

The Focus



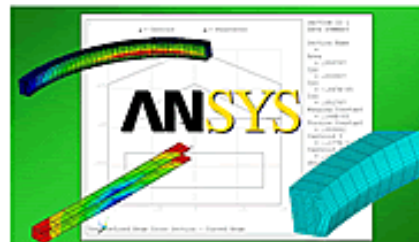
A Publication for ANSYS Users

Contents

Feature Articles

- [Beam Analysis with ANSYS](#)
- [ANSYS Inc. Announces AI*ENVIRONMENT](#)
- [Changing the File Locking and Blue Screen Features](#)

Using Beams In ANSYS

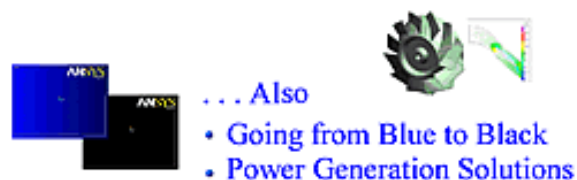


On the Web

- [PADT: Power Generation Solutions](#)

Resources

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



ANSYS
Support Distributor

© 2002, by Phoenix Analysis & Design Technologies, Inc. All rights reserved.

The Focus



A Publication for ANSYS Users

Beam Analysis with ANSYS

by [Eric Miller](#) of PADT

Note: The following Microsoft PowerPoint presentation may also be accessed directly [here](#) (1.19 MB).

SECTION ID 1
DATA SUMMARY

Section Name	=
Area	= .559787
Iyy	= .033927
Iyz	= -.647E-05
Izz	= .051787
Warping Constant	= .164E-03
Torsion Constant	= .059882
Centroid Y	= -.177E-01
Centroid Z	= .33E-01
Shear Center	= .33E-01

Use Defined Beam Cross Section - Curved Beam

PADT Phoenix Analysis & Design Technologies

ANSYS 2002 - 1-

AnsysPak

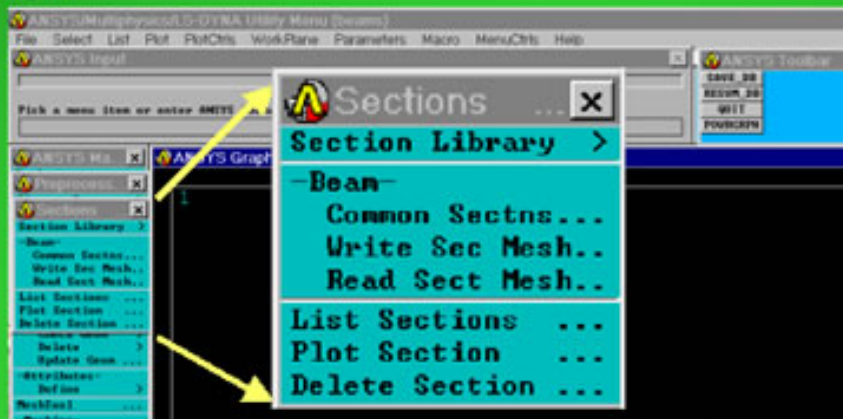
The Focus



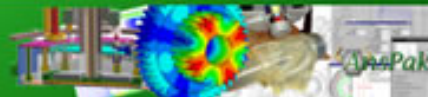
A Publication for ANSYS Users

BEAM Elements in ANSYS

Most advanced beam elements in ANSYS: BEAM188/189.
Easy to Implement with BEAMTOOL - An interactive
tool for creating common or user-defined cross sections.
BEAMTOOL is included with the standard ANSYS
package.



ANSYS Beam Overview
ANSYS
2002 - 3-



The Focus



A Publication for ANSYS Users

Many Available Beam Elements

Structural 2-D Beam			Structural 3-D Beam		
<p>Elastic Beam</p> <p>BEAM2 2 nodes 2-D space DOF: UX,UY,ROTZ</p>	<p>Plastic Beam</p> <p>BEAM23 2 nodes 2-D space DOF: UX,UY,ROTZ</p>	<p>Offset Tapered Unsymmetric Beam</p> <p>BEAM4 2 nodes 3-D space DOF: UX,UY,ROTZ</p>	<p>Elastic Beam</p> <p>BEAM3 2 nodes 3-D space DOF: UX,UY,UZ, ROTX,ROTY,ROTZ</p>	<p>Thin-Walled Beam</p> <p>BEAM4 2 nodes 3-D space DOF: UX,UY,UZ, ROTX,ROTY,ROTZ</p>	<p>Offset Tapered Unsymmetric Beam</p> <p>BEAM4 2 nodes 3-D space DOF: UX,UY,UZ, ROTX,ROTY,ROTZ</p>

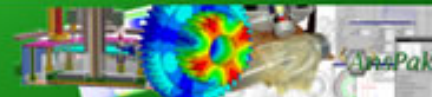
Finite Strain Beam		Explicit Dynamics
<p>Finite Strain Beam</p> <p>BEAM188 2 nodes 3-D space DOF: UX,UY,UZ, ROTX,ROTY,ROTZ</p>	<p>Finite Strain Beam</p> <p>BEAM189 3 nodes 3-D space DOF: UX,UY,UZ, ROTX,ROTY,ROTZ</p>	<p>Explicit Beam</p> <p>BEAM181 3 nodes 3-D space DOF: UX,UY,UZ, ROTX,ROTY,ROTZ, VX,VY,VZ,AX,AY,AZ</p>

Related Elements:

- 2D, 3D Spar Elements
- Mass Elements
- Linear Actuator Element
- Structural Pipe Elements
- Link Elements
- Thermal Conduction Bar Elements
- Convection Link Elements
- Thermal-Electric Elements
- Electro-Mechanical Elements
- Extensive Contact Elements
- Combination Spring/Damper Elements
- Plus More



ANSYS Beam Overview



The Focus



A Publication for ANSYS Users

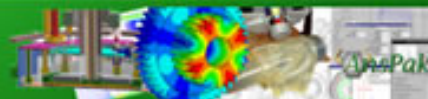
BEAMTOOL - Easy to Create Common Sections

ID	1
Name	C_Channel
Sub-Type	C
Offset To	Origin
Offset X	
Offset Y	
W1	1
W2	.9
W3	.8
t1	.1
t2	.1
t3	.1
OK Apply	
Close Preview	
Help	

Reference ID Number
 Arbitrary Reference Name
 Type of Section



Positional Options
 Schematic Diagram
 User Input Parameters

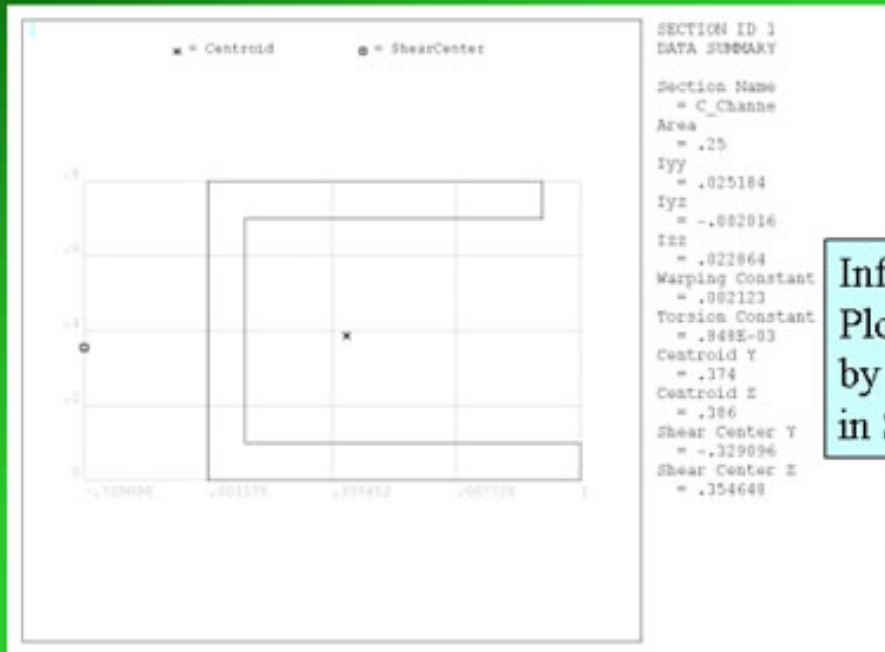


The Focus

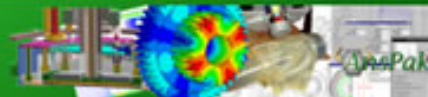


A Publication for ANSYS Users

BEAMTOOL - Easy to Verify



Informational Plot is Produced by Plot Sections in Sections Menu.

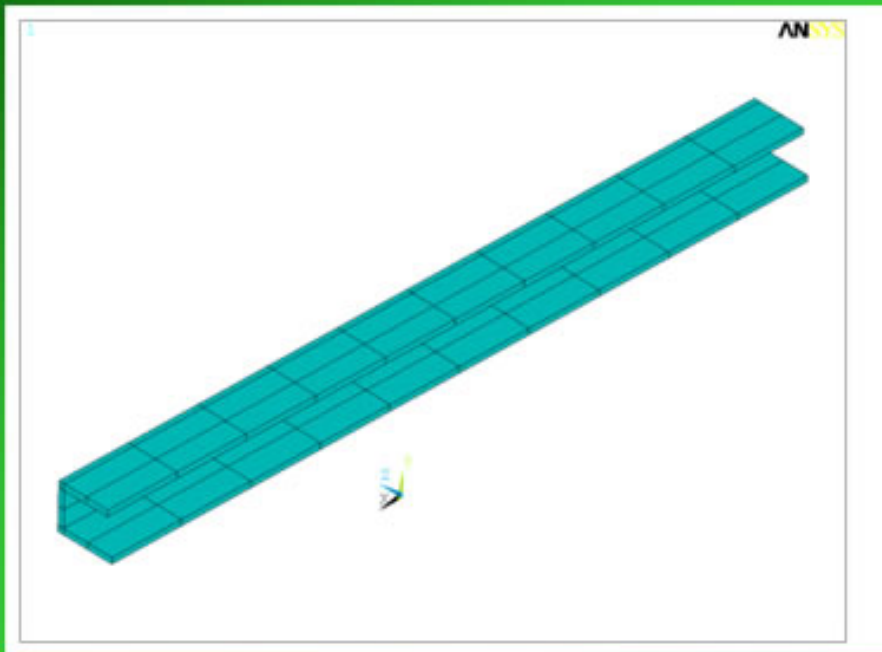


The Focus

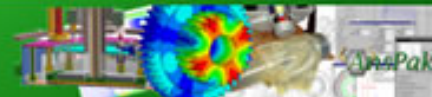


A Publication for ANSYS Users

BEAM188/189 - Easy to Visualize



Two-noded
Beam188
Elements
Displayed
with Beam
Cross Section
Added to
Plot with
/ESHAPE
Command.

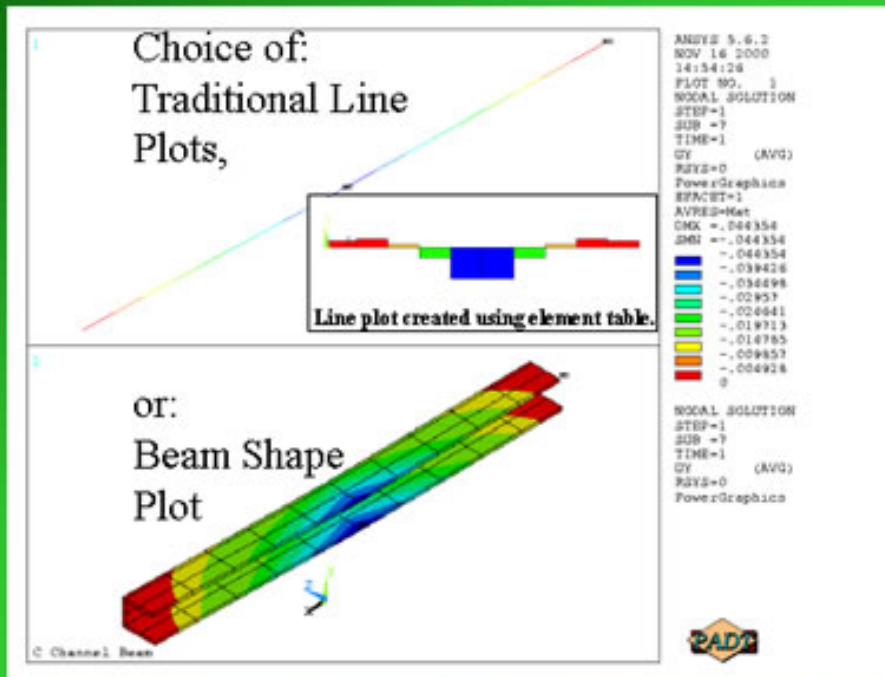


The Focus

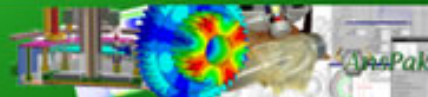


A Publication for ANSYS Users

BEAM188/189 - Easy to Postprocess Results



Beam with both ends constrained and vertical force at center of beam.



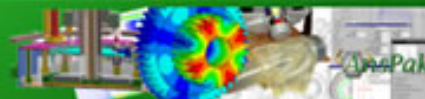
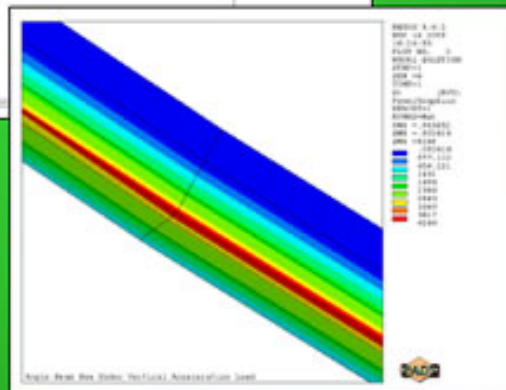
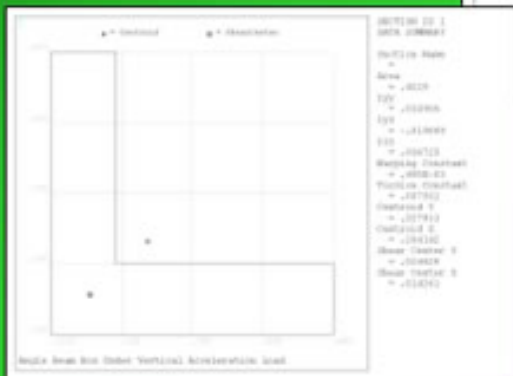
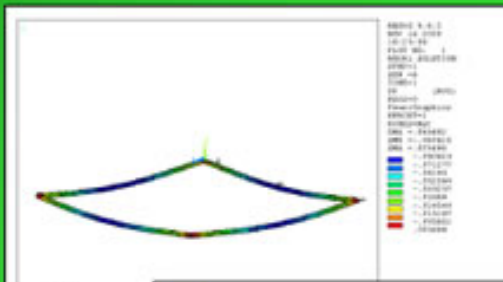
The Focus



A Publication for ANSYS Users

Other Examples

L-Angle Beam
Box Subject to
Vertical Acceleration

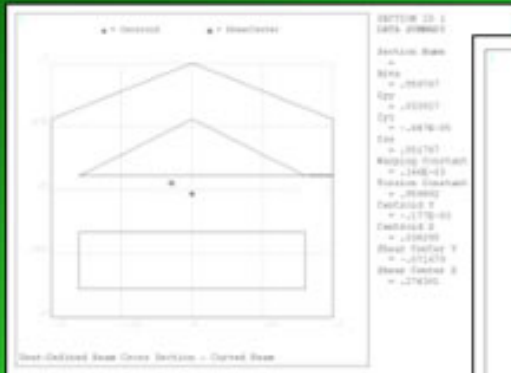


The Focus

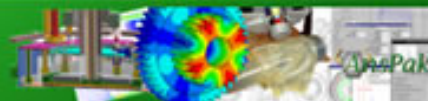
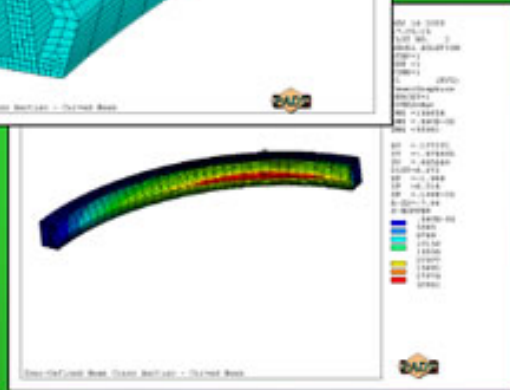
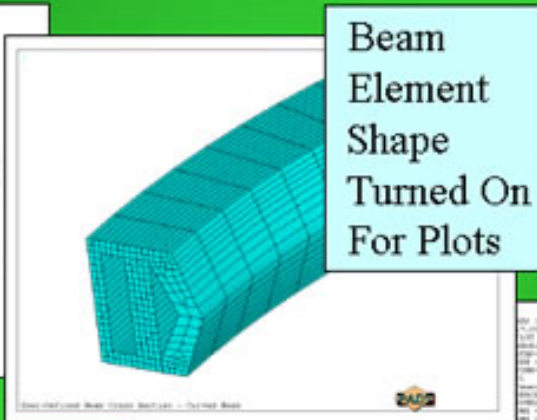


A Publication for ANSYS Users

Other Examples



User-Defined Custom Cross Section. Created In Minutes in ANSYS.



The Focus



A Publication for ANSYS Users

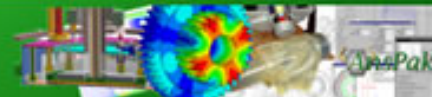
All Combined With Full ANSYS Capability

- Linear and Nonlinear Structural Static and Transient
- Extensive Contact Capability
- Dynamics, including Modal, Spectrum, Harmonic, and Random Vibration
- Linear and Nonlinear Buckling
- Thermal Steady-State and Transient
- Extensive CFD Capabilities with ANSYS/FLOTRAN
- Full Electromagnetics Capabilities
- Coupled-Field and Multiphysics Analysis
- ANSYS Parametric Design Language makes ANSYS Fully Customizable
- Full Parametric Capability
- Design Optimization Routines

ANSYS

Plus much more!

Contact PADT for More Details



The Focus



A Publication for ANSYS Users

ANSYS Inc. Announces AI*ENVIRONMENT

New Generation Pre- and Post-Processing Solution Combines Technologies from ANSYS Inc. and ICEM CFD Engineering to Serve Mechanical Industry

Canonsburg, PA November 28, 2001 ANSYS®, Inc. (NASDAQ: ANSS), the global innovator of simulation software and technologies aimed at optimizing product development processes, today announced "AI*Environment", a new generation of pre- and post-processing solutions for the mechanical engineering industry.

AI*Environment combines ICEM CFD Engineering's pre- and post-processor technologies with ANSYS Inc.'s simulation expertise. Aimed at the computer-aided engineering community, AI*Environment allows modeling and meshing for structural, thermal and CFD projects and supports CAD interfaces with CATIA, Pro/Engineer, SDRC IDEAS, Solid Works and Unigraphics. The new solution enables users to build and prepare models within their CAD system that can be exported directly to AI*Environment for meshing. AI*Environment provides links to various analysis solvers and supports direct CAD meshing for surfaces and solid models in addition to batch or interactive assemblies.

"AI*Environment is a true leader in CAD-based meshing and post-processing technology to address the needs of mechanical engineers. This solution provides the state-of-the-art geometry, input, repair and meshing tools necessary to solve the most complex meshing issues," stated Michael Wheeler, vice president of marketing for ANSYS Inc. "ANSYS Inc. continues to provide technologically advanced solutions needed to support the engineering design process and decrease the associated development costs."

AI*Environment is expected to ship in early 2002. Visit <http://www.ansys.com/> for more information.

The Focus



A Publication for ANSYS Users

About AI*Solutions

AI*Solutions from ANSYS Inc., combines the state-of-the-art, technology from ICEM CFD Engineering with ANSYS Inc.'s simulation expertise to provide product development teams the latest simulation products and technology platforms. AI*Solutions is continually expanding to include:

- **AI*Workbench**: An open and flexible application development technology platform that enables customers and partners to create customer- and industry-specific engineering simulation templates to meet their specific needs.
- **AI*EMAX**: A family of high-frequency electromagnetic analysis products for the electronics industry that supports the functionality to analyze RF/microwave passive components and circuits, electromagnetic interference and compatibility (EMI/EMC), antenna design, and object identification. AI*EMAX includes leading-edge pre- and post-processing technology from ANSYS Inc.'s ICEM CFD Engineering subsidiary.
- **AI*Environment**: The next generation of general pre- and post-processing tools for mechanical engineering. AI*Environment includes the industry's leading meshing technology from ANSYS, Inc's ICEM CFD Engineering subsidiary.

The Focus



A Publication for ANSYS Users

About ANSYS Inc.

ANSYS Inc., founded in 1970 as Swanson Analysis Systems, Inc., develops and globally markets engineering simulation software and technologies widely used by engineers and designers across a broad spectrum of industries, including aerospace, automotive, manufacturing, electronics and biomedical. Headquartered at Southpointe in Canonsburg, PA, ANSYS Inc. employs 400 people and focuses on the development of open and flexible solutions that enable users to analyze designs directly on the desktop, providing a common platform for fast, efficient and cost-conscious product development, from design concept to final-stage testing and validation. ANSYS Inc. distributes its ANSYS®, DesignSpace®, AI* Solutions" and ICEM-CFD Engineering products through a network of channel partners in 37 countries, in addition to its own direct sales offices in 18 strategic locations throughout the world.

CONTACT:

Dawn Tappy

ANSYS Inc.

PR Manager

globalpr@ansys.com

ANSYS is registered in the U.S. Patent and Trademark Office. All other trademarks and registered trademarks are the property of their respective owners.

The Focus



A Publication for ANSYS Users

Changing the File Locking and Blue Screen Features

Sometimes a user desires to disable a new feature included in a release of ANSYS. Two such features which are commonly deactivated are the pretty blue gradient background introduced in ANSYS 5.7 and also the jobname.lock file introduced in ANSYS 6.0.

The blue background does not disappear when doing a hardcopy plot to file or printer, and can be taxing on a blue printer cartridge. To disable the blue background, insert the following line of text in your start60.ans file located in the folder Docu in the ANSYS installation directory: `/ui,pbak,off`



Before



After

If you don't know where your start60.ans file is located try something like:
C:\Program Files\Ansys Inc\ANSYS60\Docu\start60.ans.

This start file is read each time ANSYS is started, and all the ANSYS commands inside this file are executed. It is similar to the old AUTOEXEC.BAT files of DOS. Many other commands are in this file, most of them commented out with the ! symbol.

ANSYS 6.0 introduced a new file named jobname.lock in the working directory. If ANSYS detects this file's existence on startup, it will ask if you are sure you want to open ANSYS in this directory. It is trying to warn you that you may already be running an ANSYS application in that directory, and that it could interfere.

Unfortunately, if ANSYS crashes it doesn't clean up the jobname.lock file. In those cases, you must hit the Y or N key to fire up ANSYS. To disable this feature, add one of the following environment variables to your system:

UNIX: setenv ANSYS_LOCK=OFF

Windows: ANSYS_LOCK=OFF

The Focus



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.