

## Inside this issue:

CAEA News & Events	2
Cool Free ANSYS Macros	2
ANSYS "Did You Know"	2
ANSYS News	3
Manager's Corner	3

## Spotlight on Consulting:

### Customized Training and Mentoring

The goal of many of CAEA's consulting projects is to provide a solution for a particular problem and to offer recommendations for improvement of a process or a design. This has been the traditional role of engineering consulting, and we have been doing it for over 20 years.

However, a less familiar form of consulting that CAEA performs is helping our clients to better utilize their software tools in order to improve their internal analysis procedures. Many organizations do not make optimal use of the simulation tools at their disposal, and turn to CAEA for help in maximizing their benefits in two complementary ways:

- Customized Training
- Mentoring Services

Customized training focuses on the types of analysis and applications specific to the client. Both lecture materials and workshop problems are developed to directly address the topics, industry-specific best practices, and custom applications relevant to the customer's needs.

Most of our larger **ANSYS** customers insist on this type of training, because students can immediately start solving practical problems important to their company. The same advantages can apply to organizations of all sizes. CAEA is a world-leader in **ANSYS** course development and training, having developed some of the general-

purpose ANSYS training notes and conducting over 1500 hours of training per year.



While training is necessary in effective use of the software, it is not a substitute for experience. Today's simulation problems can be very challenging, and engineers are sometimes faced with problems outside their experience range. Many times they ask us to share our experience and help them get started.

We work one-on-one or with a group of engineers to demonstrate state of the art procedures to most effectively accomplish their tasks. From the client's perspective, mentoring is often the most cost-efficient and fastest way to reach their objectives. Their return on investment is almost instantly realized.

The key to success of customized training or mentoring is the experience of the CAEA staff, both from an analysis and training standpoint. Our engineers come from diversified backgrounds, all with advanced degrees, many years of experience, and with the communication skills required to convey this information to others.

Customized training and mentoring can be performed on-site at your location, or at our training facility in Woodbury, CT. To inquire about these services, please contact us at [caea@caei.com](mailto:caea@caei.com).

## CAEA to Work For NIST on WTC Disaster Investigation

CAEA will conduct thermal/structural analyses of the World Trade Center Towers and Building 7 under two recently-awarded NIST contracts:

Solicitation number SB1341-03-R-0044 was awarded to Simpson Gumpertz & Heger Inc. (SGH) of Waltham, Massachusetts, partnering with Computer Aided Engineering Associates Inc. The objective of the investigation is to determine the response of structural components and systems to the fire environment with and without damage in the World Trade Center Towers and to identify probable structural collapse mechanisms. For more

information, please go to: [http://wtc.nist.gov/solicitations/wtc\\_awardR0044.htm](http://wtc.nist.gov/solicitations/wtc_awardR0044.htm).

Solicitation number SB1341-03-R-0028 was awarded to Gilsanz Murray Steficek LLP (GMS) of New York City and its team composed of Dr. John Fisher of Lehigh University, Pennsylvania, and Computer Aided Engineering Associates Inc. The GMS team will conduct analyses of the WTC 7 building's structural response to fire conditions. For more information, please go to: [http://wtc.nist.gov/solicitations/wtc\\_awardR0028.htm](http://wtc.nist.gov/solicitations/wtc_awardR0028.htm).

## CAEA News & Events: Upcoming Seminars

CAEA has had ongoing success with our special one day **ANSYS Workbench** seminar series in Connecticut, New Jersey and New York. As a result of high attendance, we will continue to run them. The **ANSYS Workbench** environment provides an intuitive user experience of unique automation, performance, and knowledge capture technology that integrates simulation activities with parametric CAD systems; all built upon the world-class **ANSYS** solver foundation. To view the CAEA presentation on **ANSYS Workbench**, go to [www.caeai.com](http://www.caeai.com) and click on **ANSYS Workbench Environment** in the News and Information area.

We will also add one day seminars in 2004 for both **ANSYS DesignModeler** and **DesignXplorer**. **DesignModeler (DM)** functions as an add-on to the **ANSYS Workbench Environment** and contains the tools necessary to modify an attached or imported model to suit the requirements of a desired simulation. **DesignXplorer** provides performance insight into product designs by integrating

simulation activities with parametric CAD models; producing intuitive performance results that lead to the natural selection of optimum design alternatives.

For more information, contact Rene Sprunger in CT at 203-483-8810 or Tony Solazzo in NY/NJ at 973-927-9970.

### ANSYS User Group: April 2004

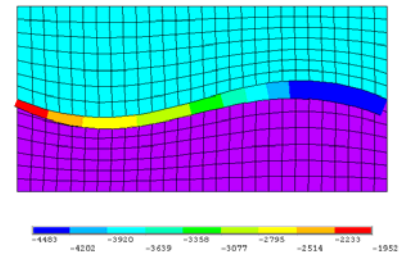
**ANSYS** users, we are currently looking for topics and speakers for our next CAEA **ANSYS** User Group meeting!

Contact Rene Sprunger or Tony Solazzo with suggestions or to volunteer as a speaker.

### ANSYS 8.0

**ANSYS** has recently released version 8.0. CAEA will be offering an update training seminar on 8.0 in February of 2004. Look for details soon at [www.caeai.com](http://www.caeai.com).

*More information on the release of ANSYS 8.0 on page 3.*



Stresses normal to the interface between materials, generated with the MAPPATH macro.

## Cool Free ANSYS Macros

It is often useful to plot and list stresses normal and tangent to component or material interfaces. These stresses can be used to predict the potential for failure in a bond layer, for example. In many cases, however, the interface is not defined by a straight line in a 2D model or by a flat surface in a 3D model, making the calculation of normal and tangential stresses difficult. We created a macro called **MAPPATH.MAC** to generate a path along a pre-existing line in the model, and to calculate stresses normal and tangent to the path, as well as resultant shear stresses along the path.

This macro requires three command line arguments: the number of the line defining the path, the node number at the starting point of the path, and the number of interpolation points on the path. The macro uses the **PVECT** command to obtain the direction cosines at each point on the path and proceeds with a series of **PDEF** and **PCALC** commands to transform stresses to the local coordinate system defined by the path orientation. The attached figure shows stresses normal to the path defined by the line between two materials. This macro works with both 2D and 3D models and can easily be modified to follow any predefined path.

*Download this and other useful ANSYS macros from our website:*  
[www.caeai.com/dlmacros.htm](http://www.caeai.com/dlmacros.htm)

## ANSYS “Did You Know”: DLL Relocation Errors in Windows

A problem has recently appeared for some **ANSYS** users running on PCs with the Windows 2000 operating system. Certain recent Microsoft service packs, if installed, will prevent **ANSYS** from executing, giving a “DLL relocation errors” message. Depending on the **ANSYS** version being run on the computer, the problem can be resolved in several ways.

The “DLL relocation errors” message will appear if Windows 2000 Service Pack 4 is installed, or if certain security service packs for Service Pack 3 are installed. If you see this error, and you have a maintenance agreement in place with ANSYS, the resolution is to download and install the following **ANSYS** Service Pack based on the version of **ANSYS**:

1. **ANSYS** 7.1 Service Pack 1.
2. **ANSYS** 7.0 Service Pack 11.
3. **ANSYS** 6.1 Service Pack 5.
4. **ANSYS** 6.0 Service Pack 3.

To download the service packs, go to [www.ansys.com](http://www.ansys.com), select **My ANSYS**, then select **Customer Portal**. Enter your username and password. Then select **Service Packs**, click on the **ANSYS** version, and follow the directions. A “read me” file can be viewed which describes the installation procedure.



## Contact Information



60 Middle Quarter Mall  
Woodbury, CT 0679

Phone: (203) 263-4606  
Fax: (203) 266-9049  
Company E-mail: [caea@caeai.com](mailto:caea@caeai.com)  
Newsletter E-mail: [editor@caeai.com](mailto:editor@caeai.com)

**President:**

Dr. Nicholas Veikos [veikos@caeai.com](mailto:veikos@caeai.com)

**Vice-President:**

Mr. Peter Barrett, P.E. [barrett@caeai.com](mailto:barrett@caeai.com)

**Sales Managers:**

Mr. Rene Sprunger (CT) [sprunger@caeai.com](mailto:sprunger@caeai.com)  
Mr. Tony Solazzo (NY,NJ) [solazzo@caeai.com](mailto:solazzo@caeai.com)

**Newsletter Editors:**

Dr. Michael Bak [editor@caeai.com](mailto:editor@caeai.com)  
Mr. James Kosloski  
Mrs. Debra Hankey

All brand and product names are registered trademarks of their respective companies.  
© Copyright Computer Aided Engineering Associates, Inc. 2003.

## CAEA Training Schedule for 2003-04

<u>Date</u>	<u>Course Title</u>
Dec. 1-2	<b>ANSYS</b> Random Vibrations
4	<b>ANSYS</b> Nonlinear One Day Seminar
8-9	<b>ANSYS</b> Dynamics
Feb. 2-4	Introduction to <b>ANSYS</b> Part I
5-6	Introduction to <b>ANSYS</b> Part II
10	Finite Element Analysis Short Course
11	Introduction to <b>ANSYS Workbench</b>
12	<b>ANSYS DesignModeler</b>
13	<b>ANSYS DesignXplorer</b>
24-25	Explicit Dynamics with <b>ANSYS/LS-DYNA</b>
Mar. 2-4	Finite Element Analysis Best Practices
5	<b>ANSYS</b> Capabilities Overview
15-17	<b>ANSYS</b> Structural Nonlinearities
18-19	<b>ANSYS</b> Advanced Structural Nonlinearities
23-25	<b>ANSYS</b> Heat Transfer

*All classes held in Woodbury, CT.*

*For a description of a listed course, visit:*

[www.caeai.com/stseminars.htm](http://www.caeai.com/stseminars.htm).

### CAEA Presents Free Nonlinear Analysis Seminars

**Dates & Locations:**

Thursday, December 4, 2003

At: CAE Associates  
60 Middle Quarter Mall  
Woodbury, CT 06798

Wednesday, December 10, 2003

At: Hewlett Packard  
1001 Frontier Road  
Bridgewater, NJ 08807

Space is limited, so sign up today by contacting Debra Hankey at [hankey@caeai.com](mailto:hankey@caeai.com) or by phone at (203)263-4606 or visit [www.caeai.com/nlseminar](http://www.caeai.com/nlseminar).

## I'm an ANSYS Expert, Why do I need Workbench?

Several **ANSYS** features are automated and more efficient in the **Workbench** environment such as:

1. Automatic contact pair generation during an assembly file import. When you attach an assembly in the **Workbench** environment, symmetric surface to surface contact pairs are generated automatically at all part to part interfaces. All contact pairs can be edited.
2. A "bolt load" in the **Workbench** environment automatically generates surface effect elements and a vector oriented pressure distribution on cylindrical surfaces.
3. A "pin support" in the **Workbench** environment automatically generates a nonlinear surface to surface contact pair with a rigid target. The contact pair only reacts on the projected surfaces opposite the net load direction.
4. Attached geometry is parametric and bi-directional. Boundary conditions are automatically remapped when the geometry is updated.
5. The **Workbench** finite element model (nodes, elements,

material properties, boundary conditions) can be loaded into the **ANSYS** environment. Nodal and element components can also be passed to the **ANSYS** environment.

**Workbench** is designed to interact with **DesignModeler**. **DesignModeler** is a feature-based solid modeling tool developed by **ANSYS** that generates or modifies solid model geometry for use in either the **ANSYS** or **Workbench** environments. **DesignModeler** works with single part files and assemblies. Geometric parameters from ProE, UG, SolidWorks, Solid Edge, Inventor, and Mechanical Desktop are bidirectional with **DesignModeler**.

Non-parametric SAT, Parasolid, and IGES files can also be read into **DesignModeler**. In **DesignModeler** features can be added and removed. Parts can be sliced to enable brick meshing. Surfaces can be extracted from the solid model for shell meshing. Parametric beam and shell geometry can be created in **DesignModeler** and used in both **ANSYS** and **Workbench**. **DesignModeler** geometry can be loaded directly into the **Workbench** environment or exported as a Parasolid or ANF file for use in **ANSYS**.