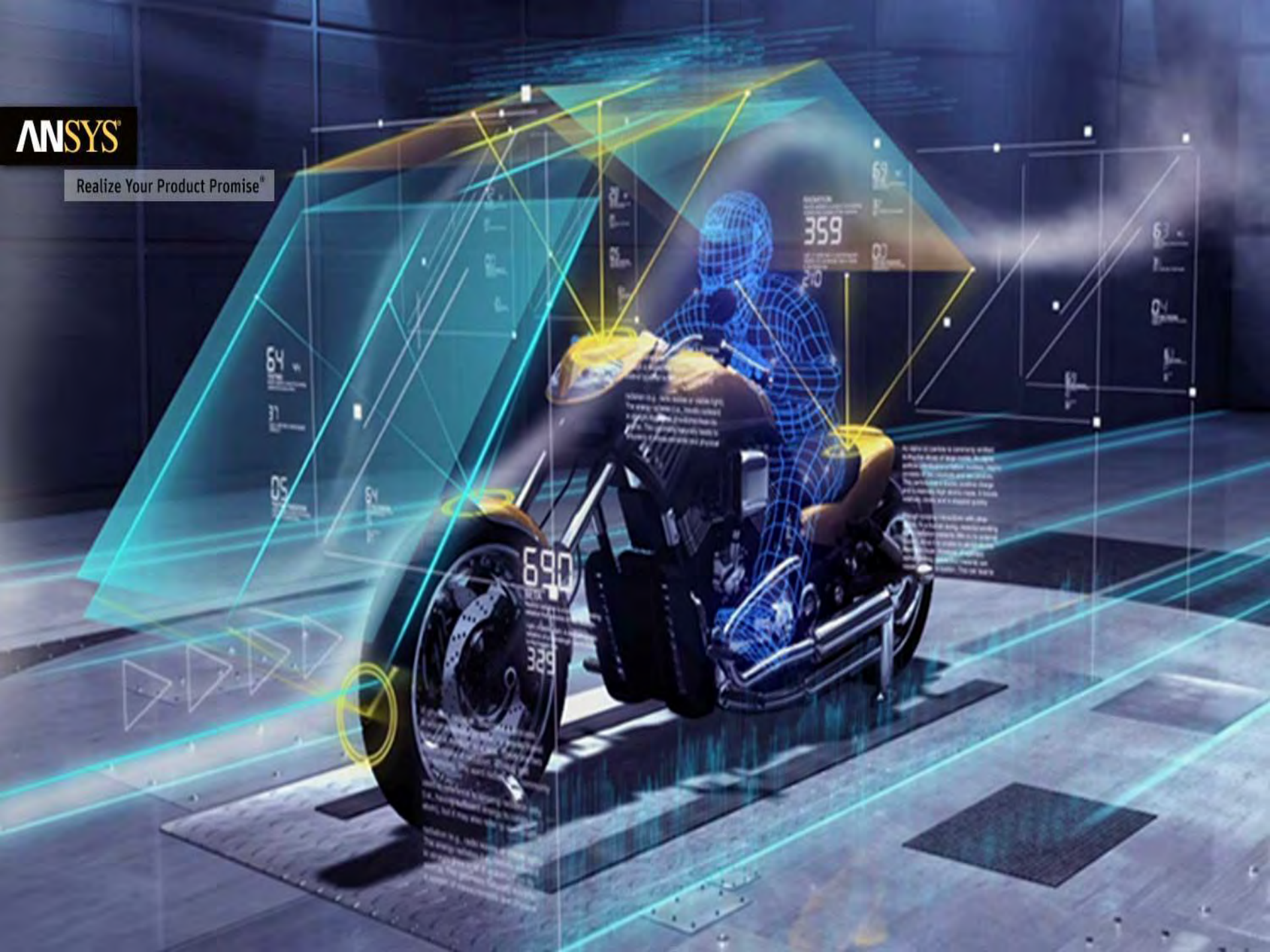
multisoft

The Basic Ansys Program

Website: www.multisoftvirtualacademy.com
Email: info@multisoftvirtualacademy.com
Contact No: +918130666206/209


ANSYS
Realize Your Product Promise®

Introduction

- ANSYS is a general purpose Finite Element Analysis (FEM) software developed by Ansys, Inc. which was founded by Dr. Jhon Swanson in 1970 in Canonsburg, Pennsylvania, USA, previously known as **Swanson Analysis Systems, Inc.** SASI
- Best known for its Ansys Mechanical and Ansys Multiphysics products
- Non exportable analysis tools incorporating pre-processing (geometry creation, meshing), solver and post-processing modules in a GUI
- General-purpose finite element modeling packages for numerically solving mechanical problems, including static/dynamic structural analysis (both linear and non-linear), heat transfer and fluid problems, as well as acoustic and electro-magnetic problems.

ANSYS Products

Featured Products	Pre-processing	Stand-alone Solvers	Application Specific
» ANSYS Mechanical	» ANSYS ICEM CFD	» ANSYS Multiphysics	» ANSYS AQWA
» ANSYS Structural	» ANSYS MeshMorpher	» ANSYS Rigid Dynamics	» ANSYS ASAS
» ANSYS Professional	» TGrid	» FLUENT for CATIA V5	» ANSYS Icepak
» ANSYS DesignModeler		» ANSYS POLYFLOW	» ANSYS TurboGrid
» ANSYS SpaceClaim Direct Modeler		» ANSYS ICEM CFD Cart3D	» ANSYS BladeModeler
» ANSYS Meshing		» ANSYS Fatigue Analysis	» ANSYS Vista TF
» ANSYS DesignXplorer			» ANSYS Airpak
» ANSYS CFD			» ANSYS Composite PrepPost
» ANSYS CFX			
» ANSYS FLUENT			
» ANSYS CFD-Post			
» ANSYS Explicit STR			
» ANSYS AUTODYN			
» ANSYS LS-DYNA			
» ANSYS EKM			

Capabilities and Features

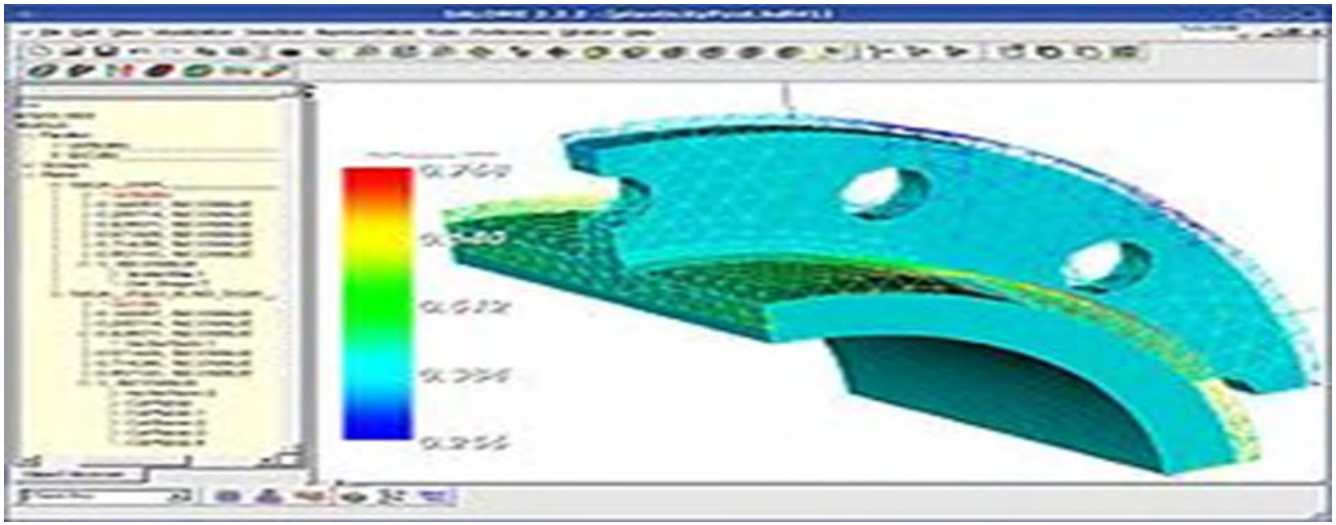
- Static structural analysis
- Dynamic structural analysis
- Dynamic response analysis
- Vibration (modal) analysis
- Fluid dynamic analysis
- Computational fluid dynamics
- Non linear analysis
- Rotor dynamic analysis
- Explicit dynamic analysis
- Thermal analysis
- Multiphase simulation
- Multi-physics simulation
- Coupled field simulations
- Fluid structural integration
- Electromagnetic simulations
- Composite material analysis
- Optimization

CONTENTS

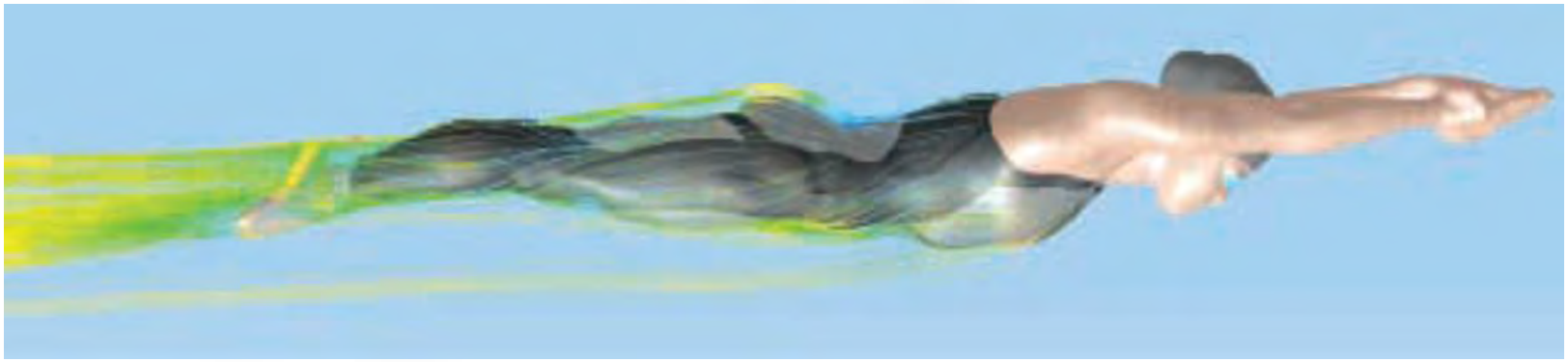
Ansys is classified by these methodology

CAE which comprises or uses

1. FEA (Finite Element Analysis)
2. Computational Fluid Dynamics (CFD)
3. Multi-body dynamics (MBD)
4. Optimization Software

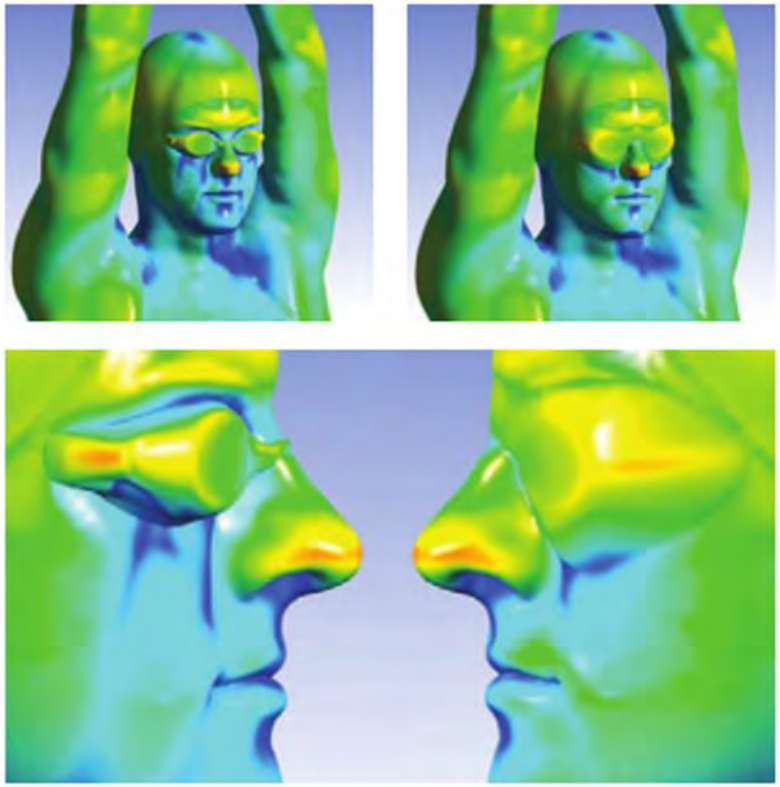


Development of swimwear for American athletes

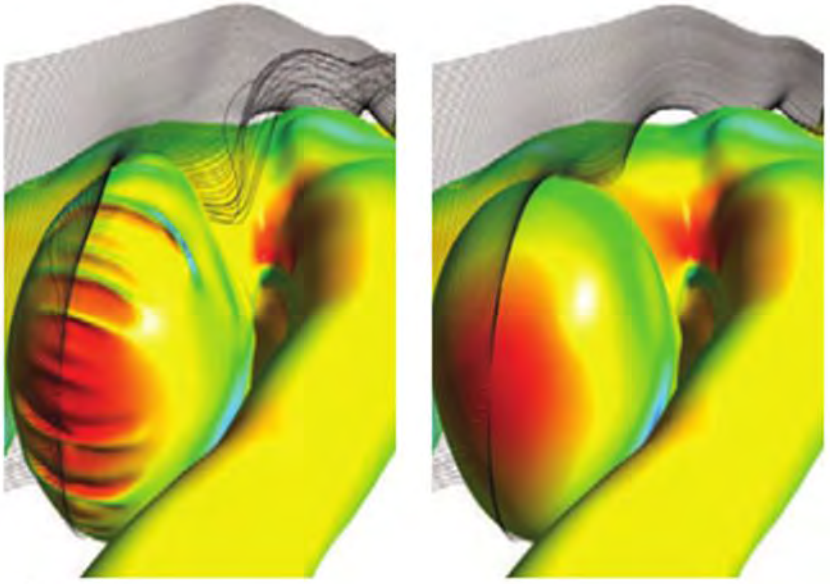


Fact: During Summer Beijing Olympics 2008, 47 Gold Medals and total 89% of all medals were won by athletes wearing *Speedo LZR RACER*

In development of Speedo Fastskin racing system



In analyzing the near-surface flow velocity fields of existing goggle design (left) and a next-generation design concept (right), Speedo engineers were able to see an immediate improvement in hydrodynamic performance.



A CFD simulation using ANSYS Fluent revealed significant flow-field disruptions – and fluidic drag – caused by an inefficient swim cap shape.

FEA

- Finite Element Analysis (FEA) is a numerical method which provides solutions to problems that would otherwise be difficult to obtain.
- Finite element analysis (FEA) is a computer simulation technique used in engineering analysis. Finite Element Analysis (FEA) was first developed in 1943 by R. Courant, who utilized the Ritz method of numerical analysis and minimization of variational calculus.

- A paper published in 1956 by M. J. Turner, R. W. Clough, H. C. Martin, and L. J. Topp established a broader definition of numerical analysis. The paper centered on the "stiffness and deflection of complex structures".

Basic Solution Approaches for solving FEA problem using ANSYS

- **Preprocessing: Defining the Problem**

1. Define key points/lines/areas/volumes,
2. Define element type and material/geometric properties,
3. Mesh lines/areas/ volumes as required.

- **Solution: Assigning loads, constraints, and solving**

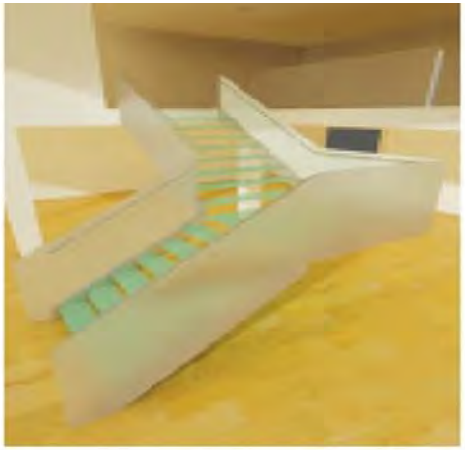
1. Loads (point or pressure),
2. Constraints (translational and rotational)
3. Solver

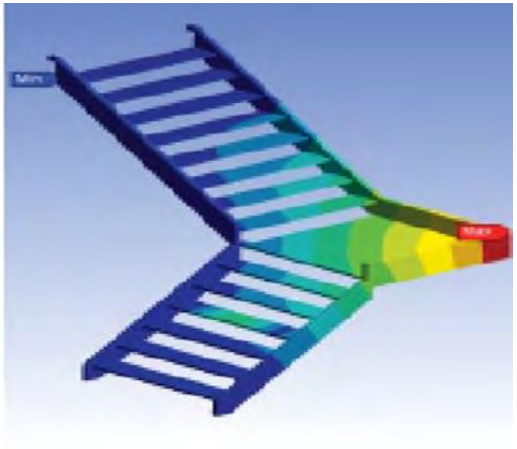
- **Post processing**

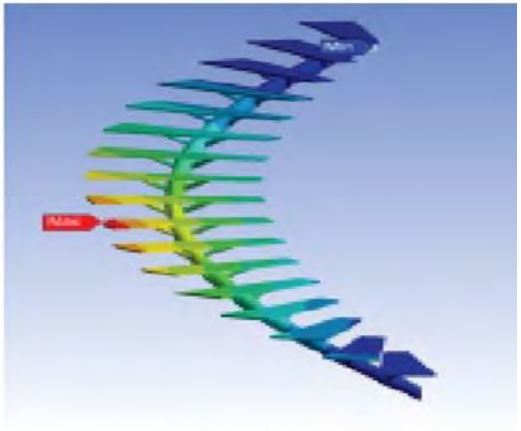
To Analyse lists of

1. Nodal displacements,
2. Element forces and moments
3. Deflection plots
4. Stress contour diagrams or temperature maps.

Application in Civil structural analysis



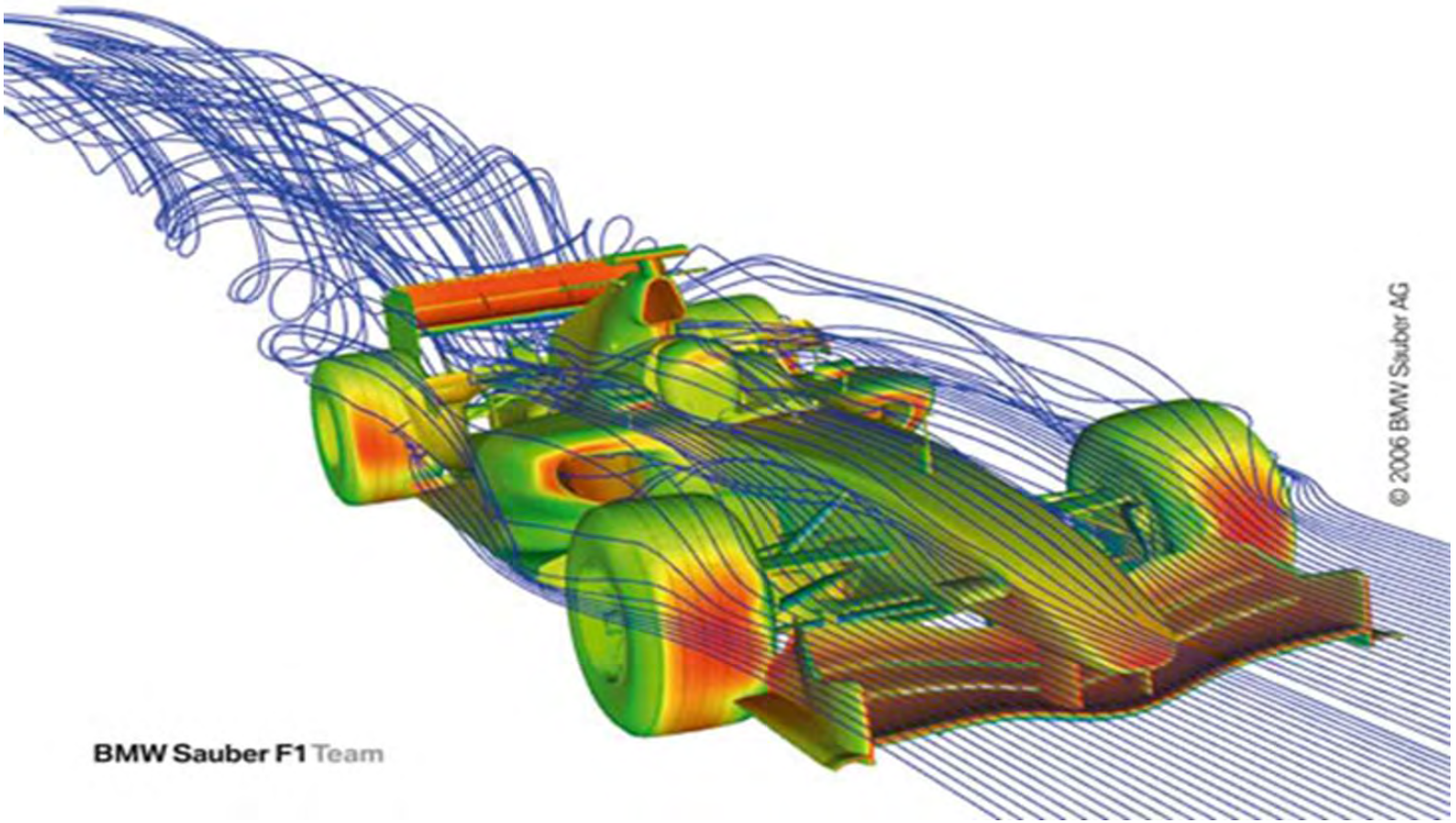




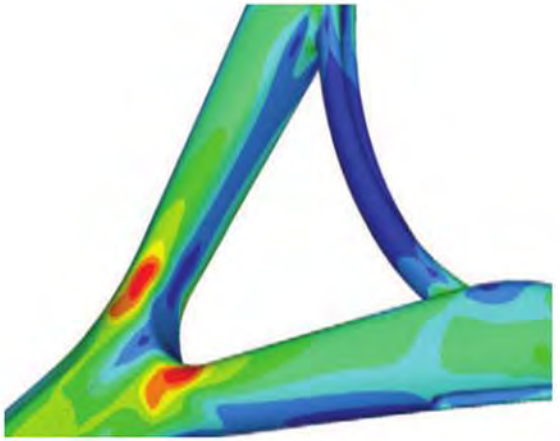
As part of a high school research project, Scott Taylor analyzed loading variations for two staircase configurations: angled (above) and curved (below).

FEA results of staircase analysis

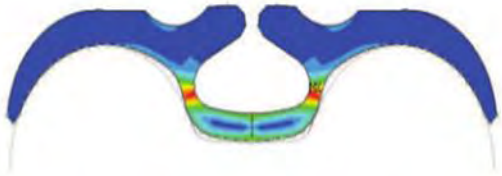
CFD Analysis Of F 1 Car



Development Lawn Tennis using HEAD Rackets



Shaft area of a tennis racket under bending load. The stress level is too high, so engineers reinforced the design.

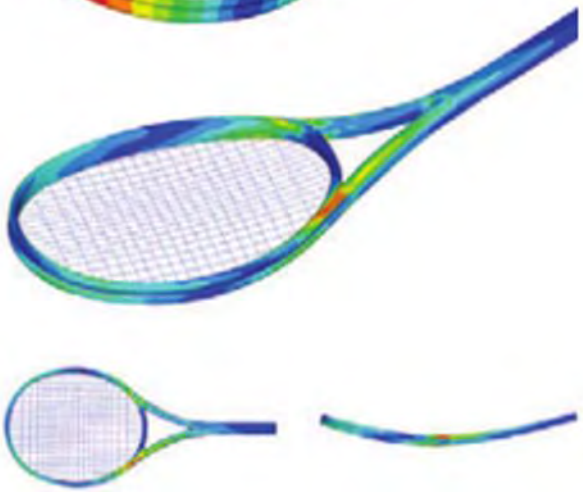


Simulation of bumper verifying that it has the right pre-stress to fit well on the frame

Study of various vibration modes for the HEAD rackets

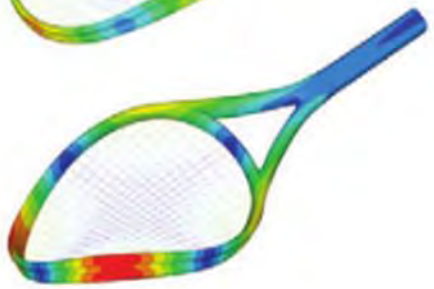
First bending mode of a strung tennis racket in elongation. Hitting the ball at the blue node line on the strings does not excite this mode. This is the sweet spot of the racket.

First bending mode in tension

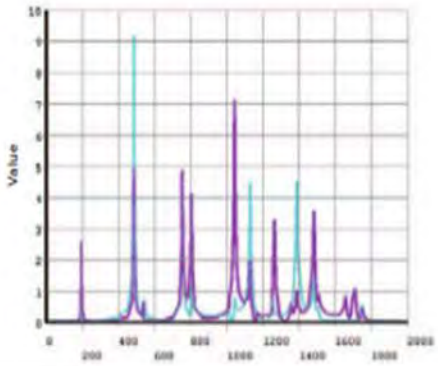


Second bending mode of a strung tennis racket

Torsion mode of a tennis racquet illustrating how a non-center hit introduces torsion



Harmonic analysis shows frequency spectrum of a strung tennis racket with a special excitation load. The spectrum shows all excited natural modes up to 2kHz. Blue represents the center strings and purple the off-center strings.



Application In Industries

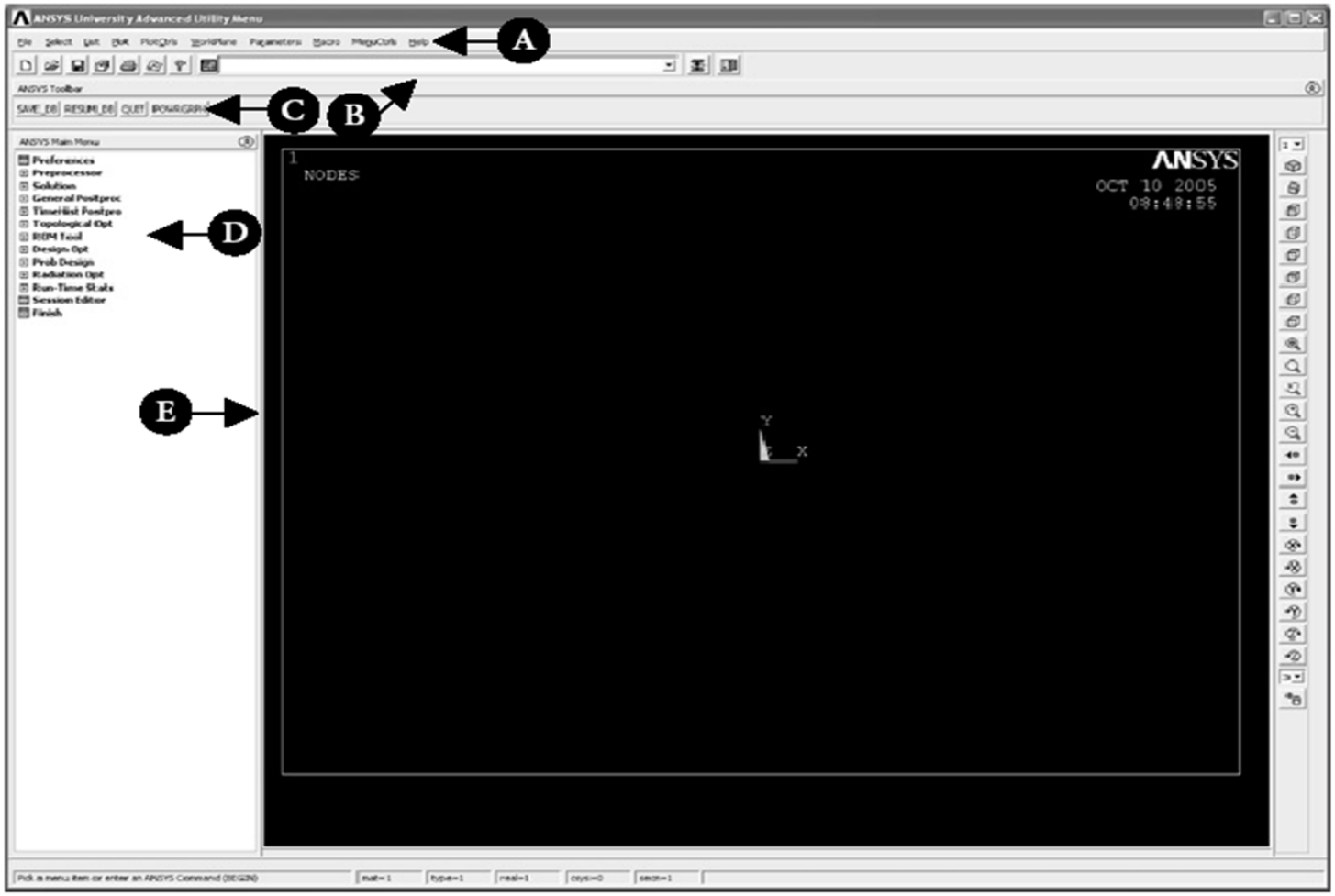
- **Aerospace**
- **Oil & Gas**
- **Automotive**
- **Built Environment & HVAC**
- **Power Generation**
- **Consumer Products**
- **Metals**
- **Turbo machinery**
- **Healthcare**
- **Chemical & Petrochemical**
- **Industrial Equipment**
- **Semiconductor**
- **Civil Engineering**
- **Government & Defense**
- **Marine & Offshore**
- **Sport & Leisure**
- **Electronics**
- **Environmental**
- **Plastics and Rubber**

Working Environments (Simulation related to Mechanical Engineering)

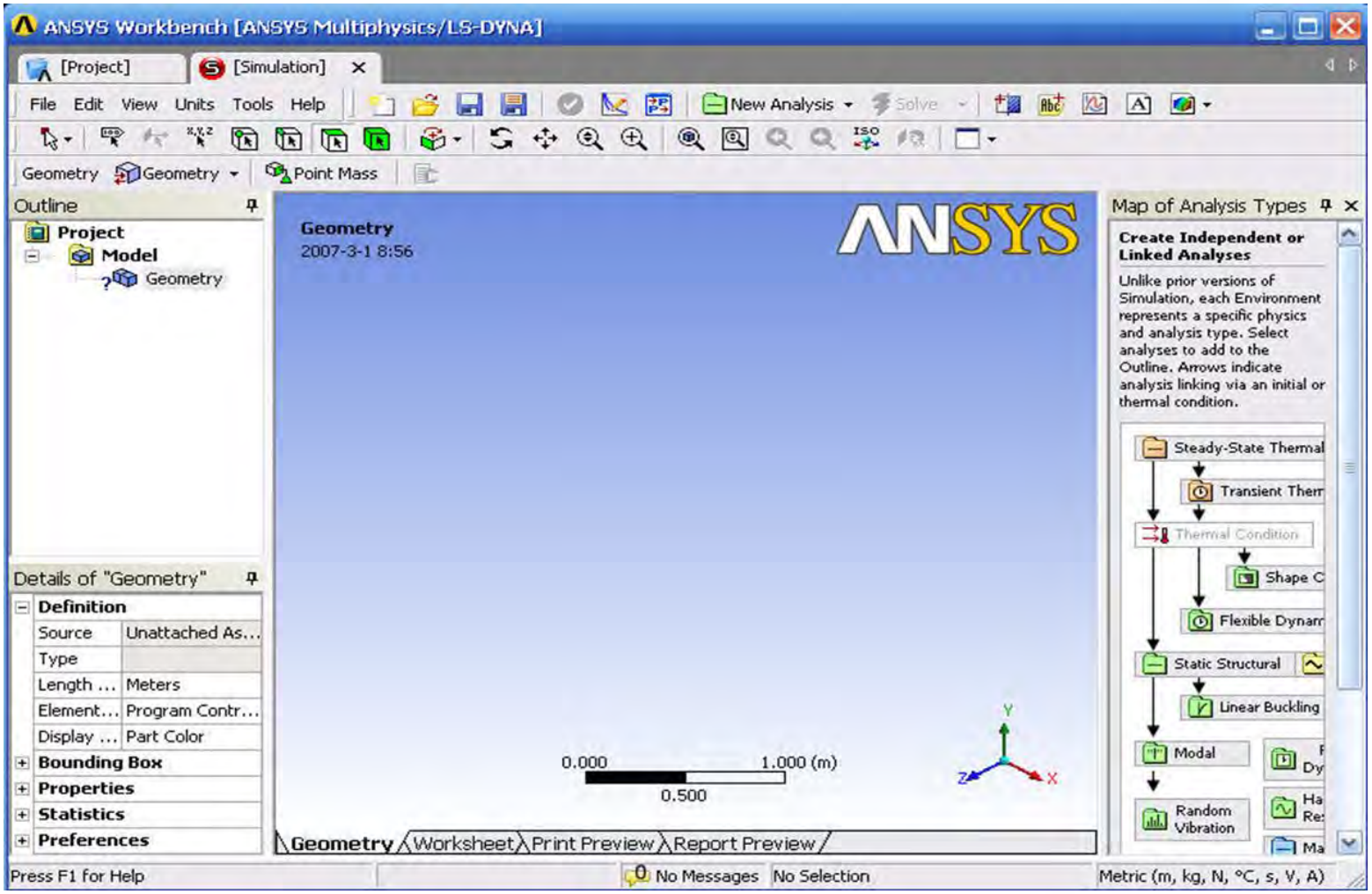
- **Ansys Mechanical (Multiphysics)**
  1. Basic working simulation environment
  2. More emphasis have been given to FEA modeling
  3. Supports user input programming capability

- **Ansys Workbench**
  1. More User friendly GUI
  2. Manipulation of problem can be done in a limited way
  3. Very easy to understand and handle

GUI of Ansys Multiphysics



GUI of Ansys Workbench



multisoft
Virtual Academy

Thank You

FOR MORE DETAILS, CONTACT UNDERSIGNED


enquiry@multisoftvirtualacademy.com


(+91) 8130666206 / 209


www.multisoftvirtualacademy.com