentioned above, ANSYS Mechanical is used to define the model and review results. The ANSYS Customization Toolkit (ACT) is how users customize the user interface, automate tasks in the interface, add tools to the interface, and access the model database. This type of customization can be as simple as a new feature, presented as an app, or it can be used to create a focused tool to streamline a certain type of simulation – what we call a vertical application.



A Vertical Application Written in ANSYS ACT by PADT for Automating the Design of Turbine Disks

Unlike APDL, ACT does not have its own language. It uses Python and is a collection of Application Programmer Interface (API) calls from Python. This is a very powerful toolset that increases in capability at every release. PADT has written stand alone applications using ACT to reduce simulation time significantly. We have also written features and apps for ourselves and users that make everyday use of ANSYS Mechanical better.

Do note that ACT is supported in most of the major ANSYS products and more capability is being added across the available programs over time, not just in ANSYS Mechanical. You can also use ACT to connect ANSYS Mechanical to in-house or 3rd party software.

Because this is a standard environment, you can share your ACT applications on the ANSYS App Store found [here](#). Take a look and you can see what users have done with ACT across the ANSYS Product suite, including ANSYS Mechanical. PADT has two in the library, one for adding a **PID controller** to your model and the other is a tool for **saving** your ANSYS Mechanical APDL database.

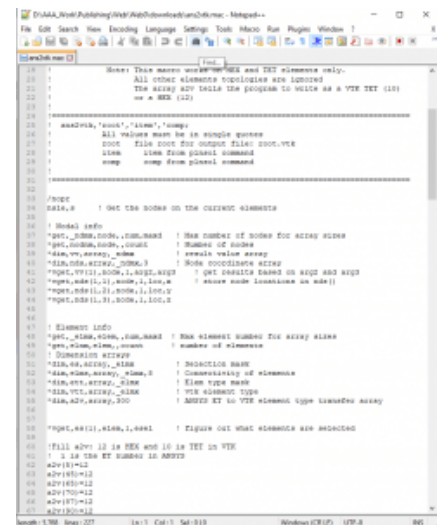
Another great aspect of ACT is that it is fully documented. If you go to the Customization Suite documentation in the ANSYS help library you can find everything you need.

Customization Tool 3: APDL for Automating the Solve

With Code Snippets we talked about using APDL to access solver functions from ANSYS Mechanical that were not supported in ANSYS Mechanical. You can also use APDL to automate what is going on during the solve. Every capability in the ANSYS solver is accessible through APDL.

The most common usage of APDL is to create a tool that solves in batch mode. APDL programs are used to carry out tasks without going back to ANSYS Mechanical. As an example, maybe you want to solve a load step, save some information from the solve, export it, read it in to a 3rd party program, modify it, modify some property in your model, then solve the next load step. You can do all of that with APDL in batch mode.

This is not for the faint of heart, you are getting into complex programming with a custom language. But if you take the time, it can be very powerful. All of the commands are documented in the ANSYS Mechanical APDL help and details on the language are in the ANSYS Parametric Design Language Guide. The PADT Blog is full of articles going back over a decade on using APDL in this way.



Customization Tool 4: User Programmable Features in the Solver

One of the most powerful capabilities in the ANSYS Mechanical solver is the ability for end-users to add their own subroutines. These User Programmable Features, or UPF's, allow you to create your own elements, make custom material models, customize loads, or customize contact behavior.

```
C*****
C
      keycut    = 0
      dsigdep   = ZERO
      pleq_t    = ustatev(1)
      pleq      = pleq_t
C *** get Young's modulus and Poisson's ratio, initial yield stress and others
      young     = prop(1)
      posn      = prop(2)
      sigy0     = prop(3)
C *** plastic strain tensor
      call vmove(ustatev(2), epsPl(1), ncomp)
C *** calculate plastic slope
      dsigdep   = young*prop(4) / (young-prop(4))
      twoG      = young / (ONE+posn)
      threeG    = ONEHALF * twoG
      elast1=young*posn/((1.0D0+posn)*(1.0D0-TWO*posn))
      elast2=HALF*twoG
C *** define tsstif(1) since it is used for calculation of hourglass stiffness
      tsstif(1) = elast2
C
```

There are other general purpose FEA tools on the market that heavily publicize their user elements and user materials and they try to use it to differentiate themselves from ANSYS. However, ANSYS Mechanical APDL has always had this capability. Many universities and companies add new capability to ANSYS using this method.

To learn more about how to do create your own custom version of ANSYS, consult the Programmer's Reference in the ANSYS Help. PADT also has a webinar sharing how to make a custom material [here](#).

Next Steps

The key to successful customization ANSYS is to know your options, understand what you really want to do, and to use the wide range of tools you have available. Everything is documented in the help and this blog has some great examples. Start small with a simple project and work your way up.

Or, you can leverage PADT's expertise and [contract with PADT](#) to do your customization. This is what a half-dozen companies large and small have done over the years. We understand ANSYS, we get engineering, and we know how to program. A perfect combination.



Regardless of how you customize ANSYS Mechanical, you will find it a rewording experience. Greater functionality and more efficient usage are only a few lines of custom code away.

Share this:



Like this:

Like

Be the first to like this.

Related

[How to use APDL Command Snippets in your ANSYS Mechanical Model](#)
July 8, 2019
In "News"

[10x with ANSYS 17.0 – Get an Order of Magnitude Impact](#)
February 3, 2016
In "The Focus"

[Updates & Additions in ANSYS Mechanical 2019 R3 – Webinar](#)
October 24, 2019
In "Webinar"

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

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

< **Presentation: If you are not scared, you are doing it wrong! Successful founders take risks**

“Equation Based Surface” for Conformal and Non-Planar Antenna Design >

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

Subscribe

UPCOMING EVENTS

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

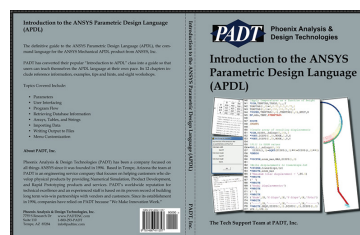
04/04/2022 - 04/07/2022
[37th Space Symposium](#)
[Arizona Space Industry Booth](#)

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.

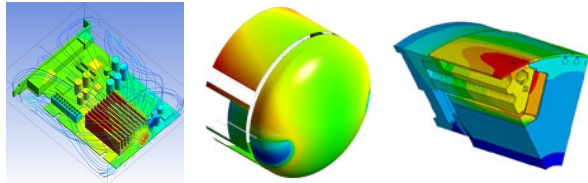
Now available on Kindle as well as paperback.

[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

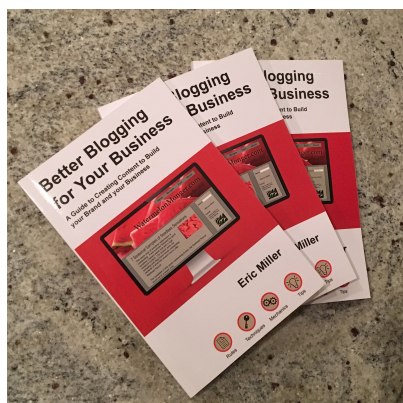
The Focus

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:

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

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!

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?

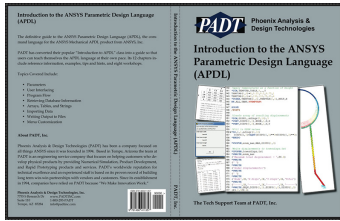


Workstations,
Servers, and
Clusters
designed by

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

Webinar



5