

The Focus



A Publication for ANSYS Users

Contents

Feature Articles

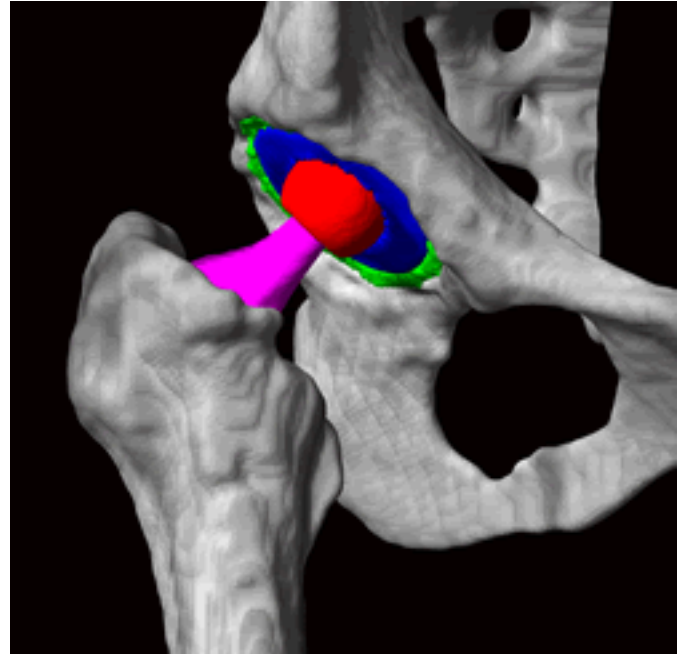
- [Simpleware: From 3D Image to Mesh](#)
- [Soldering Process](#)
- [The Ill-Conditioned Matrix](#)

On the Web

- [ANSYS Tips Archive](#)
- [Job Opportunities at PADT](#)

Resources

- [PADT Support: How can we help?](#)
- [Upcoming Training at PADT](#)
- [About *The Focus*](#)
 - *The Focus* Library
 - Contributor Information
 - Subscribe / Unsubscribe
 - Legal Disclaimer



ANSYS[®]
Support Distributor

The Focus



A Publication for ANSYS Users

Simpleware: From 3D Image to Mesh

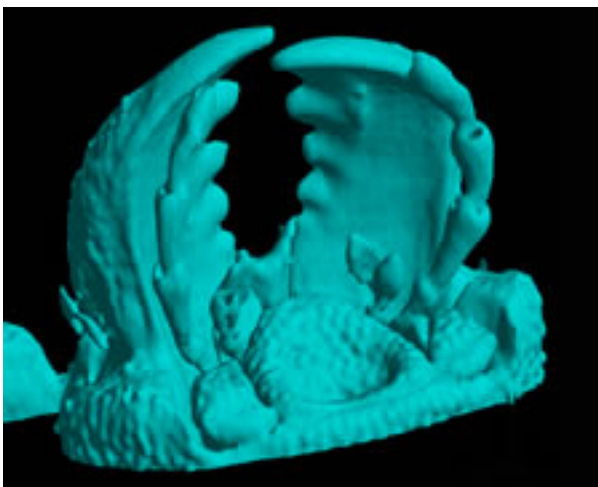
by [Emma Johnson](#), PR Officer, Simpleware Ltd.

In the past, converting 3D images into meshes for use in FE analysis often necessitated time consuming processes and a gross simplification of the model geometry. UK-based, imaging specialist [Simpleware Ltd.](#) currently offers two new software products, ScanIP and ScanFE, as an advanced solution to this problem, enabling users to quickly and accurately convert any 3D dataset, such as an MRI (magnetic resonance imaging) scan, into high-quality meshes in minutes.

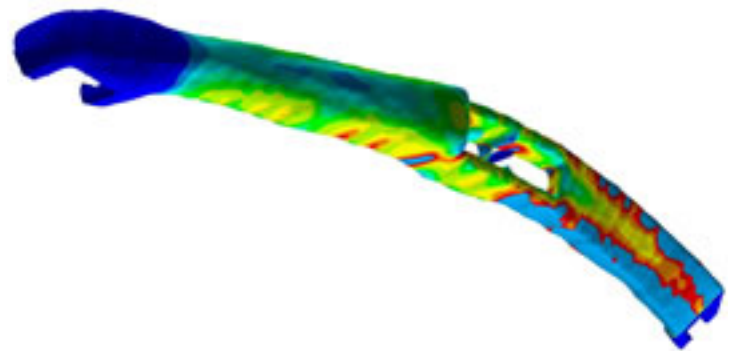


Simpleware Ltd. has established itself as the world leader in the provision of software and services for the conversion of 3D imaging data into numerical models. ScanIP provides image visualisation and processing and allows files to be exported to CAD and rapid prototyping software programs. ScanFE is a mesh-generation module that creates volumetric meshes which can be exported directly to ANSYS and other leading FE (and CFD) commercial solvers.

Simpleware's flagship software product ScanIP provides an extensive range of image processing and meshing tools to generate highly accurate models based on data from 3D imaging modalities such as MRI, Ultrasound and Computed Tomography (CT). Features of particular interest include: a metal artifact removal filter (for artifacts in CT scans); improved topology and volume-preserving smoothing algorithms; and a broad range of visualisation modalities.



Beetle (in ScanFE)



Windshield Wiper Blade (in ANSYS)

Novel proprietary algorithms and techniques developed by Simpleware, also permit fully automated, robust generation of FE meshes based on 3D imaging data. Mesh generation based on imaging data is an area of great interest in the FE community but the majority of approaches to date have involved



A Publication for ANSYS Users

generating a surface model from the scan data which is then exported to a commercial mesher – a process which is time consuming, not very robust, and virtually intractable for the complex topologies typical of image data. A more direct approach is to combine the geometric detection and mesh creation stages in one process. The process involves identifying volumes of interest (segmentation) and then meshing based on an orthotropic grid intersected by interfaces defining the boundaries. In effect, a base Cartesian mesh of the whole volume defined by the sampling rate is tetrahedralised at boundary interfaces based on cutting planes defined by interpolation points. The process incorporates an adaptive meshing scheme, which is fully automated and robust, creating smooth meshes with low element distortions regardless of the complexity of the segmented data.

A sophisticated assignment of material properties based on signal strength allows a general mapping function between greyscale and density or Young's Modulus to be defined (several different functions can be assigned to each part). These features are all in addition to the proprietary technology which ensures high quality multi-part meshes which conform perfectly at part interfaces (for both STL and volume meshes).

In addition to simplifying the meshing process dramatically, the mesh generation from scan data has several important advantages:

- a. Accuracy of meshed topology/morphology is only contingent on image quality. The geometry of the structure is reproduced in the finite element mesh at sub-voxel accuracy.
- b. Structures consisting of several different materials can be meshed automatically.
- c. Interfacial contacts can be modelled.

As well as performing convergence studies of field parameters of interest by increasing mesh density, convergence of models to morphology with increased image resolutions can be carried out. Where the properties vary in a continuous fashion throughout the structure, the approach can be used to derive a relationship or mapping function between the signal strength and the material properties which can be extremely useful for studying a wide range of problems including open celled foams, soil samples, and bone.

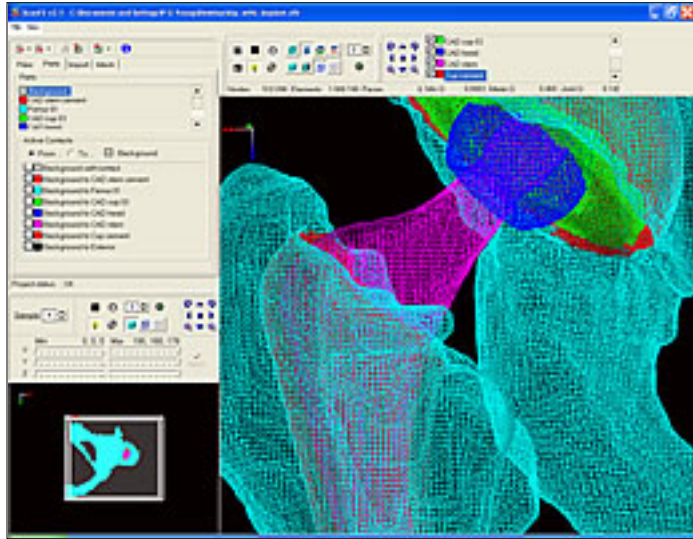
Case Study

A case study was carried out to explore the feasibility of using clinical data for post-clinical structural evaluation of implant performance. An *in vivo* clinical scan of a patient fitted with a total hip replacement (THR) system was used to explore the influence of mesh density on the predicted response as well as the influence of the assumed contact model at the cup–implant interface.

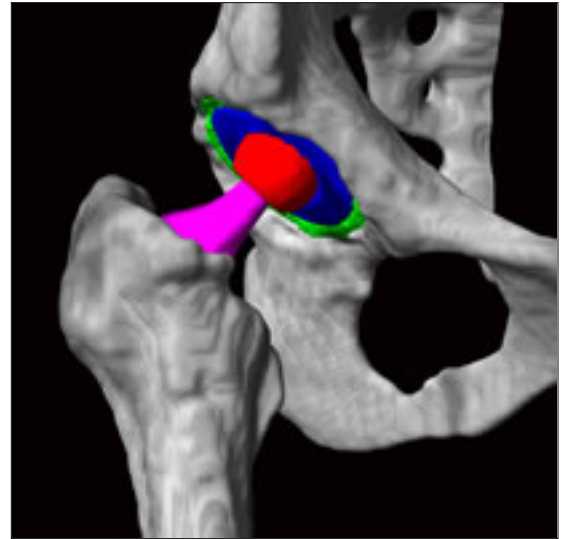
The Focus



A Publication for ANSYS Users



ScanFE: Hip, Femur, and Implant



ScanFE: Rendered at High Resolution

Methodology

A CT scan of in-plane resolution 0.77 mm and slice-to-slice separation 1 mm was re-sampled and a Metal Artifact Removal (MAR) filter applied. Six masks were created using ScanIP: (1) Pelvis, (2) Cement, (3) Cup, (4) Stem, (5) Cement mantle, (6) Proximal Femur. Based on the six segmented structures, two smooth models of different mesh densities were generated using ScanFE taking less than 3 minutes each. Additionally, a rapid prototyped model replica with the exact geometry as the FE mesh topology was generated. Using ANSYS, material properties, boundary conditions and loads – including muscle forces – were applied. Nodes at the top of the pelvis and distal part of femur were defined in ScanFE. The response of the system was analysed under static loading conditions with a sliding interface at cup-implant interface. The total solution time (on an Intel 2.8 GHz) for the low density mesh was a little over 2 hours, and 6 hours for the high density model.

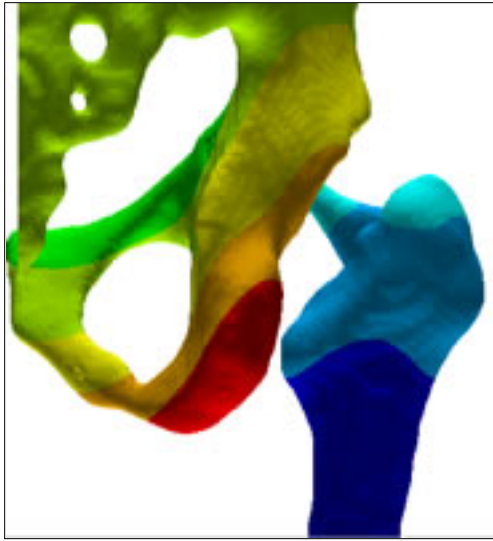
Results

The study demonstrated the potential of the proposed approach for the generation of patient specific FE models based on in vivo clinical scans. In spite of their complexity and sophistication, full FE simulations can be carried out on inexpensive and commonly available hardware platforms.

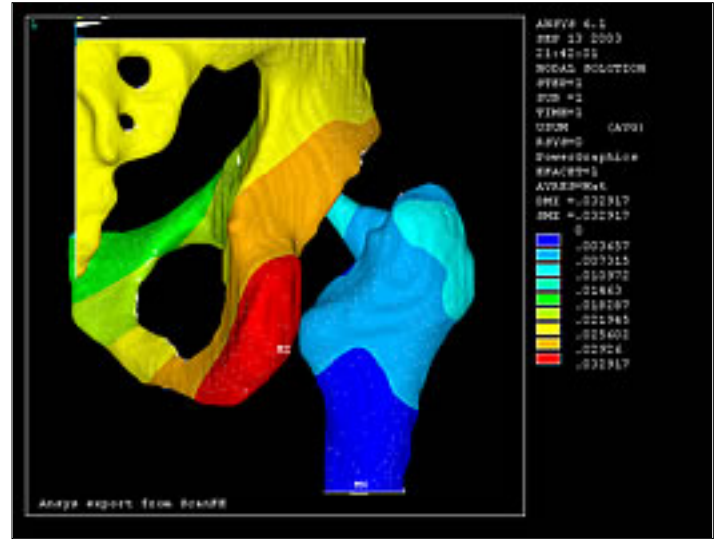
The Focus



A Publication for ANSYS Users



ANSYS: Hip Analysis

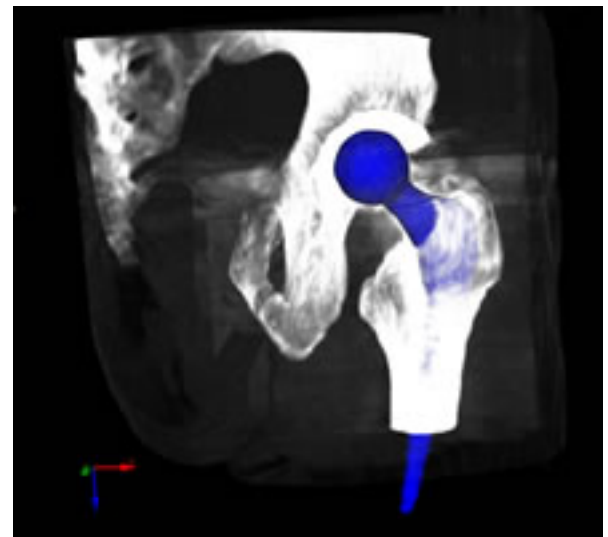


ANSYS: Hip Analysis Results

The ease and accuracy with which models can be generated have opened up a wide range of previously difficult or intractable problems to numerical analysis, including blood flow, material characterisation of nano-structural composites and patient specific implant design. If the system is coupled with rapid prototyping hardware, it is also possible to produce a solid polymer or metal facsimile of the object in question – this part of the process can then be effectively conceived as a 3D photocopier.

Simpleware has developed a new module, ScanCAD, which allows the import and interactive positioning of CAD models within the image masks. This can be used to bring in reaming tools, implants etc. and integrate them into the image. STL or FE models can then straightforwardly be generated. The new release will also include level set methods which are very powerful techniques for segmenting images.

For more information, please send questions to the author at info@simpleware.com.





A Publication for ANSYS Users

Soldering Process

by Rod Scholl, PADT

There's nothing too whiz-bang about this article. But the question comes up enough, it might make a good reference for others (and me) in the future.

When modeling a soldering process (or welding in some cases), it can be confusing about how to arrange one's load steps to allow for the cooling, shrinkage, and residual strains that can result. Like most things in ANSYS, the trick is just to model the actual physics in a quasi-static step-by-step approach; meaning, there is no trick.

What you *will* need:

- Solder material model with very low modulus at high temperature (i.e., liquid) and typical lead/tin modulus at low temperature

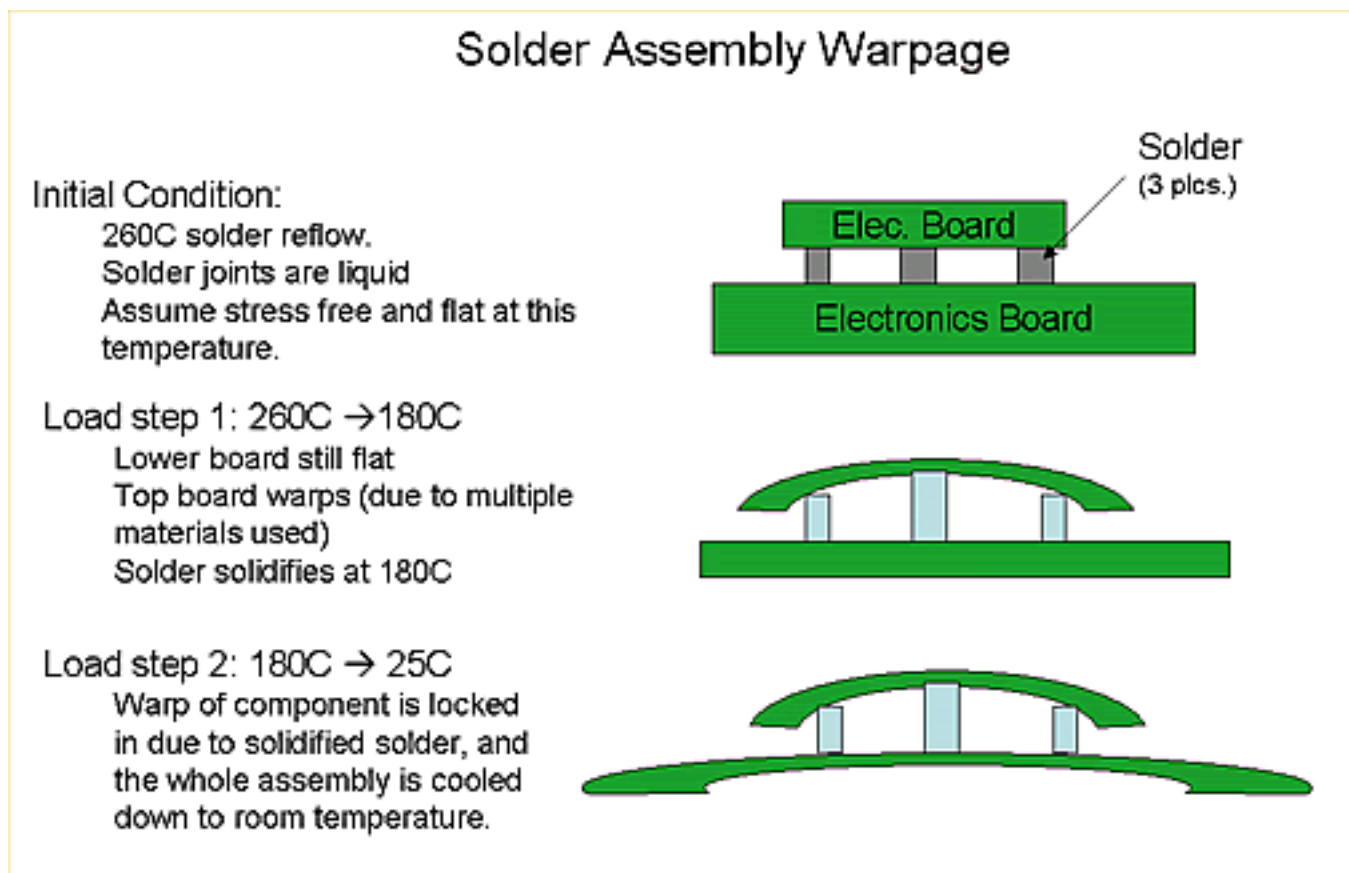
What you *won't* necessarily need:

- Update deformed mesh (UPCOORD)
- Element Birth/Death
- Plastic Material Model (e.g. KINH)
- Red spandex jogging suit
- A box of peanuts
- Twelve angry spider monkeys
- A shot of bourbon (but perhaps necessary if you include the above three)

A good example was created by [Wei Lin](#) – thankfully capturing a typical board soldering process.

Process to Capture

- Specify the temperatures/loads as they occur in the actual process.
- Specify the material properties of solder such that a phase change is approximated via a drastic modulus change at the melting point.
- Without leaving /SOLUTION (or using restarts) solve the model with multiple load steps.
- The residual stresses/strains take care of themselves.



The Focus



A Publication for ANSYS Users

Script to Capture It

```

!Soldering Process
!Rod Scholl / PADT

fini
/clear
/prep7

!*** Make Geometry ***
length=10
blc4,0,0,length,.1
blc4,0,.2,length,.05
blc4,0,.25,length,.05
blc4,.45,.1,.1,.1
blc4,.5*length,.1,.1,.1
blc4,length-.45,.1,.1,.1
aglu,all
et,1,42
asel,s,loc,y,.1,.2
esize,.01
amesh,all
esize,.03
asel,inve
amesh,all
asel,s,loc,y,0,.1
esla
emodif,all,mat,2
asel,s,loc,y,.2,.25
esla
emodif,all,mat,3
asel,s,loc,y,.25,.3
esla
emodif,all,mat,4
/pnum,mat,on
/number,1
alls

!*** Mat. Props. ***
MPTEMP,,,,,,,,
MPTEMP,1,25
MPTEMP,2,170
MPTEMP,3,180
MPTEMP,4,260
MPDATA,EX,1,,30e3
MPDATA,EX,1,,30e3
MPDATA,EX,1,,1
MPDATA,EX,1,,1
MPDATA,prxy,1,,.3
MPDATA,prxy,1,,.3
MPDATA,prxy,1,,.3
MPDATA,prxy,1,,.3
MPDATA,prxy,1,,.3
TB,kinh,1,4,2,0
TBTEMP,25
TBpt,,1,30000
TBpt,,2,60000
TBTEMP,170
TBpt,,1,30000
TBpt,,2,60000
TBTEMP,180
TBpt,,1e-3,1e-3
TBpt,,2,1
TBTEMP,260
TBpt,,1e-3,1e-3
TBpt,,2,1

mp,alpx,1,20e-6,
mp,ex,2,1e6
mp,ex,3,1e6
mp,ex,4,1e6
mp,alpx,2,1e-7
mp,alpx,3,3e-7
mp,alpx,4,1e-7

! ***BC's and Solve ***
/solu
nlgeom,on
d,node(0,0,0),all,0
d,node(length,0,0),uy,0
allsel,all
antype,0 ! static analysis

! 1st load step: from 260C to
180C solder solidification
tref,260
tunif,180
deltim,.25
solve

! 2nd load step: from 180C to
25C room temperature
tunif,25
solve

!*** View Results ***
finish
/post1
/dscale,1,50
plns,u,sum

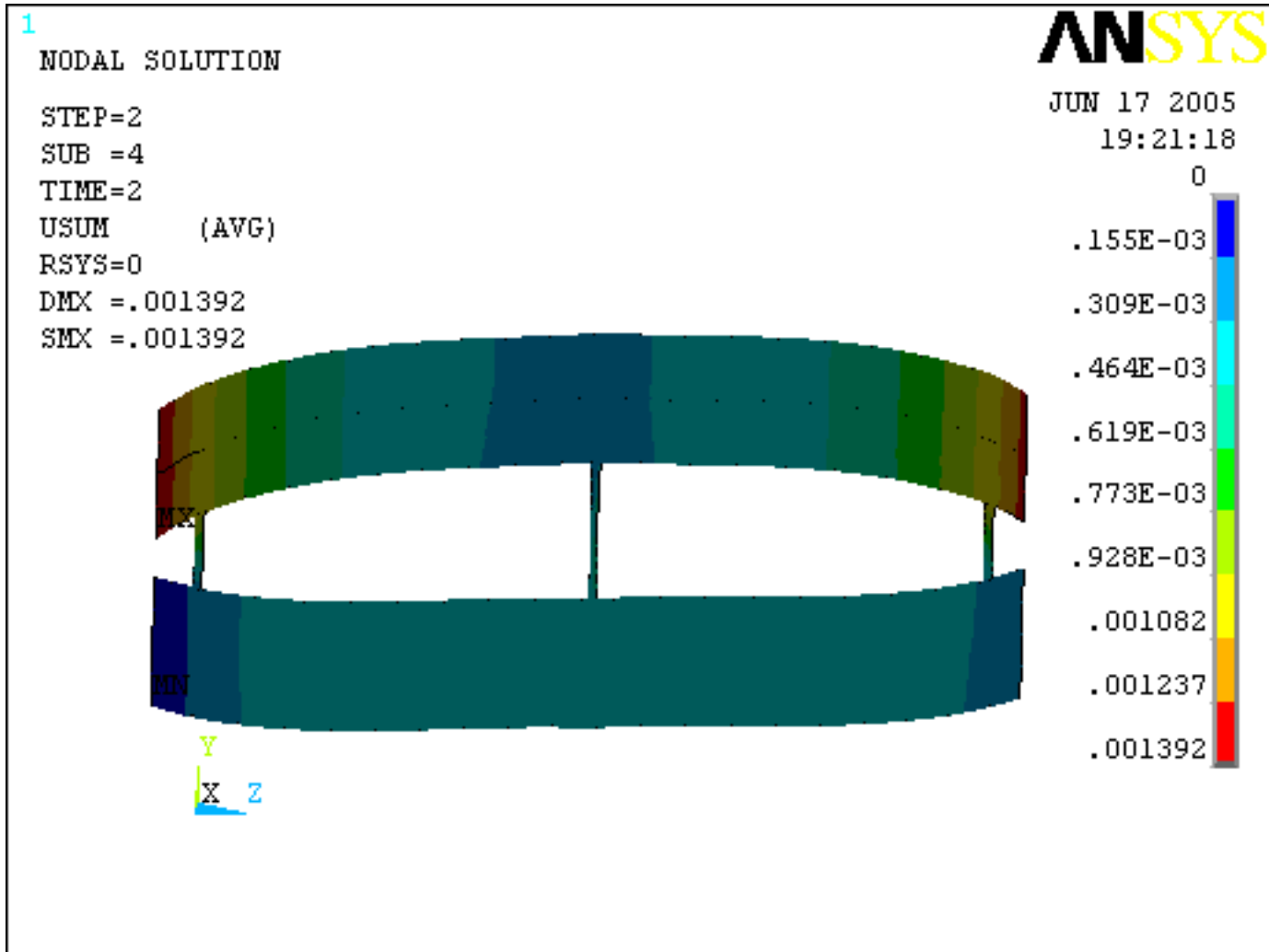
```


The Focus



A Publication for ANSYS Users

Results



Thanks again to Wei Lin for pictures and capturing of a typical process.

The Focus



A Publication for ANSYS Users

“Ill-Conditioned Matrix”: Professor Jargon's Term of the Month

by Professor F. E. Jargon, Jargon Technical School

PADT is pleased to present F. E. Jargon, Professor Ebrius from the Jargon Technical School (JTS). Professor Jargon has kindly volunteered his services as a guest columnist for *The Focus*. Over the next several months, Prof. Jargon will be introducing common finite element terms and explaining their meaning and relevance to the finite element engineer.



Professor F. E. Jargon

Greetings, students! I am honored to have the opportunity to write for such a prestigious periodical. It pleases me to take a break from my jargon teachings to share my knowledge with you.

One term that may puzzle you is the “ill-conditioned matrix.” You sometimes get a warning when using the iterative solvers stating that the stiffness matrix may be ill-conditioned with a suggestion to use the sparse solver. This may cause you to think, “Wow, they seem to use that term so nonchalantly, as though I should have learned it in first grade. Did I have chicken pox that week?”

You may not have heard it in first grade, but the term may have reared its head in your linear algebra class. Perhaps you were too eager to get to the “real” engineering courses to have remembered. A matrix is ill-conditioned if small changes in the coefficients of the solution have drastic effects on the results, which makes iterating the solution to a small residual a tricky operation. Numerical round-off in the system can also present some challenges when it comes to solving a model having an ill-conditioned matrix.

An ill-conditioned matrix can represent the equations of two lines that are almost parallel, but not quite. Consider the following system of equations, written in the form of $[K]\{x\}=\{F\}$:

$$\begin{vmatrix} 1 & 2 \\ 2 & 4.0001 \end{vmatrix} \begin{Bmatrix} x_1 \\ x_2 \end{Bmatrix} = \begin{Bmatrix} 3 \\ 6.0001 \end{Bmatrix}$$

The answers are $x_1 = x_2 = 1$. But, round off 6.0001 to 6 and we get $x_1 = 3$ and $x_2 = 0$. Leave the 6.0001 number alone and round off 4.0001 to 4 and we get two conflicting equations. Round off both 4.0001 and 6.0001 to 4 and 6 and we end up with two redundant equations. Just tampering with the values slightly has resulted in radically different solutions!

Another type of ill-conditioned matrix — and one more relevant to the finite element analyst — is when we have $[K]$ matrix values that vary by several degrees of magnitude. Like the “near parallel” example,



A Publication for ANSYS Users

this can also result in widely discrepant results due to small perturbations in the load values. This scenario is caused by radical differences in stiffnesses (or conductivities, etc.) within the model. This sometimes happens in shell models where the tensile stiffnesses are much higher than the rotational stiffnesses. It can also occur in a model having two materials whose Young's moduli differ by several orders of magnitude. I've also seen this in 3D models that have a large amount of bending flexibility compared to the overall tensile stiffnesses. Short beam elements having a much higher stiffness than other elements in the model can also cause ill-conditioning. Poor meshes having elements with large aspect ratios or Jacobians can contribute to this phenomenon as well.

The ill-conditioned matrix is particularly troublesome when trying to use the iterative solvers. Because ANSYS is guessing at an initial solution and iterating to a certain tolerance value, the divergent nature of the ill-conditioned solution makes the iterative process inefficient compared to using a direct solver. However, a slow solution is only a minor inconvenience, comparatively speaking. Because of the potential for discrepant results, it is quite possible to come up with results that are just plain wrong, regardless of the solver used. For now, the only way to avoid this is to be aware of the pitfalls and avoid them. The skilled ANSYS user will take extra steps to achieve a robust mesh and verify the accuracy of his or her results.

Ever run across a word in the ANSYS world that you were unsure about but were afraid to ask about? [Send us an e-mail](#) and we will forward it to Professor Jargon. All submissions will be kept strictly anonymous.

References

Gene Poole, Technical Fellow, ANSYS Inc.

Chapra, Steve C. and Raymond P. Canale. *Numerical Methods for Engineers*. 2nd Ed. New York: McGraw Hill, 1988.

The Focus



A Publication for ANSYS Users

Upcoming Training at PADT

A complete schedule of the training courses for the next three months is shown below. [Learn more](#) about how the **Training Services** offered by PADT can save you time and money. Or, feel free to drop an e-mail to our training coordinator, [Ted Harris!](#)

Month	Start	End	Course	Description	Location
Jun '05	6/27	6/28	301	Heat Transfer	Tempe, AZ
	6/29	6/30	107	ANSYS Workbench DesignModeler	Irvine, CA
Jul '05	7/11	7/13	104	ANSYS Workbench Simulation - Intro	Albuquerque, NM
	7/14	7/15	801	ANSYS Customization with APDL	Albuquerque, NM
	7/25	7/26	203	Dynamics	Albuquerque, NM
	7/27	7/27	108	Paramesh	Albuquerque, NM
Aug '05	8/1	8/3	104	ANSYS Workbench Simulation - Intro	Tempe, AZ
	8/4	8/5	100	Engineering with Finite Element Analysis	Tempe, AZ
	8/8	8/10	152	AI*Environment	Tempe, AZ
	8/11	8/12	202	Advanced Structural Nonlinearities	Tempe, AZ
	8/15	8/16	501	ANSYS/LS-DYNA	Tempe, AZ
	8/18	8/19	604	Introduction to CFX	Tempe, AZ
	8/24	8/26	402	High Frequency Electromagnetics	Tempe, AZ
Sep '05	9/1	9/2	301	Heat Transfer	Albuquerque, NM
	9/9	9/9	108	Paramesh	Tempe, AZ
	9/12	9/14	101	Introduction to ANSYS, Part I	Tempe, AZ
	9/19	9/22	802	Advanced APDL and Customization	Tempe, AZ

Whether it s one of our regularly-scheduled classes at our training facility, or a customized class tailored to your specifications and location, you can be assured that the training you receive will have immediate, positive results on your capabilities in design, analysis, and product and process improvement.





A Publication for ANSYS Users

About *The Focus*

The Focus is a periodic electronic publication published by PADT, aimed at the general ANSYS user. The goal of the feature articles is to inform users of the capabilities ANSYS offers and to provide useful tips and hints on using these products more effectively. *The Focus* may be freely redistributed in its entirety. For administrative questions, please contact [Rod Scholl](#) at PADT.

The Focus Library

All past issues of *The Focus* are maintained in an online [library](#), which can be searched in a variety of different ways.

Contributor Information

Please don't hesitate to send in a contribution! Articles and information helpful to ANSYS users are very much welcomed and appreciated. We encourage you to send your contributions via e-mail to [Rod Scholl](#).

Subscribe / Unsubscribe

To subscribe to or unsubscribe from *The Focus*, please visit the PADT e-Publication [subscriptions](#) management page.

Legal Disclaimer

Phoenix Analysis and Design Technologies (PADT) makes no representations about the suitability of the information contained in these documents and related graphics for any purpose. All such document and related graphics are provided as is without warranty of any kind and are subject to change without notice. The entire risk arising out of their use remains with the recipient. In no event, including inaccurate information, shall PADT be liable for any direct, consequential, incidental, special, punitive or other damages whatsoever (including without limitation, damages for loss of business information), even if PADT has been advised of the possibility of such damages.

The views expressed in *The Focus* are solely those of PADT and are not necessarily those of ANSYS, Inc.