The Focus
PADT
February 28, 2006 | A Publication for ANSYS Users | Issue 45

Modeling the “Plastic Knee” in Workbench

By Doug Oatis

So you’ve mastered the basics of Workbench and are ready to tackle a plastic analysis. If you look at your stress-strain curve and it looks like a bilinear model is not going to work, you need to be aware of how the WB multilinear model works to make sure you are capturing all of your plastic behavior, especially if your material has a big “plastic knee”. To understand the why and how, we first need to look at the way it used to be done in ANSYS.

In olden days, you gave the elastic behavior as Young’s modulus (EX), a yield stress, and the whole stress-strain curve. If the equivalent stress was under the yield stress, ANSYS calculated strain using the Young’s modulus. If it was over, it used the curve. There were three rules that you had to meet:

1: The initial slope of the curve MUST equal the Young’s modulus

2: The slopes of all the multi-linear segments must have a slope less than or equal to the Young’s modulus

3: No segment can have a slope of less than zero

The first rule proved to be especially troublesome. But if you didn’t have it, ANSYS would predict a jump in strain when a stress moved from below to above yield. Even the best analyst would get models that would not solve because the first slope was slightly off of the Young’s modulus. This approach used a TB,MISO material property to define multi-linear plasticity.

Based upon this issue and several others, the new multilinear plasticity model, TB, PLAS was developed so that users only had to define the plastic portion of the stress-strain curve. You do this by defining the Total Strain - 0.2% (Yield Strain) for the x-axis and the total stress for the y-axis. Since this is simpler and produces a lot less user frustration, it was chosen for the multi-linear plasticity model in Workbench.

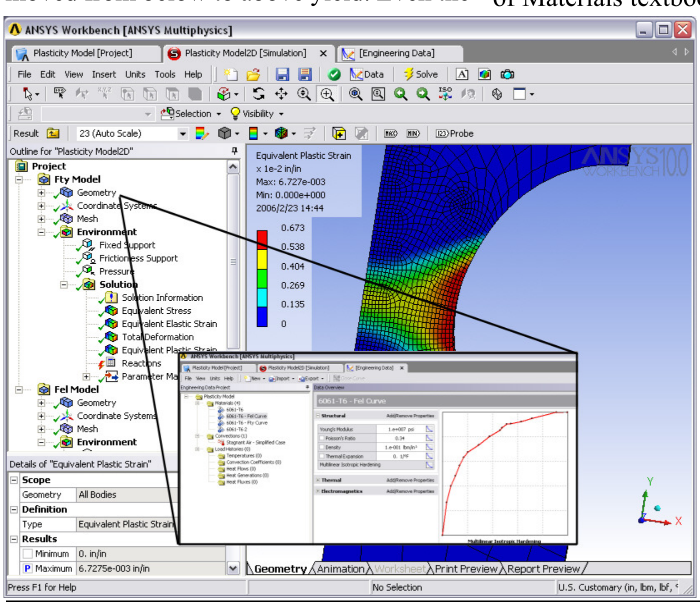
So for Workbench the following rules apply:

1: The first data point must be at 0 for plastic strain (X) and the Yield Stress for stress (Y)

2: The slopes of all the multi-linear segments must have a slope less than or equal to the Young’s modulus

3: No segment can have a slope less than zero

So instead of calculating a total strain from a curve when stress is above yield, the solver calculates a plastic strain and adds it to the elastic strain. For many materials this is perfect and saves a lot of headache. But, if you can remember back to your Strength of Materials textbook (like “Introduction to Mechanics of Solids”, Popov, Pg 111) a lot of materials show behavior like the curve shown in Figure 1. Pulling out a definitive yield point is tough so a 0.002 in/in offset of the linear portion of the curve is used to set a yield stress.



So, if your material doesn’t have a large non-linear “knee” before the yield stress, you can move on to reading other articles in this issue. But if your material shows this behavior, you should be using the elastic limit or proportional limit (which are often very close), as your yield stress and use that point as zero for your strain.

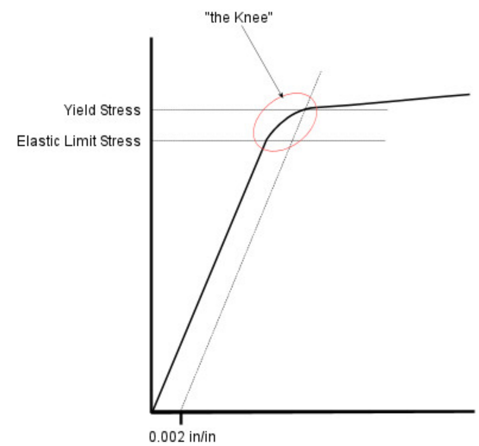


Figure 1: Typical Stress Strain Curve

If you do this, however, you run into two problems. The first is that you need to redefine the yield strength of your material in Workbench. This is because Workbench does an internal check to make sure all rules are met with the plastic curve. If the predefined yield stress doesn’t equal the initial y-value of the plasticity curve, the solve won’t be executed. This problem gets even more complicated if you have a stress tool calculating margin off of the yield strength, since your yield strength is now actually your elastic limit. You will need to set the criteria to calculate margin from a user-defined value.

The second problem this creates is that the elastic limit is not a commonly listed material property. Finding this point on an actual curve obtained from MIL-HDBK can prove to be a long and tedious task.

(Continued on Page 2)

Contents

("Plastic Knee" cont.)

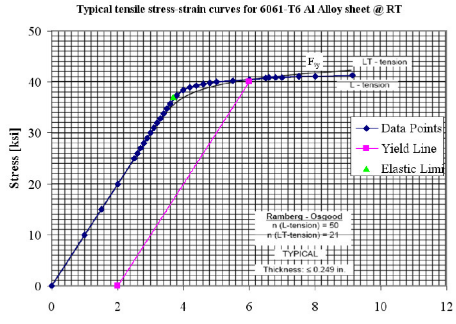


Figure 2: Material Curve Used

So now you must be asking, "Well, is it really worth the extra effort to use my starting point as the elastic limit instead of using the easy-to-obtain 0.2% yield stress?"

In order to answer this, I created three models:

Platform	Plasticity Model	Starting Point
Workbench	PLASTIC,,,MISO	$0,F_{ty}$
Workbench	PLASTIC,,,MISO	$0,F_{el}$
Classic	MISO	0,0

The same geometry, element types, and loading were used for each model. The ANSYS Classic model used the tried-and-true method which requires the first slope equal the Young's Modulus. Aluminum shows a significant "knee" between F_{el} and F_{ty} so it was chosen for this study. Figure 2 shows the data used and is based on a MIL-HDBK-5 graph.

The results between the models are shown below in Figures 3 and 4.

As you can see, all three models obtain similar strain results. However, when equivalent stresses are viewed, the Workbench model which used $0,F_{ty}$ as it's starting point is different from the other two, and by a considerable amount.

So, after all is said and done, if you want to more closely match ANSYS Classic methodology, you need to define $0,F_{el}$ as your starting point. Just ignore the values on the x-axis and define the non-linear portion of the stress strain curve.

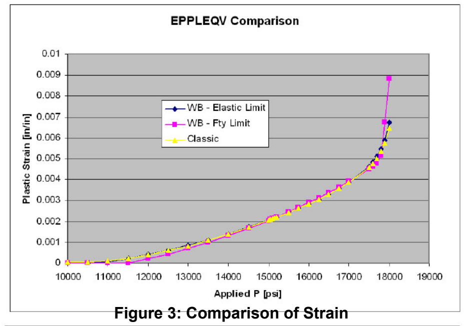


Figure 3: Comparison of Strain

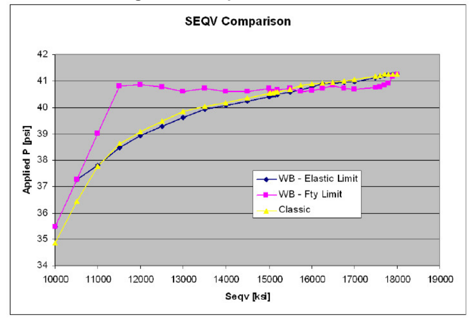


Figure 4: Comparison of Stress

Reviewing Substructuring

By Rod Scholl

Substructuring, once a cornerstone of FEA system analysis, has continued to fade in popularity as computer speed and available memory have increased dramatically. It still has its place in structural analysis, but it is definitely diminished. And when this technique is applicable, the savings can be huge and makes a nice implement on the Bat Belt of the FEA analyst.

What it isn't

First let's note that performing a substructred modal anlaysis (prerequisite for many types of dynamic analysis) requires Component Mode Synthesis (CMS) and although a type of substructuring, is not covered in this article, but was covered in the a previous issue Further, the term "submodeling" is often confused with the term "substructuring". Submodelling refers to the method of taking a coarsely-meshed solution, remodeling a portion of it , meshing with greater refinement, and applying the results of the previous run as boundary conditions (CBDOF) to the new model and resolving.

What it Is

Substructuring is about breaking a model up into a set of smaller models, or "superelements" and then solving them simultaneously. Since we know solution time is roughly related to N^3 for (NxN) matrices… a method to break a matrix into say, quarters, and solve four (N/2 x N/2) matrices can represent significant savings in solution time and memory requirements. Of course these matrices have to be relatively independent of one another as we will see. Also, for some nonlinear problems, the predominately linear portion can be scuttled away as a substructure to be solved essentially once, leaving only the nonlinear portion to be iterated upon. Afterward the whole solution is reassembled. Of course not all problems lend themselves to substructuring… (if not, why would one ever mess with the huge NxN global stiffness matrix?!) So expect that only a limited class of problems are benefited by the substructuring method.

How It Works

The help manual is decent in this respect… but with all the files and naming to keep track of, an extra example is useful. (attach macros 1,2,3). These three macros can be run sequentially to model two blocks pushed into contact. The majority of the block is sub-structured, leaving only the contact surfaces to be solved during the contact iterations. The only real trick to substructuring is getting the file naming correct. There are a bunch of ways to do this.

Again, read the manual for more details, but substructuring analysis can be broken into these steps:

Get the macros at:
ftp.padtinc.com/public/downloads

1) TSUB1.mac – Generation Pass
Define how the model will be sliced up into chunks, and form a Matrix50 Super-Element (couldn't resist making the graphic here, get it, "Super Element") of each chunk. What exactly is a super-element? Well, you know that feeling when you lean back in a chair… and you lean back too far, and you thrash

(Continued on Page 3)

quickly and barely recover? That was a little help from the often invisible, but ever-present Super-Element… fighting evil in our daily lives. Okay, okay, it's an element with varying number of DOF's at the boundary it encloses which has an equivalent stiffness for each DOF. What makes the savings, is that only the stiffness at the boundary DOF's need to be carried… or more accurately, at the Master DOF's defined at the boundaries.

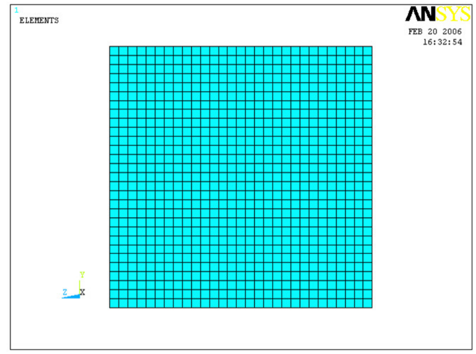
You don't actually need each node at the boundary to be a Master DOF node… but realize force/deflections only transfer at the Master DOF's, so I can't think of a case where a coarse mesh with every node defined as a Master isn't superior to a fine mesh with only some boundary nodes defined as Masters.

2) TSUB2.mac – Use Pass
Solve the model of superelements and any elements that were not put into superelements.

3) TSUB3.mac – Expansion Pass
If you need to know stresses and deflections for the regions inside the superelement, you run an "expansion pass" which takes the system level results at the superelement boundaries and applies them to the superelement and solves. Some people still call this an "Unwind" pass.

Another Example:
These 3 macros are very similar to TSUB1-3… except they do not have the contact elements, rather the blocks are connected via couples, and only one of them is placed inside the superelement.

The Big Caveat
The size of the boundary between substructured regions is the FIRST thing to consider when determining if substructuring is right for you. This often overlooked step (guilty) has sent many folks down the substructuring road only to find longer solves times, or complete failure of runs due to explosively increasing RAM requirements. As we mentioned, if breaking a simple cube into 8 smaller cubes actually saved time and memory… then that would already be in the ANSYS solver! The cases we are looking for, is where regions of the model are relatively independent and share only a few nodes with one another, but yet the regions are needed in their fully geometric detail and complexity for proper system behavior -- or, also cases where the non-linearity is localized, such as resolving plasticity in a small fillet, or small regions of contact between many relatively large rigid bodies, etc.



There is a maximum wavefront limitation of 46340 at the superelement boundary. But on many machines RAM limitations will run out first. On my 2GB Ram, WINXP machine the maximum size of the boundary that I could solve was about 3,500 nodes (3 Master DOF's at each node). There wasn't much improvement going to a 6GB 32-bit Linux machine.

On the 6GB Ram 64-bit Linux machine, I'm not sure the maximum size of the boundary but hit a 8GB file size failure at about 10,000 nodes (30,000 MDOF's).

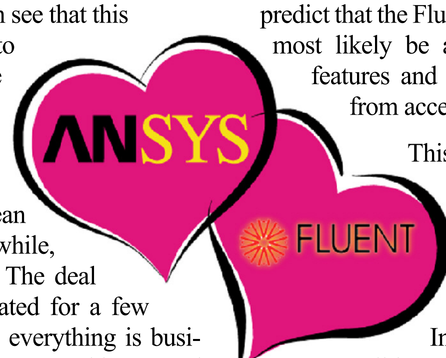
So on my machine, for example, we're talking a maximum interface of about 30x30 mid-side nodded elements. Results will vary depending on shape of the region, and of course your hardware.

Commentary: ANSYS Inc's Purchase of Fluent

By Eric Miller

They overshot Valentine's Day a bit, but ANSYS, Inc. just announced their intent to purchase the Fluent Software business from AAvid Thermal Technologies. Since the deal is not completed there is not a lot we can say about it. But we can comment on some of the facts that are included in various press releases.

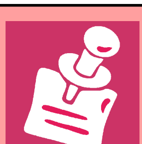
The first comment is that this deal is big. Looking at the numbers, Fluent has the largest share of the CFD market. The Fluent web site quotes Daratech as giving them "over one-third of the estimated global CFD market, and is twice as big as its nearest competitor." That nearest competitor is probably ANSYS/CFX or CD-Adapco, depending on what you count as revenue. According to published numbers, Fluent's 2005 revenue was around $121 million from software and services. This is a huge increase in market for the new company. Looking at ANSYS, Inc.'s 2005 revenue of $153.3 million you can see that this purchase is going to greatly expand the number of customers under the ANSYS umbrella.



So what does this mean to the users? For a while, nothing will change. The deal will not be consummated for a few months and until then everything is business as usual. After that, past history and official comment points toward taking their time on integrating different parts of the business. What we have seen in the past is that the first benefit to users is the sharing of brain power across companies. Published comments predict that the Fluent "development plans will most likely be augmented with additional features and capabilities that will come from access to ANSYS technology."

This will take some time to do correctly but a few years down the road there should be some awesome things happening. In the end, ANSYS, Inc. users will have a compatible and robust CAE solution that will be able to tackle almost any simulation problem.

Learn more at : media.corporate-ir.net/media_files/IROL/11/118715/021606.pdf



Resources

View movies that show the functionality of ANSYS, Inc products at the ANSYS Demo Room www-harwell.ansys.com/demoroom

A new book, "The Finite Element Method and Applications in Engineering Using ANSYS" is now available from two University of Arizona Professors. Learn more at Amazon.Com

An Engineer's Travelogue:

Anticipating the 2006 International ANSYS Conference

By: Egbertius Fog, MBE, PE

Those of you have been faithful readers of my travel stories know that I love a good jaunt to an exotic local that features fascinating technology. In your letters you have expressed your appreciation for tales of my close encounter at the Balanese Robotics Plantations and my four weeks at the South Pole studying the bridge building habits of Poindexter Penguins. It is therefore with a delicious anticipation that I tell you about my next adventure: a journey to the wilds of Pittsburgh to partake in that biennial congress of adventurers and risk takers: the 2006 International ANSYS Conference.

This congress of capable calculators supplies that unique combination of thrill and knowledge that causes my adrenaline to surge just thinking about it. The roller coaster ride begins when we leave the relative civility of the airport and journey towards the hidden splendor of the city itself. Perched at the confluence of two great and wild rivers, Pittsburgh is shielded from the outside world by high bluffs that are magnificently pierced by long, tile-lined tunnels that are vivid reminders of adolescent locker room nightmares. Bursting from the tunnels into the light of day you are greeted by one of the finest collections of steel bridges in the world. In a fascinating array of shapes and colors, they leap across the surging rivers linking the wild center of the city with the outside world.

And what a center it is. Buildings of metal, glass and stone surge upwards to great a leaden and cloudy sky. If you are lucky, a slight rain will be falling, just enough to impart a sense of scale and proportion to the amazing tableau before you.

In and amongst this urban jungle sits a marvel of modern engineering: the convention center. Featuring cantilevered terraces and soaring cables that anchor the roof, this building is a showplace of "green" design that manages to merge function and beauty into one. But what excites me as I sit down to prepare with my trusted outfitters is not the wild landscape it is to be held in, and not the beauty of the structure that will be my home for five days. No, what excites me is the chance to meet the members of the various ANSYS, Inc. tribes.

Sir Fog's pre-journey discussion will be continued in the next issue of The Focus under the title:
"Interacting with the Natives and Understanding what they Worship"

The Top 10 Reasons
Why You Should Attend
IAC'06

10: Meet developers face to face and discuss your needs and concerns

9: Learn about how other companies are using ANSYS products

8: Be able to finish a story with "he meshed his sheet metal parts with Tets!" and get a laugh every time

7: Attend presentations that focus on real world usage of ANSYS.

6: Network with potential future bosses... not that your intent is to find new employment, but you never know...

5: Meet John Swanson

4: Take training classes from experts

3: You can have dinner with people who not only know what a Newton-Raphson is, but they actually want to talk about it at dinner

2: Voice your opinions to people who can actually make a change.

1: It's in Pittsburgh, your spouse won't get suspicious

Visit:
www.ansys.com/conf2006
to Register

Upcoming Training Classes

Month	Start	End	#	Title	Location
Mar '06	6-Mar	8-Mar	101	Introduction to ANSYS, Part I	Albq, NM
	9-Mar	10-Mar	102	Introduction to ANSYS, Part II	Albq., NM
	16-Mar	17-Mar	203	Dynamics	Tempe, AZ
	22-Mar	24-Mar	104	ANSYS Wrkbench Sim. Intro	Tempe, AZ
	27-Mar	29-Mar	902	Multiphysics Sim. for MEMS	Tempe, AZ
Apr '06	3-Apr	5-Apr	101	Introduction to ANSYS, Part I	Tempe, AZ
	6-Apr	7-Apr	107	ANSYS Wrkbnch DsgnMdlr	Tempe, AZ
	10-Apr	12-Apr	401	Introduction to Electromagnetics	Tempe, AZ
	13-Apr	14-Apr	604	Introduction to CFX	Tempe, AZ
	18-Apr	19-Apr	201	Basic Structural Nonlinearities	Tempe, AZ
	20-Apr	21-Apr	204	Adv. Cont. & Bolt Pretension	Tempe, AZ
	26-Apr	28-Apr	152	AI*Environment	Tempe, AZ

Links

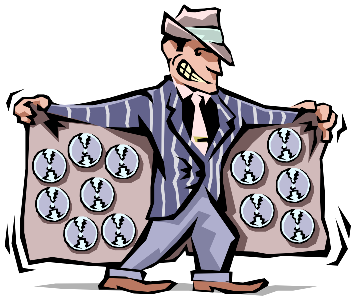
The Home for all things ANSYS on the web is:
www.ansys.net

Looking for News on Technology in Arizona?
Visit Jeff Strain's "Arizona Technology Roundup"
aztechroundup.blogspot.com

News

- Starting with Version 11.0, ANSYS, Inc. Will Ship Software on DVD's
- ANSYS Completes Fourth Quarter and Full Year 2005 With Record Revenue and Earnings

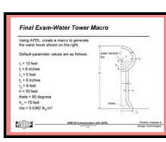
The Focus is a periodic publication of Phoenix Analysis & Design Technologies (PADT). Its goal is to educate and entertain the worldwide ANSYS user community. More information on this publication can be found at: http://www.padtinc.com/epubs/focus/about



The Shameless Advertising Page

Outsourcing your Analysis to PADT Puts Food in this Child's Mouth

Call 1-800-293-PADT or e-mail info@padtinc.com Today



Learn APDL the "Home Schooling" Way
Purchase PADT's
Guide to ANSYS Customization with APDL

ANSYS®

Enter a World of Simulation ...

... at the 2006 International ANSYS Conference, May 2 through 4 at the David L. Lawrence Convention Center in Pittsburgh, Pennsylvania, U.S.A. The conference will focus on unlocking the potential of simulation to streamline design processes and showcase advances in computer-aided engineering and related technologies.

Keynote Speakers

Is your organization struggling in its attempt to create more innovative solutions?Join us to learn from the successes of these keynote speakers:

- **Sophie Vanderbroek**, chief technology officer and president of Xerox Innovation Group, **Xerox Corporation**
- **Kyril Faenov**, director, high performance computing, **Microsoft**
- **John Gross**, research structural engineer, **National Institute of Standards and Technology**
- **Graham Hawkes**, ocean engineer, inventor and founder, **Hawkes Ocean Technologies**
- **Dale Preece**, distinguished member of technical staff, **Sandia National Laboratories**

Technical Track

Learn firsthand from product development professionals in specific industries: aerospace and aviation, automotive and transportation, biomechanics, chemical process and petroleum, civil engineering, computers and software, consumer products, education, electronics and electrical, MEMS, nuclear and power generation.

Register today at:
www.ansys.com/conf2006

Sponsors

ANSYS

Platinum

Microsoft Windows Compute Cluster Server 2003

Gold

Autodesk
hp invent
sgi

Silver

AMD
ATI FireGL
DELL
i*hydra
intel
moldflow