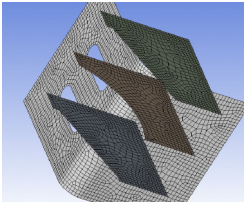


Midsurface Extraction with DesignModeler

By Doug Oatis



One of the better features of Workbench's Design Modeler (besides face delete, face imprinting, slice, etc.) is the Midsurface Extraction tool. This tool is fairly simple to use, and after some practice, you will be creating shell models in no time.

The Midsurface Extraction utility is located underneath the 'Tools' Menu (or you can put it on your toolbar using the options menu). The easiest way to create midsurfaces is to manually go through and select the face pairs. You will notice that after selecting the "top" and "bottom" face, they will appear as two different colors. It is important to note that the order in which you click the two faces determines the 'top' and 'bottom' of the shell element (important for applying pressures/defining contact). The shell normal is positive going from the pink (2nd pick) to the purple (1st pick) surface.

You can select multiple face pairs, then go back and hit the apply button for "Face Pairs" in the detail window. If there are several face pairs that have the same thickness, you can select one pair, then go back and change the selection type to be automatic. This will fill in some blanks in the details window (i.e. maximum and minimum threshold thicknesses). Finally, change "Find Face Pairs Now" to be "Yes" and DM will go out and find all face pairs that meet this criteria. Hit generate and all of your midplanes will be generated, and all thicknesses will be stored for later use in Simulation.

One thing you need to keep in mind while creating midplanes is whether the selected faces belong to the same body or are separate bodies. One example is the creation of a midsurface representation of an L-bracket with ribs. If the rib is included in the L-bracket (unfreeze parts together), then the rib geometry will be automatically extended

to the midsurface of the bracket. If the rib is a separate body, then there will be a gap between the rib and bracket midsurface (bracket thickness/2).

The surface extension tool is an extremely robust way to join midsurfaces together. Simply select the edge of the surface you want to extend and select the method (specific offset, to plane, to face, etc), and the surface will be extended. I've found myself using 'To Surface' most often, since it extends the edge to the projected surface, allowing you to join two midplanes using two surface extensions. Another nice feature of the surface extension tool is that it follows the profile of the existing face. What this means is that if you have a rib with a rounded edge, if you extend the edge, the surface will be extended and follow the pre-existing curvature.

After you've gone through and extracted all your midplanes and done all your surface extensions/modifications, you probably want to join all of them together (i.e. areas have continuous mesh). The first step in doing this is to form a new part of all the areas. Make sure you have your selection filter set to bodies, not faces, and select all the areas you want to join. Right click in the graphics window and select 'Form New Part'. Next, go Tools > Joint, which will insert a joint object in the model tree. All of your areas should be selected from the new part formation, but if not, select the multi-body part, and hit apply in the joint details window. Hit generate and Workbench will highlight all the continuous edges. Make sure you don't create a multi-body part that consists of solids and shells, since this is not allowed in Simulation (you can have an assembly of shells/solids, just not a single part consisting of both).

When you bring this geometry into Simulation and mesh it, all shell/beam elements will be shown with their real constant set turned on (/eshape,1). There is no toggle for this behavior, so you must manually go into the geometry branch of Simulation and

Details of MidSurf7	
Mid-Surface	MidSurf7
Face Pairs	6
<input type="checkbox"/> FD1, Thickness Tolerance (>=0)	0.01 in
<input type="checkbox"/> FD2, Sewing Tolerance (>=0)	0.001 in
Extra Trimming	Intersect Untrimmed w...
Preserve Bodies?	No

Figure 1: Manual Midplane Details View

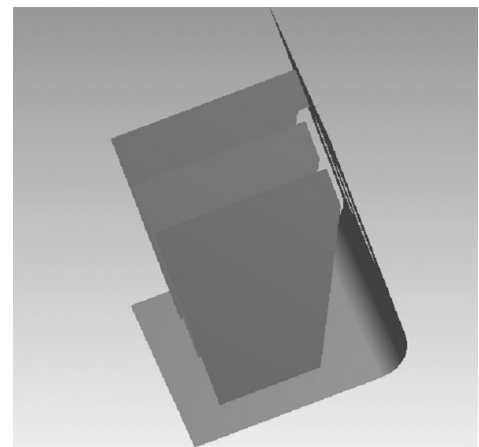
Details View	
Details of MidSurf7	
Mid-Surface	MidSurf7
Face Pairs	Apply Cancel
Selection Method	Automatic
Bodies To Search	Visible Bodies
Minimum Threshold	0.25 in
Maximum Threshold	0.5 in
Find Face Pairs Now?	No
<input type="checkbox"/> FD1, Thickness Tolerance (>=0)	0.01 in
<input type="checkbox"/> FD2, Sewing Tolerance (>=0)	0.001 in
Extra Trimming	Intersect Untrimmed w...

Figure 2: Automatic Midplane Details View

set the thicknesses to be 0 to display the planar shell elements.

If you have an assembly of shells and solids, Simulation does not automatically detect

(Continued on Page 2)



Contents

Midsurface Extraction.....	1
Sub Modeling.....	2
MOR for ANSYS.....	4
ANSYS Conference Travel.....	5

("Midplane" cont.)

edge/face contact (unless you've changed this setting in 'Options' beforehand). You can change this option in the contact branch details window, then regenerate all contact pairs. If you want to define edge/face contact, the edge must be the contact and the face must be the target. If you have shell face/edge or shell face/solid contact, pay attention to the contact normal directions (displayed red and blue). The contact normals are going to be aligned with the shell normals by default (esurf,top), which are determined by the face selection order when the midplane was created. If you need to reverse the normals, you can insert a command snippet under the offending contact pair to reverse the contact and/or target normals.



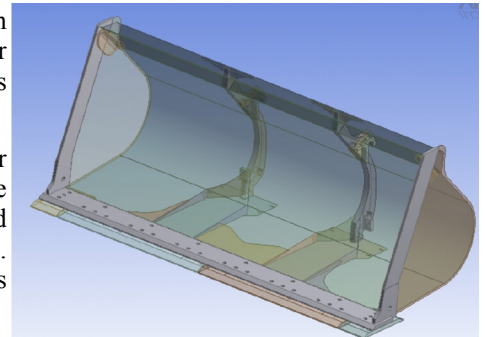
Here are a few of the tips/tricks I've picked up:

- You can easily fill in "voids" by creating sketches and then forming surfaces from sketches (Concept > Sketches from surfaces).
- Don't be afraid to use the slice tool to cut up surfaces, then extend individual surface edges, and then unfreeze the surfaces back into one

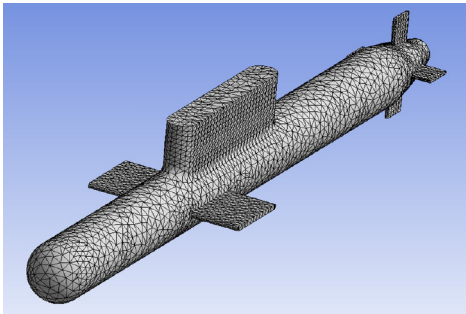
Don't be afraid to create excess areas. I will sometimes overextend an area, slice it in some way, then suppress the area I no longer need. The joint tool created edges and points that can be used for future CAD operations

After enough practice, you can impress your friends and make fairly complex models like a tractor bucket. This mixed shell/solid model was created from a solid CAD model. In case you're wondering, this picture looks great hanging on my fridge...

Workbench (and Design Modeler) of course comes with your ANSYS installation CD's. However there is a license (and associated fee) to use Design Modeler. Talk to your ASD about getting a demo key. With the cost of Mid-Plane extraction (and waiting on the designer to get around to doing it) this feature alone can justify the relatively low cost of Design Modeler.



Sub Modeling in ANSYS



By Ted Harris

A submodel is a refined, separate finite element model that is typically used to model a portion of a coarser finite element model with more detail and accuracy. In the old days when wavefront limits were fairly small, the frontal solver was the only solver, and computer resources were meager by today's standards submodeling was routinely used to get accurate results in complex fillet areas or in other features that just couldn't be meshed with sufficient refinement to accurately predict stresses without ending up with a model that was too big to run.

That scenario doesn't happen as much these days. However, for large models solution time, memory or disk space limitations can still become the limiting factor in getting

accurate results. Submodeling becomes useful especially for simulations that don't lend themselves to substructuring but have very tiny yet important features contained within a much larger geometric envelope. In other words, the overall model cannot be simplified due to boundary condition requirements, but the small features of interest cannot be meshed with sufficient refinement for accurate results without the overall model becoming too big to solve.

In such cases a relatively coarse overall model can be constructed which accurately captures the stiffness and deflections of the structure. A submodel of a small portion of the overall geometry that contains the detailed region of interest is then created. This submodel might have more geometric detail, such as fillets, than in the coarse model as long as the overall stiffness of the structure is not significantly affected. The submodel can have a much finer mesh than the coarse model to help minimize mesh discretization error.

Submodeling takes advantage of St. Venant's principle. Recall that this principle states that once you get away from the immediate vicinity of load application, the stress distribution is the same regardless of

(Continued on Page 3)

Definition

sub \səb\ [L, under] under or below [1]

model \məd-əl\ [L, small measure] A schematic description of a system, theory, or phenomenon that accounts for its known or inferred properties and may be used for further study of its characteristics [2]

submodeling \səb-mäd-līŋ\ A finite element technique that you can use to obtain more accurate results in a particular region of a model [3]. It involves building a more detailed local model with boundary conditions mapped from a larger, typically coarser model. No, it has nothing to do with submarines or The Hunt for Red October, en.wikipedia.org/wiki/The_Hunt_For_Red_October although I must say I enjoyed the book far more than the movie.

substructuring \səb-strək-chə-rīŋ\ An essentially unrelated but similar-sounding finite element concept involving breaking up an FE model into smaller linear 'chunks', which are generated independently but linked through master degrees of freedom. Please see Rod Scholl's Focus article www.padtinc.com/epubs/focus/2006/0045_0228/TheFocus_45.pdf#Page=2 for more information on the applicability and implementation of substructuring.

the manner in which the load is applied. This holds as long as the loading in the submodel is statically equivalent to the loading in the coarse model. The boundaries of the submodel should be sufficiently removed from local stress gradients.

Submodeling in ANSYS is implemented by mapping displacement boundary conditions from the coarse model onto the exterior 'cut' faces of the submodel. This is accomplished by selecting the cut face nodes of the submodel and writing them to a node file with the NWRITE command in /PREP7. Submodeling in ANSYS is accomplished in the General Postprocessor. The displacement boundary condition mapping is accomplished with the CBDSP command in /POST1.

If the coarse model has an applied temperature distribution this can be mapped to the submodel as well. The process here is to write all the submodel nodes to a node file with NWRITE. The temperature mapping from the coarse model is then accomplished with the BFINT command in /POST1. Note that any other loads that were applied in the coarse model should be applied to the submodel as well, such as pressure loading or rotation velocity.

An assumption of the submodeling method is that the analysis is linear. However, if the nonlinear effects are contained within the submodel and do not affect the stiffness of the structure, it is possible to consider them in the submodel. Please see Sheldon Imaoka's excellent article on submodeling with nonlinearities:

ansys.net/ansys/tips_sheldon/STI02_Nonlinear_Submodeling.zip for more information.

Once CBDSP or BFINT are complete, the resulting input files can be read into the submodel database in ANSYS using the /INPUT command in /PREP7. Once any other required loads are applied the solution can be solved and postprocessed in normal fashion.

Let's say our task is to perform a stress analysis on a submarine, with particular interest in the axial stresses where the forward part of the conning tower intersects the hull. Even though submodeling is currently supported only in the ANSYS environment and not in Workbench, we can leverage Workbench to make the coarse model and submodel setup easier. Our initial geometry was constructed in DesignModeler and an enclosure was used to model the water surrounding our submarine, as shown in Figure 1. CFX Mesh and CFX were utilized within the Workbench Environment to perform a CFD analysis on the submarine, results of which can be seen in Figure 2.

Workbench Simulation was then used for the initial stress analysis of the sub, with pressures from the CFX simulation easily transferred to the structural model to provide the structural loading environment. Results of the coarse model stress solution are shown in Figures 3 and 4.

DesignModeler was used again to create the submodel geometry from the full model, as shown in Figure 5.

The submodel was meshed in Workbench Simulation (see Figure 6) and the pressure load was once again easily applied from the CFX Simulation. A Named Selection of the cut boundary faces was created to facilitate selection of the appropriate cut face nodes in ANSYS. The FE model was then exported to ANSYS by highlighting the Solution branch and clicking on Tools > Write ANSYS Input File. This file was then read into ANSYS and the boundary condition mapping was accomplished using the coarse model ANSYS results file. That file is automatically saved during the Workbench Simulation solution as long as the Simulation Solution option to save ANSYS files is checked.

The mapped displacements were then read into the ANSYS submodel database and the submodel was solved and postprocessed in ANSYS, as shown in Figure 7. In all seven submodels with increasing mesh density

(Continued on Page 4)

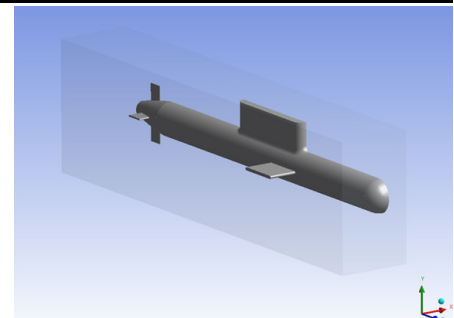


Figure 1: DM Geometry - Submarine and Enclosure

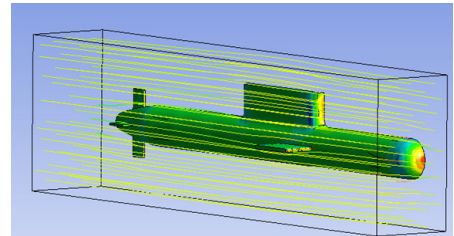


Figure 2: CFX Simulation Pressure Calculations. Running Silent, Running Deep.

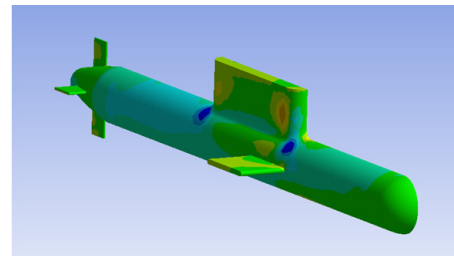


Figure 3: Coarse Model Axial Stress Results

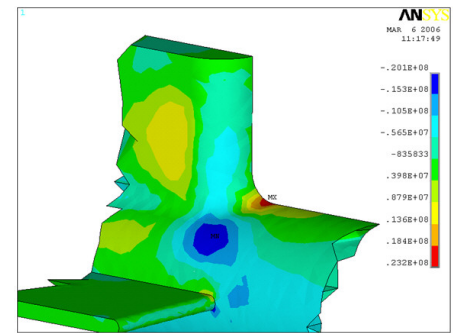


Figure 4: Coarse Model, SZ in Region of Interest (ANSYS)

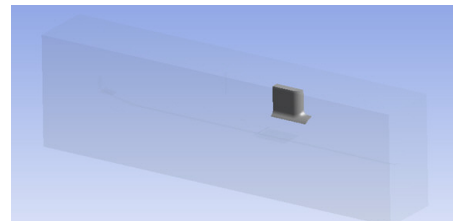


Figure 5: Submodel Geometry for Region of Interest



Resources

Want to get the latest news on CAD/CAM/CAE companies? Get the latest news daily. Subscribe to www.tenlinks.com by filling out the "Subscribe to E-Mail" form.

Answer your technical support questions on your own by accessing the same Database the Tech Support Uses: Log in to the ANSYS Customer portal and visit the Knowledge Base: www1.ansys.com/customer

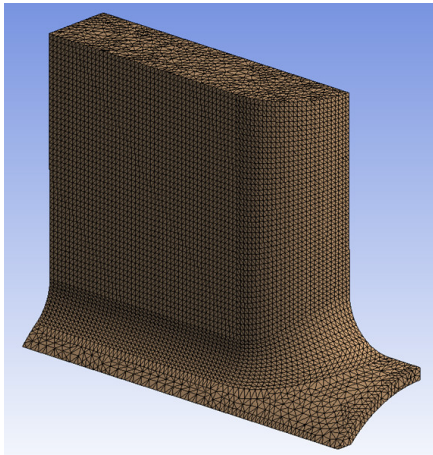


Figure 2: Submodel Mesh Created in WB Simulation

("Sub Modeling," Cont.)

were completed to demonstrate the mesh convergence behavior seen in Figure 8.

So, was a submodel required for our submarine? The answer is probably not since we could have created an accurate enough mesh using local mesh controls in our fillet of interest.

Note that it's also possible to nest submodels to capture features that are extremely small and detailed relative to the required coarse model. An example implementation of this is shown in Figure 9. You can experiment with submodeling and nested submodeling yourself by downloading the

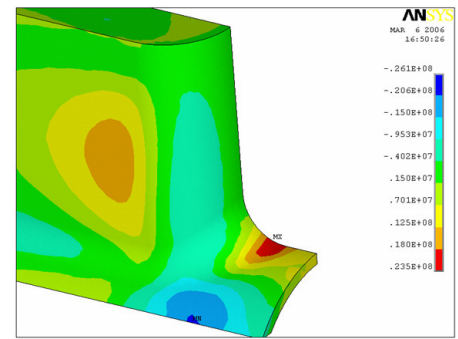


Figure 7: Submodel SZ

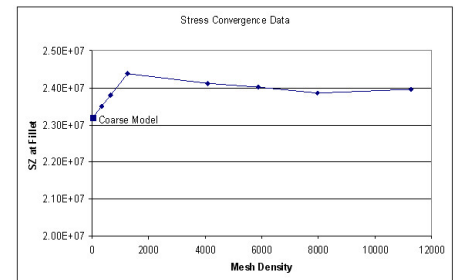


Figure 8: SZ vs. Mesh Density for Coarse and Submodel

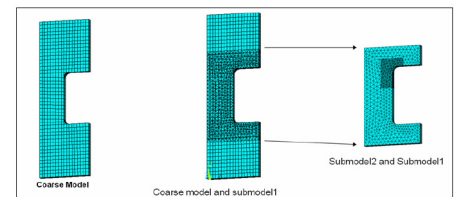


Figure 9: Nested Submodel Example

MOR for ANSYS: Model Order Reduction for ANSYS

By: Evgenii B. Rudnyi, Jan G. Korvink

Lab of Simulation, Department of Microsystem Engineering, University of Freiburg - IMTEK,

Model reduction is not something new for engineers. The mode superposition method and Guyan reduction methods have been implemented in ANSYS almost since the beginning. Yet, a topic of model reduction has recently received much attention from mathematicians and nowadays this makes a large difference. We suggest you a test: compare two recent books on model reduction, one written by an engineer [1] and one by a mathematician [2]. You will see that now mathematicians have much deeper understanding on why model reduction is possible and how to perform it in the most efficient way.

MOR for ANSYS is open-source software developed at IMTEK to research the possibilities of modern model reduction for engineering design and system level simulation. It reads binary ANSYS FULL and EMAT files in order to extract the system matrices and then runs the block Arnoldi algorithm to build a reduced model.

We have started the research on modern model reduction in 2001 and in our subsequent experience modern model reduction has proven to hold great potential for engineering. The key observations are listed below:

- The Arnoldi algorithm is the most efficient computational method to perform model reduction. Time to perform model reduction is comparable with that for a static solution provided there is a plenty of memory (with 4 Gb of RAM this observation is valid for many finite element models up to 500 000 degrees of freedom).
- Because of a), MOR for ANSYS allows us to obtain harmonic or transient simulation for a linear model for the computational cost comparable with a static solution!
- MOR for ANSYS can be used for any linear ANSYS model: thermal, mechanical, piezoelectric, acoustic including structure-fluid interaction, electromagnetic, even thermomechanical with non-uniform temperature.

Research papers describing results obtained with MOR for ANSYS are available from the MOR for ANSYS site.

Finally, we would like to direct your attention to new results on parametric model reduction. It happens that one can formally preserve several parameters within system matrices in the symbolic form during a modified model reduction procedure. This feature has not been implemented in MOR for ANSYS yet but the research phase has almost been completed and an update will follow.

More information on MOR for ANSYS

MOR for ANSYS site:

www.imtek.uni-freiburg.de/simulation/mor4ansys/

MOR for ANSYS tutorial:

www.imtek.uni-freiburg.de/simulation/mor4ansys/lab/thruster_rudnyi02SU.pdf

Introduction to modern model reduction at engineering level:

E. B. Rudnyi, J. G. Korvink, Review: Automatic Model Reduction for Transient Simulation of MEMS-based Devices. Sensors Update v. 11, p. 3-33, 2002.

dx.doi.org/10.1002/seup.200211105

or preprint at

www.imtek.uni-freiburg.de/simulation/mor4ansys/pdf/rudnyi02SU.pdf

Recent results on parametric model reduction:

www.imtek.uni-freiburg.de/simulation/mor4ansys/publications.html#parametric

An Engineer's Travelogue:

Interacting with the Natives and Understanding What they Worship



By: Egbertius Fog,
MBE, PE

I have just returned from my outfitter and all

is prepared for my pending journey to Pittsburgh for my favorite biennial event: the 2006 International ANSYS Conference. In my last dispatch to you, my faithful readers, I shared the anticipation of the sojourn to this congress of calculators. And, as promised, I would now like to prepare you further by sharing some insight into the natives and their tribes.

The first thing to be prepared for is how they worship. They will begin their meeting by taking part in ritual training sessions. Gathering in large rooms, the wisest and most experienced amongst them share their knowledge with others. In many ways this is the most powerful event at these gatherings. With the lights dimmed the attendees ingest large quantities of caffeinated beverages and stare at images projected on a large white rectangle for three to four hours. The words and paintings are indecipherable to most, but to the initiate they are not just interesting, but so fascinating that they often illicit cheers and clapping. Interviews of those who participate often result in comments similar to "that was the most useful class I have ever taken."

After the first two day of ritual training, the entire congress moves towards the main convention. Gathering in a single room they listen to stories from people both inside and outside the tribe about how the tools they all worship can make life better. Some speakers are leaders within the group, some are leaders from other groups. But all share a passion for engineering and creating products and the stories relate to how the world of engineering is changing and how members can be better at it and can contribute more to the business tribes for whom they work.

The other ritual that you should be aware of is a special village in the midst of the facility for another tribe referred to as vendors. As with many societies, trade between different groups is key to the success of all. This agora is a place for the users to meet vendors who have products that they can use to improve the quality, speed and efficiency of ANSYS usage. This year is especially exciting because of the strong focus on the growing need for high performance computing.

There is much more for you to know, but I must be off to my doctor for my vaccinations. I hope that this little taste is enough to convince you to join me at the conference.

"Sub Structuring" References

- [1] My high school Latin classes.
- [2] The American Heritage® Dictionary of the English Language, Fourth Edition copyright ©2000 by Houghton Mifflin Company.
- [3] The ANSYS 10.0 Help System, Chapter 9 of the Advanced Analysis Techniques Guide

"MOR for ANSYS" References:

- [1] Zu-Qing Qu, Model Order Reduction Techniques : with Applications in Finite Element Analysis. Springer, 2004, ISBN: 1852338075.
- [2] Athanasios C. Antoulas, Approximation of Large-Scale Dynamical Systems. Society for Industrial and Applied Mathematic, 2005, ISBN: 0898715296.



Upcoming Training Classes

Month	Start	End	#	Title	Location
Apr '06	6-Apr	7-Apr	107	ANSYS WB DsgnMdlr	Tempe, AZ
	10-Apr	12-Apr	401	Introduction to EMAG	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
May '06	1-May	3-May	104	ANSYS WB Sim Intro	Tempe, AZ
	4-May	4-May	105	ANSYS WB Sim Struct Nonlin	Tempe, AZ
	8-May	9-May	100	Engineering with FE Analysis	Tempe, AZ
	11-May	12-May	803	Tcl/Tk for ANSYS	Tempe, AZ
	15-May	16-May	604	Introduction to CFX	Irvine, CA
	18-May	19-May	203	Dynamics	Irvine, CA
	25-May	25-May	106	ANSYS WB DesignXplorer	Tempe, AZ
	31-May	31-May	108	Paramesh	Tempe, AZ



Links

Just Found out MIL-HDBK-5 is gone!?!?
Visit its Replacement at www.mmpds.org

Computer Benchmark Data for ANSYS?

www.padtinc.com/support/benchmark

Want to Work for ANSYS, Inc.?

ansys.recruitmax.com/candidates



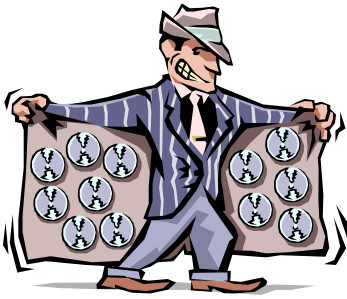
News

- [ANSYS Stock Price Breaks \\$50/Share](#)

- [International ANSYS Conference Gaining Steam](#)

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>

XANSYS
A Mailing List for the
ANSYS User Community



The Shameless Advertising Page

FOR *Better Toast*.. USE A **UNIVERSAL**

BETTER TOAST! That's what the New UNIVERSAL Automatic Toaster promises...and delivers. Every slice perfect. Just the way you want it, too...light brown, dark brown or medium. Moreover, the toast **CANNOT BURN**, because this Toaster is equipped with a New Selective Toast-Minder which removes the toast the moment it is ready.

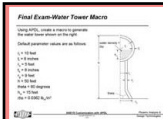
TOASTS 2 SLICES AT A TIME — BOTH SIDES AT ONCE — *quicker!*

2006

ANSYS CPD Windows XP/2000 Microsoft Office 2000/2003

*Be Cool and
Know what
Day it Is!*

Download the 2006 PADT
ANSYS Calendar



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**
- **Kyrl 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

© 2006 ANSYS, Inc. All Rights Reserved

Sponsors

ANSYS®

Platinum

Windows
Compute Cluster Server 2003

Gold

Autodesk

hp
invent

sgi

Silver

AMD

ATI
FIREGL

DELL

i*hydra
Quiet cool... for performance

intel

moldflow