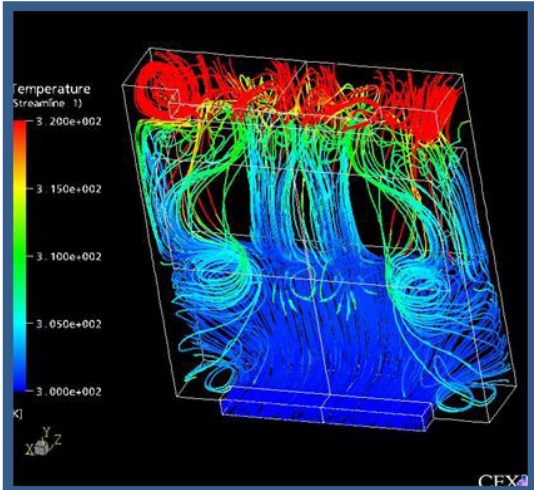


## Computational Fluid Dynamics... With ANSYS CFX



A view of ANSYS CFX, the computation fluid dynamics code used to model fluid flow, heat transfer, and related phenomena.

Computational fluid dynamics (CFD), is a modeling tool for simulating the behavior of systems, processes, and equipment involving fluid flow, heat transfer and related physical phenomena. The ANSYS CFX mission is to improve customers' efficiency and profitability by delivering outstanding CFD solutions and support.

CFX is of value to engineers tasked with developing or improving products and processes involving fluid flow, heat transfer and /or chemical reaction. CFX delivers the ability to apply the most powerful and precise CFD technology to virtually every fluid engineering problem.

These features allow CFX to set a new standard for high performance CFD by providing benefits which differentiate it from the competition:

- Accurate solutions using smaller models
- Robust solutions – no need to tweak the solver to get converged results
- Fast convergence
- Flexibility for customization

The CFX strategy moving forward is to increase its leadership in the ability to solve challenging industrial flow problems, while focusing on usability in order to make this technology accessible to more users.

A big part of this strategy is the integration of CFX-5 into ANSYS Workbench. At ANSYS release 8.1, the ANSYS meshing tools for CFX (ICEM CFD and CFX Mesh) are incorporated into Workbench. By the end of 2004, it is anticipated that CFX-5 will be fully integrated within the Workbench environment.

Geometry, meshing, physics, solver, and post-processing for CFD will all be accessible as Tabs within Workbench - the new CFD platform. This will set a new standard for an integrated and easy-to-use general CFD system.

Longer term, Workbench will become an unparalleled platform for future growth into a general Multiphysics system, with probabilistic and robust design tools available for all simulations.

We are thrilled to be able to offer our customers the power of a world-class CFD tool and think the combination of ANSYS and CFX will be unbeatable when it comes to serving our customers' simulation needs.

For more information, please contact Rene Sprunger or Tony Solazzo. A brief presentation on CFX can also be found on our website, [www.caeai.com](http://www.caeai.com), under CAEAI News.

Also, please contact Dr. Tsuei (p3) if CAEA can help you meet your CFD analysis requirements.

Ease-of-use, robustness and accuracy are ensured by CFX's combination of:

- A wide range of physical models that greatly expand the range of industrial fluid flow applications that can be quickly evaluated with CFD, for example: multiple frame of reference, turbulence, combustion and radiation, Eulerian two phase and free surface flow.
- In addition, CFX offers an open architecture that encourages customization on all levels. Both input and results are in accessible formats that allow easy customization enabling full integration within your existing engineering software environment.
- Advanced coupled multigrid linear solver technology
- Unmatched meshing flexibility
- Superb parallel efficiency
- Excellent pre- and post-processing capabilities

### Inside this issue:

ANSYS Did You Know	2
Cool Free ANSYS Macros	2
ANSYS Version 8.1 Released	2
CAEA Welcomes Dr. Tsuei	3
Introducing ANSYS AI* Environment 5.0	3
Manager's Corner - ParaMesh	3
CAEA Training Seminars	4
ANSYS DesignModeler	4

## ANSYS Did You Know? Bonded Contact Using MPC Contact

In the Fall 2002 CAEAI Newsletter, the procedure for attaching two mismatched meshes together using bonded contact was presented. Starting with ANSYS version 8.0, a more efficient procedure for enforcing compatibility between mismatched meshes is available: bonded contact using the MPC algorithm.

The MPC algorithm in ANSYS enforces compatibility at an interface using internally generated constraint equations. Highlights of the MPC method include:

- Degrees of freedom of the contact nodes are eliminated.
- No normal or tangential stiffness is required.
- No iteration is needed for small deformation problems, i.e. represents "true linear contact" behavior.

- For large deformation problems, the MPC equations are updated at each iteration.
- Method can be used to bond solid-to-solid, shell-to-shell, shell-to-solid, and beam-to-solid/shell.
- Improved solution efficiency over traditional bonded contact.

The procedure to create a solid-to-solid bonded contact interface has the following steps:

1. Create surface-to-surface contact pairs on the surfaces.
2. Set contact behavior to bonded: KEYOPT(12) = 5 or 6.
3. Switch contact detection to nodes: KEYOPT(4) = 1 or 2.
4. Set contact algorithm to MPC: KEYOPT(2) = 2.



*Example of MPC Bonded Contact to Connect Different Regions in an Assembly.*

## Cool Free ANSYS Macros: Automating Submodeling

Submodeling is a technique to obtain accurate stresses in a local region of a model by creating a new, independent model of the local region. To ensure that the submodel behaves in the same way as the global model, the boundary conditions on the cut boundary of the local model are obtained from the displacement field from the global model.

The procedure to perform submodeling is partially automated by the use of the **CBDOF** command that interpolates the global model displacements onto the submodel boundaries. However, care must be taken when switching between global and local models, and in applying boundary conditions accurately.

A macro that streamlines the submodeling procedure has been developed. The macro is called from the submodel and requires four inputs:

- The name of the global model filename.
- The load step and substep from the global results file that will be used to define the loading condition to be used in the submodel.
- The name of the component containing the nodes on the submodel boundary.

The macro will automatically perform the cut boundary procedure and apply the boundary conditions on the submodel. After running the macro, the user should insure that all other loads are applied on the submodel, and the analysis can proceed.

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

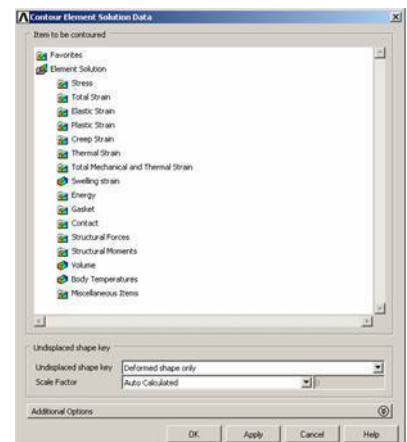
## ANSYS News: Version 8.1 Released

The release of ANSYS 8.1 contains the following useful new features and updates in structural analysis:

- Enhanced contact pair management: Can now run a partial solution to determine the initial configuration and post-process contact quantities at time 0.
- Nonlinear diagnostics: Can track specific contact quantities during solution.
- Post-processing: The GUIs for plotting nodal and element contours have been improved.
- Tabular loading: Tabular loads can now support loading as a function of X, Y and Z of a local coordinate system.

- Equivalent Stress for PSD Analysis: The calculation of equivalent stress has been improved for random vibration (PSD) analysis using the Segalman-Reese algorithm.
- Macro error handling: If repeated warnings occur during application of a macro, a dialog box now appears and allows the user to stop and exit the macro cleanly.

*More new features can be found at:*  
[www.ansys.com/ansys/new\\_features/index.htm](http://www.ansys.com/ansys/new_features/index.htm)



*New Element Solution Contour GUI*

## CAEA Welcomes Dr. Hsin-Hua Tsuei as Manager of CFD

Dr. Tsuei comes to CAEA from the University of Tennessee Space Institute, where he was a research assistant professor. He has over 15 years of experience in the general thermal-fluids area, including aerodynamics, fluid dynamics, thermal analysis, and combustion, with industrial applications to turbomachinery (compressors, turbines, and pumps), rocket engines, auxiliary thrusters, regenerative film cooling, pulse detonation engines, projectile aerodynamics, spray combustions, and chemical vapor depositions. Dr. Tsuei is intimately familiar with CFD code

development, including algorithms of time-marching implicit and explicit numerical methods, space-time CE/SE methods, finite volume formulations, moving grid/boundary, unstructured mesh with mixed elements capability, preconditioning methods for complex fluids of arbitrary equation of state, and physical modeling such as turbulence and cavitation.

He has supervised the development of commercial turbomachinery CFD design software, and has also won SBIR awards from the National Science Foundation and NASA.



## Introducing AI\* Environment 5.0

ANSYS has just released the latest version of AI\*Environment, a pre- and post-processing software dedicated to structural analysis. It uses the established market-leading Icem CFD meshing technology, specially geared towards meshing structural models. Version 5.0 is a major upgrade for AI\*Environment and can be run within the ANSYS Workbench Environment under the Advanced Meshing Tab. With powerful tools to clean or simplify the geometry - including feature detection/removal and mid-surfacing - AI\*Environment grants the user total control over a geometric model.

Specific capabilities include the following:

- Robust CAD import from Catia, Pro/E, UG, Parasolid, IDEAS, SolidWorks, SolidEdge, IGES, ACIS, and others
- Direct link to AI\*Environment from within the ANSYS Workbench environment

- Surface-based patch-independent meshing with the ability to ignore specific features and to pave over imperfections
- Powerful Tetra, Hexa, and Hex-dominant meshing capabilities.
- Mid-surface extraction including thickness property generation for tapered sections.
- A wide range of geometry repair and simplification tools.
- Export to ANSYS, LS-Dyna, NASTRAN, and ABAQUS.
- Import a mesh from ANSYS, LS-Dyna, NASTRAN, and STL.
- Numerous element quality and mesh error checks.
- Advanced mesh smoothing, mesh refinement, and mesh coarsening tools.



AI\*Environment Mesh

Please refer to <http://www.ansys.com/ansys/aienvironment/index.htm> for more details.

## Manager's Corner: ParaMesh

Do you have archived finite element models that took weeks or even months to develop and yet have no associated CAD model? Do you have the need to evaluate parts that are out of spec, damaged or require minor design changes? Do these models have boundary conditions, loads, couples, etc. that would require significant effort to replace upon remeshing? If your answer is yes to any of these questions, ANSYS has a new tool that can save you time and money.

ANSYS ParaMesh provides the capability to alter a finite element mesh without the requirement of having solid model or CAD geometry associated with it. This enables multiple model configurations to be tested without having updated geometry. Therefore, even legacy models may now be parameterized. With ANSYS ParaMesh, only nodal coordinates are modified. No new elements are created and none are deleted.

The old mesh is transformed based on projections of the existing node/element geometry or mapped to an imported CAD model. The "morphed" meshes are smoothed and then exported for analysis. Model rebuilds that previously took weeks are made in a matter of minutes using ParaMesh.

ParaMesh defines the geometry changes parametrically and can pass these parameters along to ANSYS or other codes in order to perform DOE, DFSS, or optimization studies. ParaMesh can work with ANSYS, NASTRAN (MSC and EDS NX), MSC PATRAN, as well as a variety of CFD files.

To sign up for a one-hour Webinar on ParaMesh, please go to:

<http://www.ansys.com/ansys/paramesh-md.htm>.

Please enter CAEA in the "Referred By" field.



stiffening bracket without adding mass

## 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 2004

### August 2004

3rd-5th Multiphysics Simulation for MEMS  
17th Intro ANSYS Workbench  
18th ANSYS DesignModeler/DesignExplorer  
19th Adv ANSYS Workbench

### September 2004

13th-15th Intro ANSYS Part I  
16th-17th Intro ANSYS Part II  
21st-23rd FEA Best Practices  
28th FEA Short Course  
29th CFD Short Course

### October 2004

5th ANSYS Custom & Programming (APDL)  
6th UIDL/TC-TK (GUI Customize Tool)  
12th-14th ANSYS Structural Nonlinearities  
19th-21st ANSYS Heat Transfer

### November 2004

8th-10th Intro ANSYS Part I  
11th-12th Intro ANSYS Part II  
16th Intro ANSYS Workbench  
17th ANSYS DesignModeler/DesignExplorer  
18th Adv ANSYS Workbench

### December 2004

7th-8th ANSYS Dynamics  
9th-10th ANSYS Random Vibrations

*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)

*To schedule an onsite class at your facility or to discuss a custom class, please contact Peter Barrett*

## ANSYS DesignModeler

ANSYS DesignModeler, formerly known as AGP, is a modeling and geometry editing tool specially geared toward simulation. Modeling requirements for simulation can be quite different than those provided by standard CAD software. DesignModeler is a Workbench application from ANSYS which fills this void by allowing users to easily modify CAD geometry to make it more amenable to simulation. Some examples are:

- Splitting surfaces for load application
- Splitting bodies for mapped/sweep meshing or better mesh control
- Merging parts into one body
- Defining welds
- Feature deletion/creation
- Surface extraction
- Surface thickening

In addition to editing CAD geometry, DesignModeler offers a wide range of its own geometry creation features in order to create fully parametric models from scratch or to augment imported CAD models. While not intended to replace CAD systems for geometry creation, it offers parametric modeling capability in their absence.

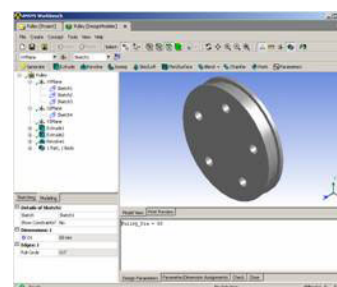
Unique to DesignModeler is its ability to create parametric concept models using a combination of beam, shell, and solid elements for solution in DesignSimulation. These types of models are very efficient from a simulation standpoint, allowing fast product decisions early in the design cycle.

DesignModeler is easy to learn and use. It can read geometry from any Workbench supported CAD system. Parametrically associative geometry will remain associative. Geometry output formats include Parasolid, IGES, and ANSYS ANF files.

Please visit:

<http://www.ansys.com/ansys/designmodeler.htm>

for additional information.



*Parametric modeling  
with DesignModeler  
In an Assembly*

*For a brief web  
demonstration,  
please go to:*

[http://www.ansys.com/ansys/  
designmodeler/dm\\_front\\_page.  
htm?referrer=caea](http://www.ansys.com/ansys/designmodeler/dm_front_page.htm?referrer=caea)