

# ANSYS Multiphysics

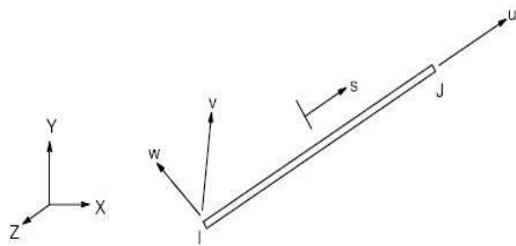
Student: Tarakanov Pavel

Speciality: Applied Mechanics

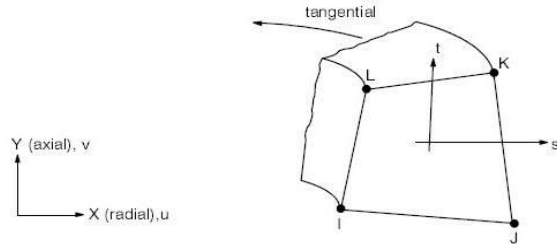
# Agenda

- Finite element method
- ANSYS Inc
- ANSYS Multiphysics

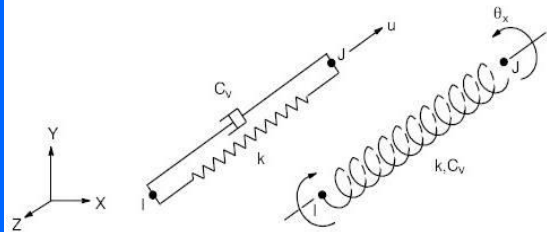
**LINK8 — 3-D Spar (or Truss)**



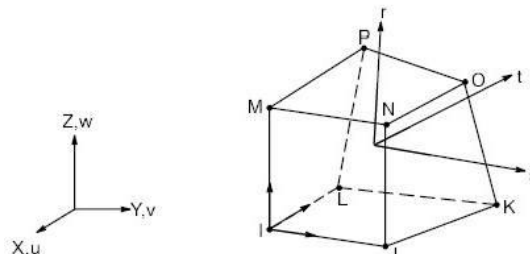
**PLANE25 — 4-Node Axisymmetric-Harmonic Structural Solid**



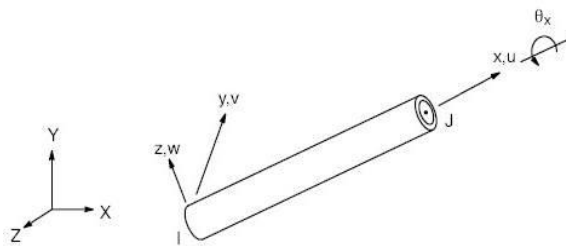
**COMBIN14 — Spring-Damper**



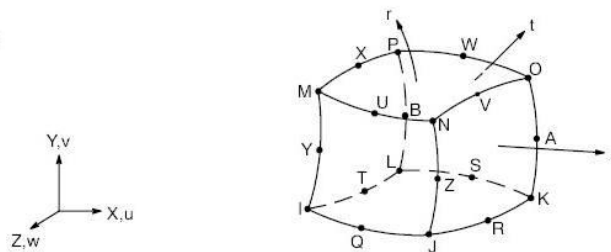
**SOLID45 — 3-D Structural Solid**



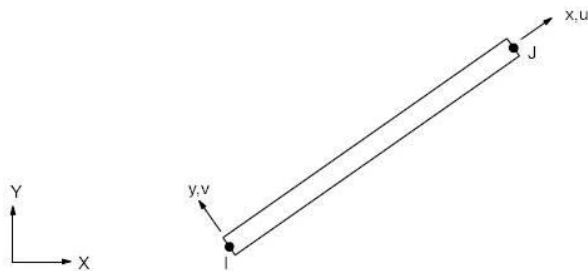
**PIPE16 — Elastic Straight Pipe**



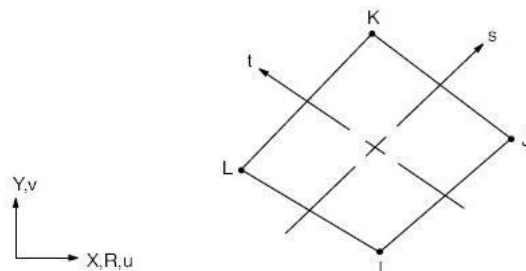
**HF120 — High-Frequency Brick Solid**



**BEAM23 — 2-D Plastic Beam**



**PLANE13 — 2-D Coupled-Field Solid**



**Basic steps in Finite element method**

1. Create and discretize the solution domain into finite elements.
2. Assume a shape function to represent the physical behavior of an element.
3. Develop equations for an element.
4. Construct the global stiffness matrix.
5. Apply boundary conditions, initial conditions, and loading.
6. Solve algebraic equations to obtain nodal results, such as displacements values at different nodes etc.
7. Obtain other important information.



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
U.S.A.

Toll-Free: 1.866.267.9724  
Toll-Free Mexico: 001.866.267.9724  
724.746.3304  
ansysinfo@ansys.com  
www.ansys.com



ANSYS is registered in the U.S. Patent and Trademark Office.  
© 2005 ANSYS, Inc. All Rights Reserved.

ANSYS, ANSYS Workbench, CFX, AUTODYN and any and all ANSYS, Inc. product and service names are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICGM CFD is a trademark licensed by ANSYS, Inc. All other trademarks or registered trademarks are the property of their respective owners.

Some images are courtesy of BMW, Berlin Heart, The Cleveland Clinic Foundation, EMU European Union Project-BLOODSIM, Otof Granlund, Petrobras, Siemens Canada Ltd., SVA (Potsdam Germany), URS Corporation, Universidade de Coimbra-Portugal, University of Applied Science Kiel-Yacht Research Unit, University of Canterbury and University of Colorado Health Science Center.

MBR0045AUG05 Printed in U.S.A.



Achieve  
New Heights  
in Product  
Innovation



# ANSYS

"ANSYS software is an effective tool in simulating the function of the spine. Contact elements are extremely useful in modeling the complex configuration of components that touch one another. Nonlinear capabilities of the software are critical in accurately representing the material properties as well as behavior of the artificial disc and surrounding cartilage, ligaments and muscle tissue. Such powerful finite element modeling and analysis technologies are essential in studying the impact, stresses and loading in various parts of the spine. Simply put, our work could not have been done as quickly and accurately any other way."

Dr. Missoum Moumene  
Senior Principal Engineer, Research & Development  
DePuy Spine, Inc., a Johnson & Johnson Company

Businesses today – faced with mounting competitive challenges, customer demands and financial pressures – are racing to find new ways to engineer more reliable, innovative products while minimizing costs. Increasingly, these businesses are discovering that simulation-driven design and development can be a highly effective way to address this challenge.

It is not surprising that ANSYS is the catalyst for this simulation-driven evolution in computer-aided engineering. Bringing simulation and analysis to the design engineering level of product development lays the foundation for a fast, efficient and cost-effective process.



Since its inception, ANSYS has been focused on advancing engineering simulation by developing new technologies, introducing new solutions, integrating synergistic businesses and adding new partners to address a growing base of customers worldwide.

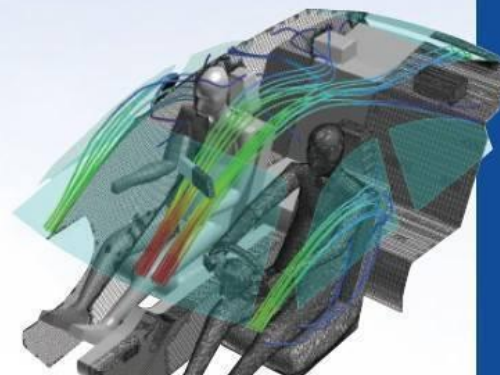
As ANSYS looks ahead, the company moves forward with a clear vision, a sound and consistent strategy, financial strength and an unwavering focus on engineering simulation – the same foundation that has led ANSYS' growth and success for more than three decades. Strategic alliances and acquisitions,

as well as industry breakthroughs, such as solving 100 million degrees of freedom and introducing a solution for fluid structure interaction (FSI), have contributed to ANSYS' growth, helping the company meet customer needs more quickly.

ANSYS looks forward to many more years of developing innovative, reliable technologies that will solve tomorrow's complex product design and development challenges in both traditional and emerging industries. ANSYS is proud to help cultivate a new generation of product design and development professionals.

ANSYS designs, develops, markets and globally supports engineering simulation solutions used to predict how product designs will behave in manufacturing and real-world environments. Its integrated, modular and extensible set of solutions addresses the needs of organizations in a wide range of industries. ANSYS' solutions qualify risk, enabling organizations to know if their designs are acceptable or unacceptable – not just that they will function as designed. ANSYS helps organizations achieve:

- Innovative, reliable and high-quality products and processes
- Fewer physical prototypes and test setups
- Faster return on investment due to reduced development time
- A more flexible and responsive information-based development process, enabling the modification of designs at later stages of development
- A front-end simulation strategy that offers a superior method for bringing products to market in less time and with fewer costs
- Seamless working exchange of data regardless of location, industry, CAD environment, etc.



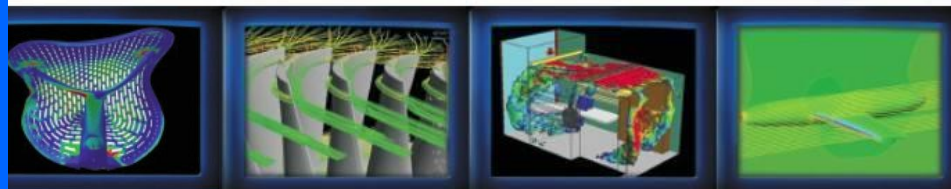
Automotive air conditioning and engine cooling systems manufacturer Behr GmbH & Co. KG of Stuttgart, Germany, investigates climatization concepts with ANSYS' ICEM CFD™ in early stages of the product development process. Behr uses ANSYS ICEM CFD software to optimize and validate climate systems with respect to passenger comfort and safety, to ensure high-end climate products.

## Engineering Simulation Focused

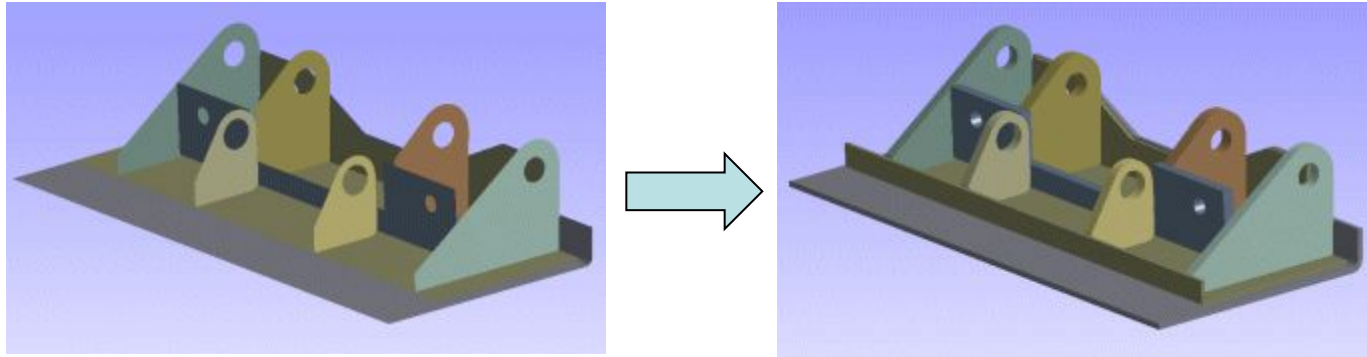
ANSYS remains uniquely committed to the core engineering simulation technologies, which set it apart from others in the CAE industry. ANSYS' solutions are unmatched in terms of the functionality and power necessary to optimize components, subsystems and systems. By deploying ANSYS solutions at various stages of product development, many leading organizations are leveraging digital design performance information to make timely decisions. Working with customers, technology partners and research institutions, ANSYS delivers the most robust, reliable and open simulation solutions – solutions that meet the ever-growing needs of organizations worldwide.

## Customer Focused

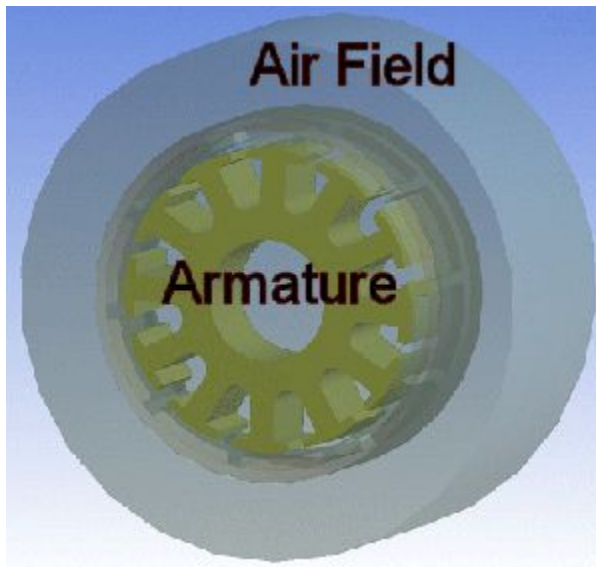
Fueling ANSYS' vision and strategy is the feedback and input of the company's growing base of customers, which includes 83 of the top FORTUNE 100 industrial companies. In turn, ANSYS commits more than 20 percent of the company's revenue to research and development – the highest in the CAE industry. This investment allows ANSYS to deliver on its vision by developing and delivering the most innovative and powerful engineering simulation solutions with unparalleled depth. Strong and reliable customer relationships have enabled ANSYS to develop easy-to-use software solutions that align with the customer process – not forcing the process to fit the solution. Many of ANSYS' industry-focused applications were driven by actual customer applications.



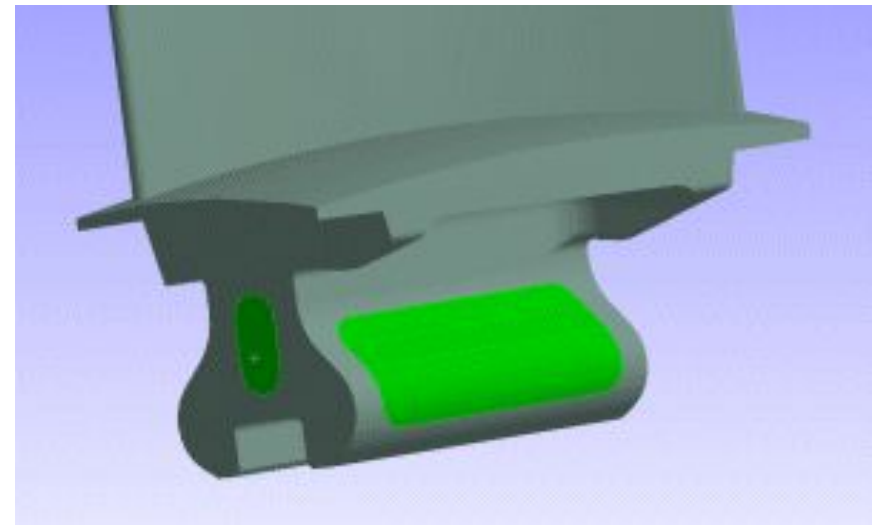
## Detail Geometry Creation



Solid geometry of an assembly

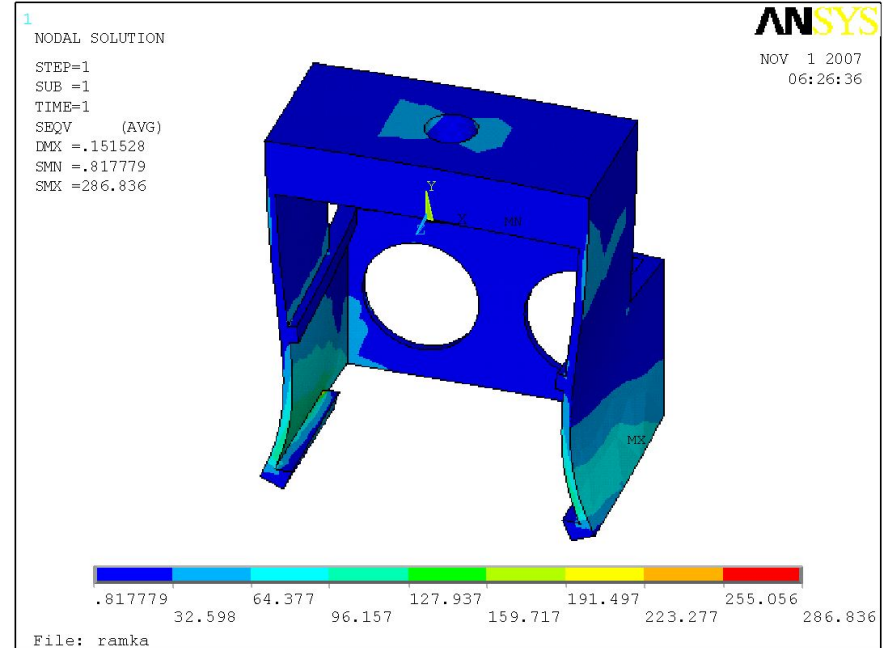
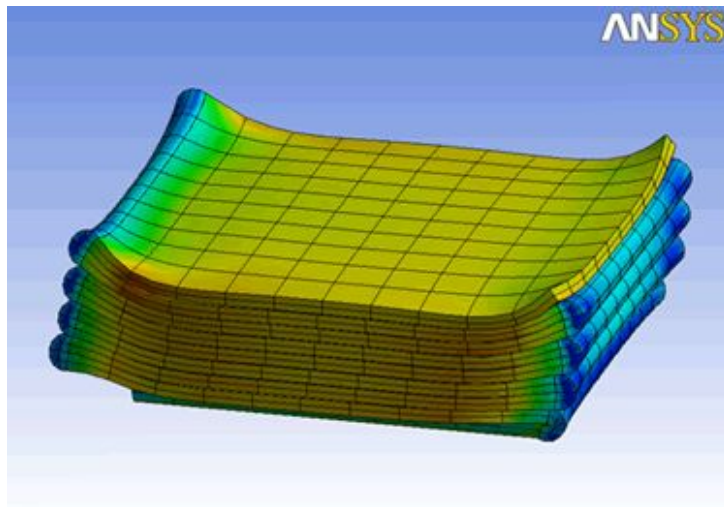
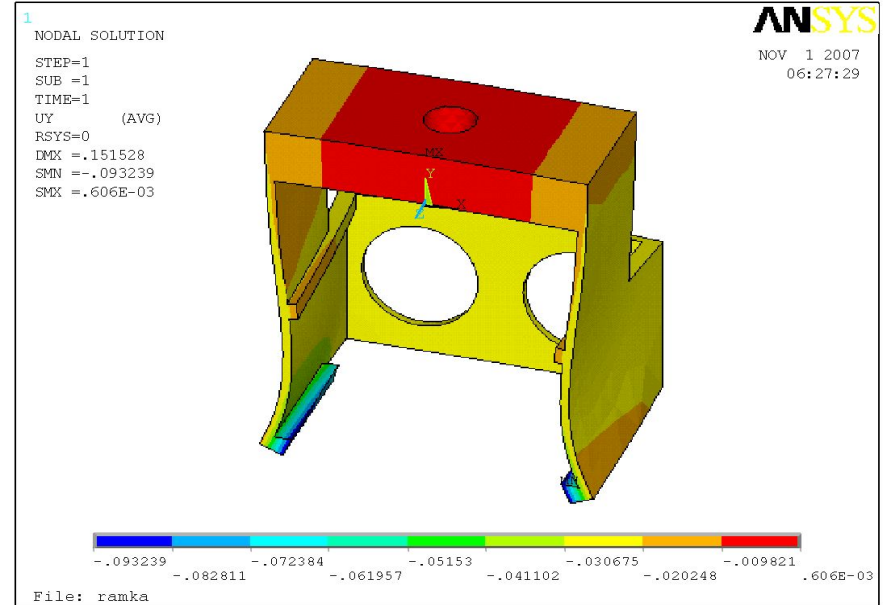
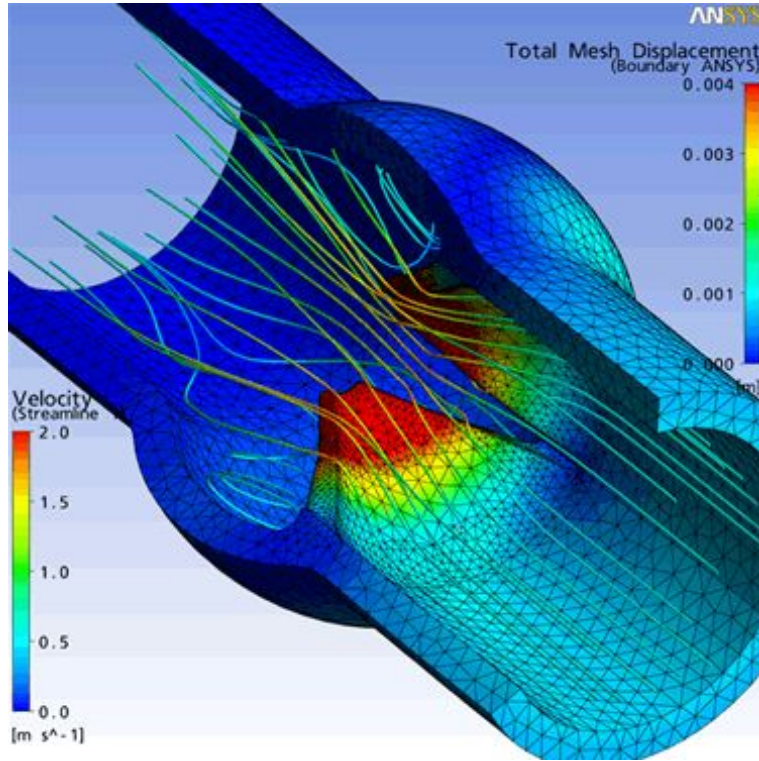


Geometry representing the air around an electric motor armature is created for a later field analysis.

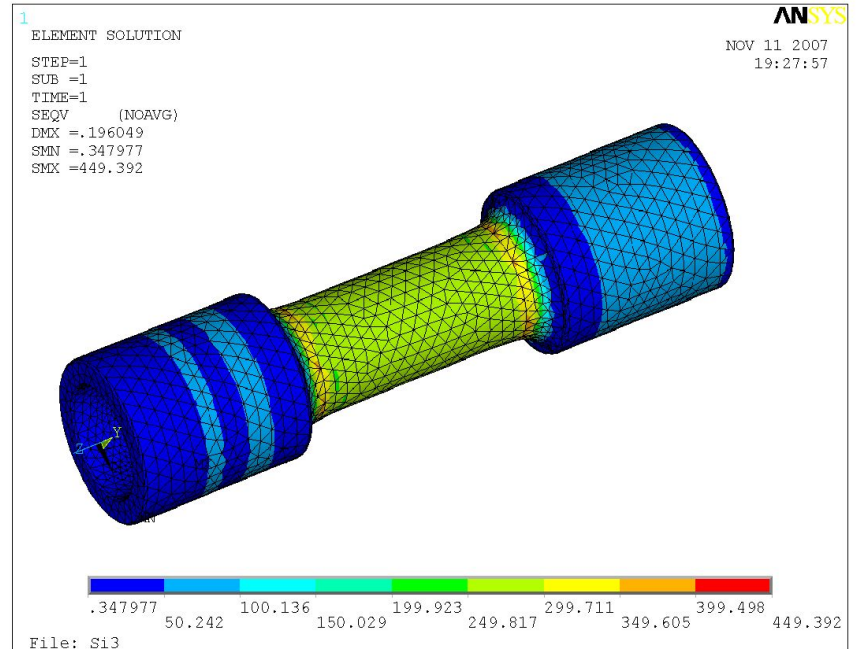
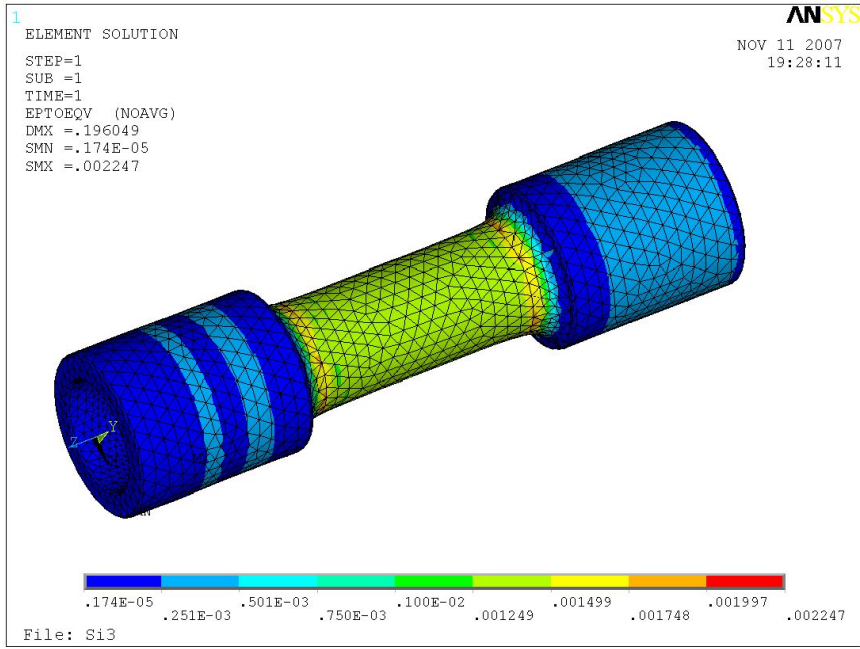
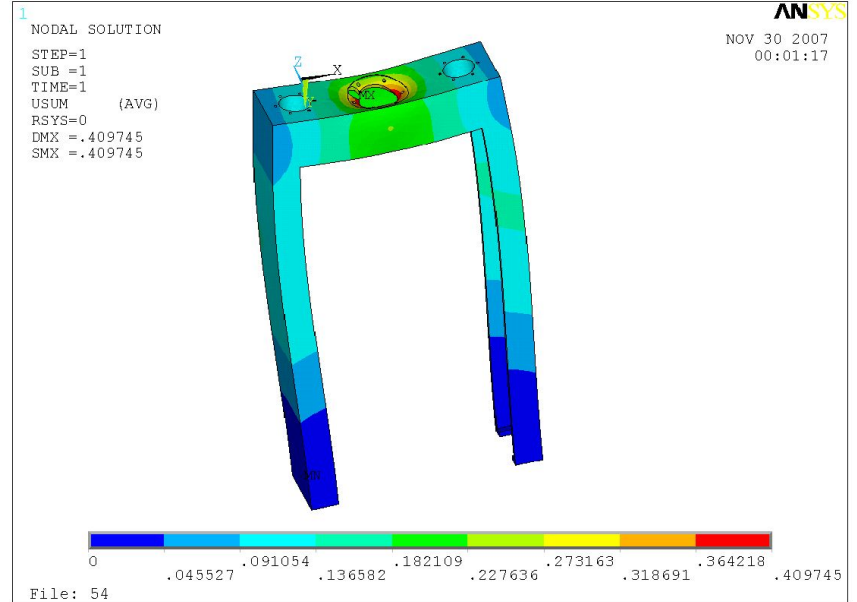
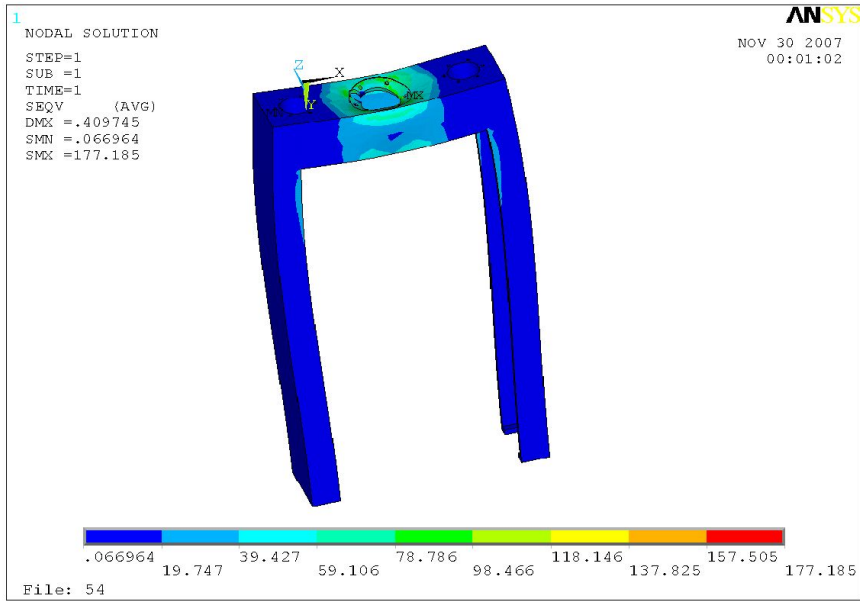


Patch surfaces created on larger areas for applying boundary conditions.

# ANSYS Multiphysics (examples)



# ANSYS Multiphysics (examples)

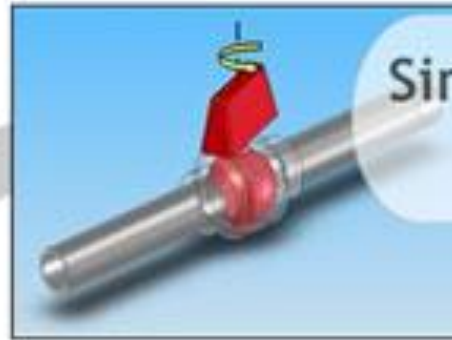




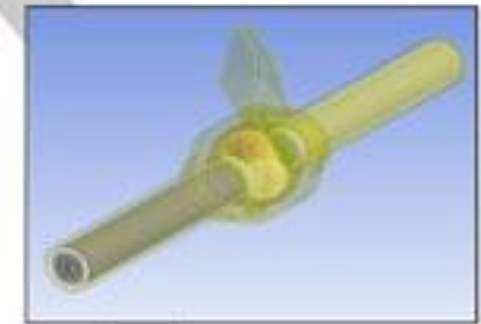
# ANSYS CFX

## CFD Design Iteration

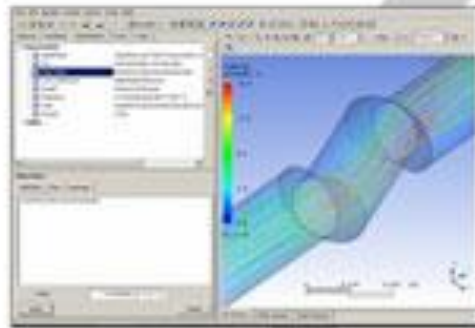
Single Design Platform  
ANSYS Workbench



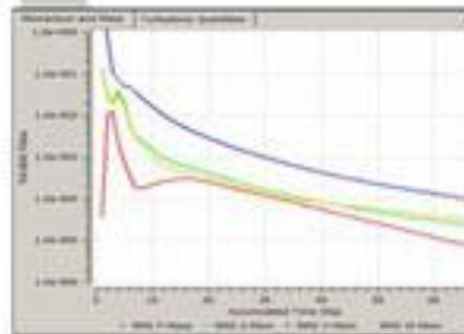
Parametric CAD Connection



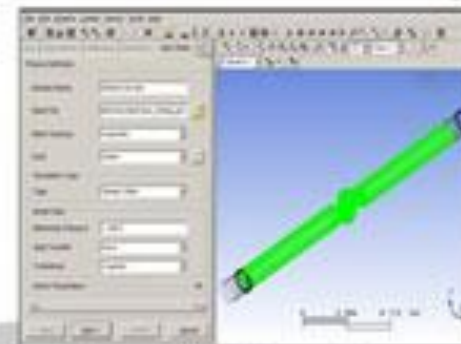
High-quality CFD mesh



Powerful post-processing



Rapid solution



Easy physics setup

- Thank you for your attention!