ANSYS Mechanical Scripting: HOWTO Part 1

by **Matt Sutton** 📅 November 10, 2010 🕐 11:18 am   💬 **Leave a comment**   🔖 **The Focus**

Background

Over the past 10+ years, ANSYS has steadily migrated more and more functionality into the DesignSpace, wait Workbench… no I mean Workbench Simulation or rather now ANSYS Mechanical product line. If you've used ANSYS in the last 5 years you've probably come to know this interface as Workbench, and you've probably delineated it in your mind as standing in opposition to the old interface, "Classic" or Mechanical APDL. Perhaps these qualities of each interface come to mind for you as they do for me

Mechanical APDL is:

1. Old looking and visually clunky.
2. Quirky, but powerful.
3. The only thing "real" analyst use.
4. Scriptable, Scriptable and More Scriptable.

ANSYS Mechanical is:

1. Cool, slick looking user interface
2. Super easy to use when your analysis is supported out-of-the-box.
3. Great meshing and geometry support.
4. Not scriptable.

Fortunately for us the margin in capabilities between Mechanical APDL and ANSYS Mechanical is narrowing at each release and so more and more "real" analyst are turning to ANSYS Mechanical for their simulation needs.   However, one glaring difference between the two products that has remained virtually unchanged since the beginning is in the area of scripting.   Mechanical APDL is scriptable at its core. In fact, that is the only way I personally interact with the program, which I'm sure is true for many of you as well. ANSYS Mechanical on the other hand **appears** as though it is just an impenetrable black box. It does what it does and that is that. Fortunately, that is only partially true. The reality is there is a whole underworld that exists in ANSYS Mechanical that is scriptable, it is just very cleverly and discretely hidden. (i.e. It isn't documented) ANSYS, Inc. plans to keep it that way the best I can tell. However, I've spent the last few months of my life painstakingly pulling back the layers and peeking inside. So, I hope to show you some interesting tidbits I've found along the way.

Anatomy of DesignSpace

Why am I going back to calling ANSYS Mechanical DesignSpace? Well, as you might imagine, developers aren't quite as flexible as the marketing types, so a lot of the code that is ANSYS Mechanical still carries the moniker DesignSpace, or DS. You can imagine the mutiny that would occur, if every time a suit got the bright idea to change a product name, all the developers were required to rewrite their code to reflect the new name. It ain't happening.

So, what does DS look like on the inside? Well, the best I can tell it is architected this way:
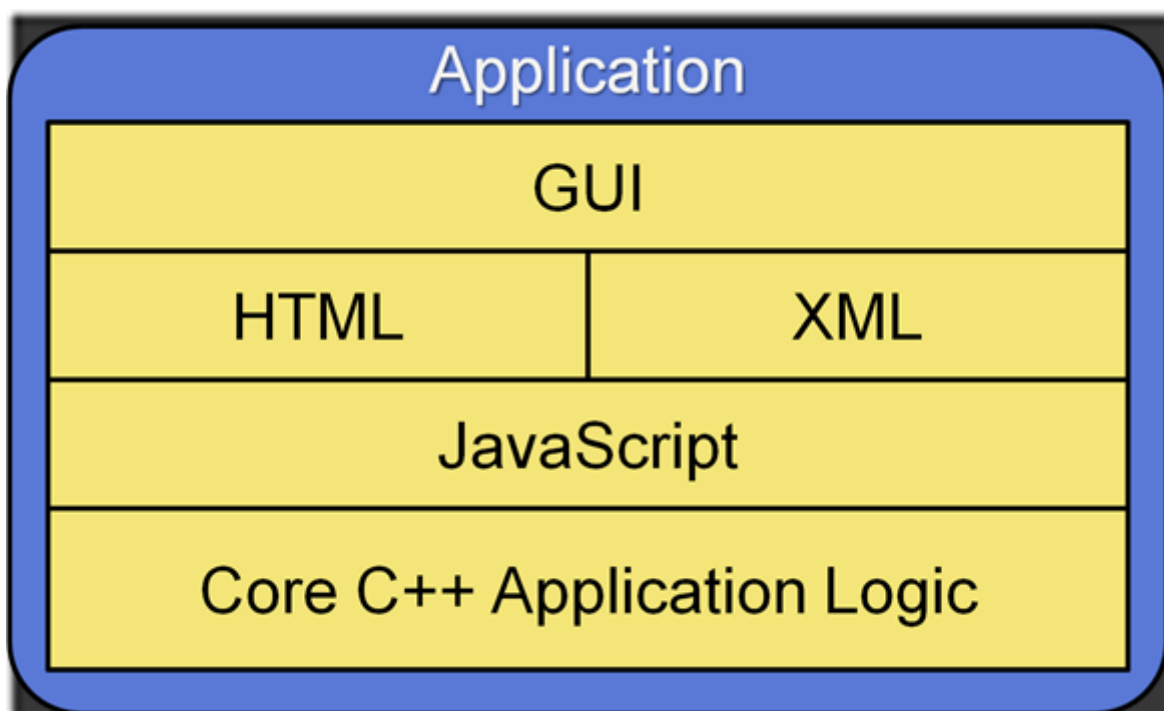
The outer shell (aka the impenetrable black box) is the GUI. Obviously, this is how the user typically interacts with the program. However, the GUI itself is not a statically compiled piece of executable code. It is more like an interpreter that builds itself every time you launch ANSYS Mechanical. (You may be asking yourself, "Self, is this why it takes so bloody long to load…" Self replies "Perhaps… BTW, I need more coffee.") So, the next logical question is "How does it know what to build?" I'm glad you asked, because this turns out to be one of the keys to unlocking the black box. **The structure of the ANSYS Mechanical GUI is described in a handful of XML files.** This is important for a couple of reasons:

1. XML is just text, so you don't have to put on your binary glasses to read it.
2. XML has structure. Brains really like structure.

It may seem like I'm getting nowhere fast with this, but hang with me. Where we are at now is that the structure of the GUI is described by an XML file. So what that means is that the text associated with all of the menus and buttons in the GUI; whether a GUI entity is a toolbar, or a toolbar button or a menu item; all of that is described textually inside these XML files. However, a GUI that doesn't interact with the user is just a G. How does the UI come into play?

I'm glad you asked that as well. The interactive part of the ANSYS Mechanical UI is handled primarily through the use of javascript. Javascript is an interpreted scripting language that is most popular in the context of web development. It is text based, i.e. it isn't compiled. The way it works is that there is a javascript interpeter built into the ANSYS Mechanical executable. Almost all of the core functionality of ANSYS Mechanical is implemented as a set of C++ libraries. These are just great big pieces of code like meshing, or virtual topology, or graphics or geometry selection that expose a set of routines that hook into the javascript interpreter. All of these functional pieces are then glued together with javascript. Thus, it turns out that the UI part of the ANSYS Mechanical GUI is just a whole bunch of javascript code juggling all of these big pieces of functionality that are implemented in C++. It is not completely unlike Mechanical APDL, where a function like ASBW hides a complex surface-to-surface intersection routine followed by a BREP patchup; however, we simply know it as "Divide these areas by the working plane." In the same way there are various javascript calls that a script can issue that will insert a new boundary condition, for example. Or, perhaps, change a mesh control and re mesh.

Finally, I'll leave you with one last piece of the puzzle for this post, then we'll pick it up with examples in the next post.   You remember that I said that the GUI is cooked up each time using an XML file as the recipe. Inside that XML file are definitions for javascript functions that are called whenever a user clicks on a button, or selects a menu item. This provides the link between the static GUI structure and the dynamic user interaction. is how we turn the key to unlock the black box. By searching in the xml file to find a

object that does what we want, we can then determine the javascript function that is called when the user interacts with that GUI object. By finding the javascript function name, we can find the function implementation inside the ANSYS installation directory. By finding the implementation, we can study it and figure out what it does.

Stay tuned for the next installment to see this in action.

Share this:

Like this:

Loading...

Related

Utilizing Element Birth and Death for Contact Elements in Workbench Mechanical
March 23, 2012
In "The Focus"

ANSYS Mechanical Scripting: HOWTO Part 2
December 15, 2010
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.

‹ **FE Modeler Part 1: Translating Meshes with ANSYS FE Modeler**

Using the 'Worksheet' Page in Mechanical ›

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

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

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
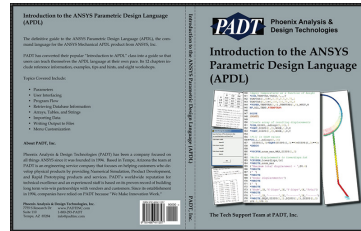
04/12/2022 - 04/14/2022
D&M West | MD&M West

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.
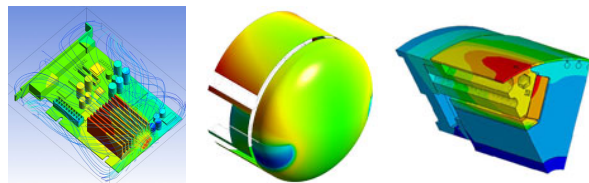
Now available on Kindle as well as paperback.

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
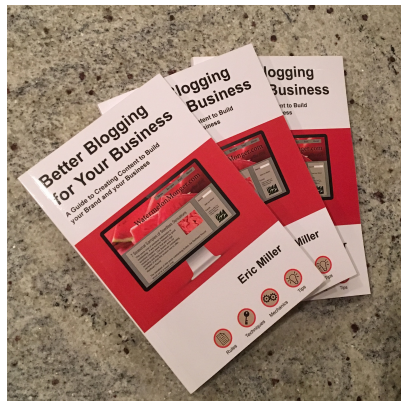
Podcast

Product Development

Publications

Startups

Stratasys

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

Privacy - Terms

*Email

[Email input field]

State/Province

[State/Province input field]

Company

[Company input field]

*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 input field] 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.
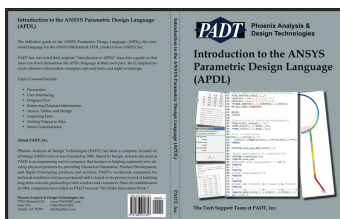
TRYING TO FIND A COMPUTER BUILT FOR SIMULATION?

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

Connect with PADT

Privacy - Terms