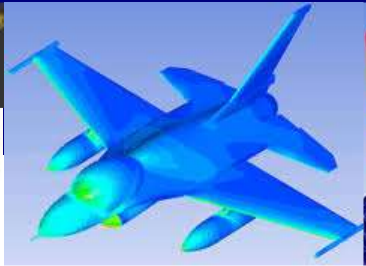
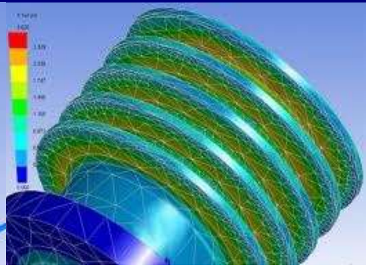
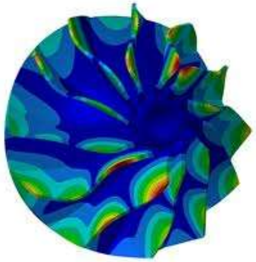
I.I.Vorovich Institute
Mathematics
Mechanics
Computer Science

Finite Element Method in Computer Simulation and Engineering

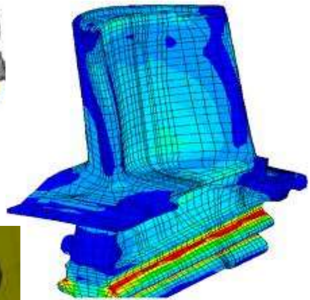
Dr. Anna A. Nasedkina

nasedkina@math.sfedu.ru

Institute of Mathematics,

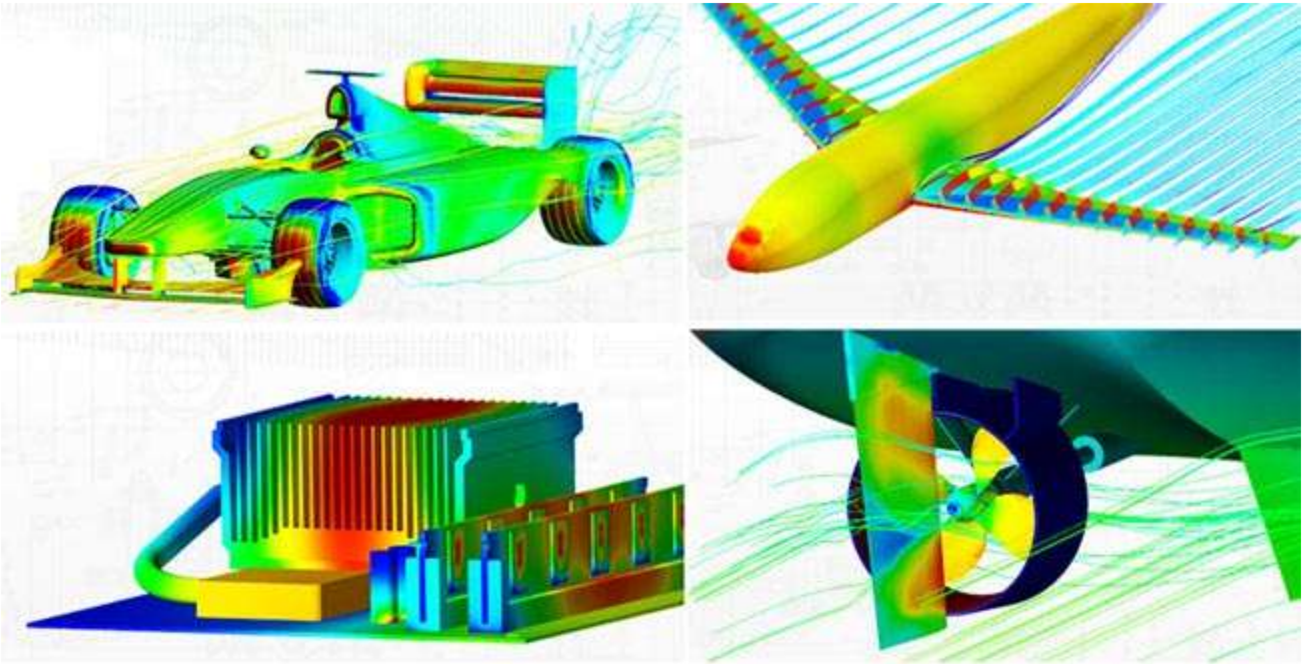
Mechanics and Computer Science

Southern Federal University


Outline of the lecture

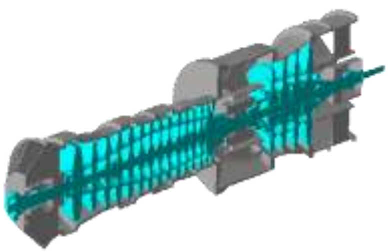
- **Computer Aided Engineering and Finite Element Analysis**
- **Main concepts of Finite Element Method**
- **Overview of Finite Element Software packages**



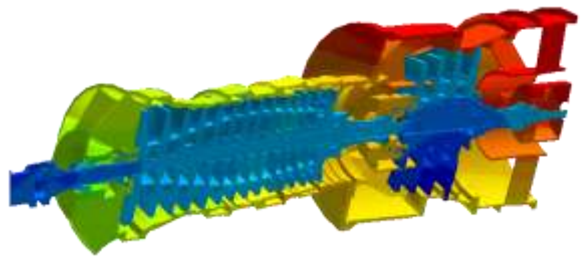
Computer Aided Engineering and Finite Element Analysis

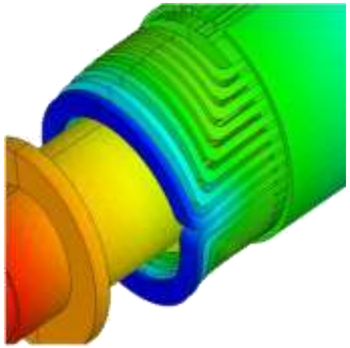
CAD/CAE/CAM

CAD –
Computer Aided Design

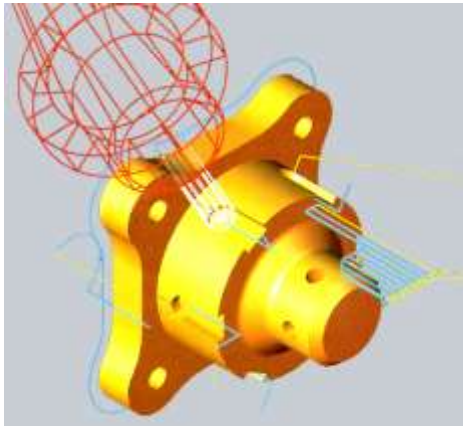


CAE –
Computer Aided Engineering





CAM –
Computer Aided Manufacturing



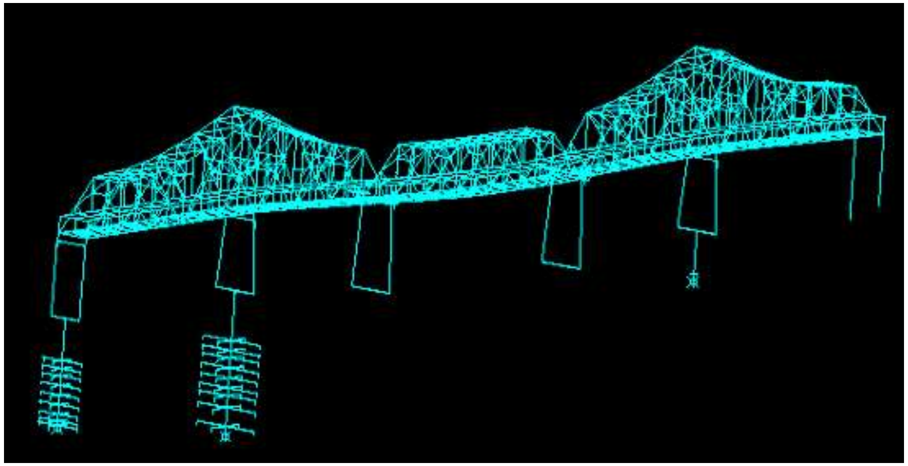
Role of simulation in Engineering: CAD example

Boeing 777

- First jetliner that is 100% digitally designed using 3D solid technology
- Throughout the design process, the airplane was "preassembled" on the computer, eliminating the need for a costly, full-scale mock-up (experimental model)
- $4 billion in CAD infrastructure for design

CAD system: CATIA
CAE system: ELFINI
both by Dassault Systemes (France)

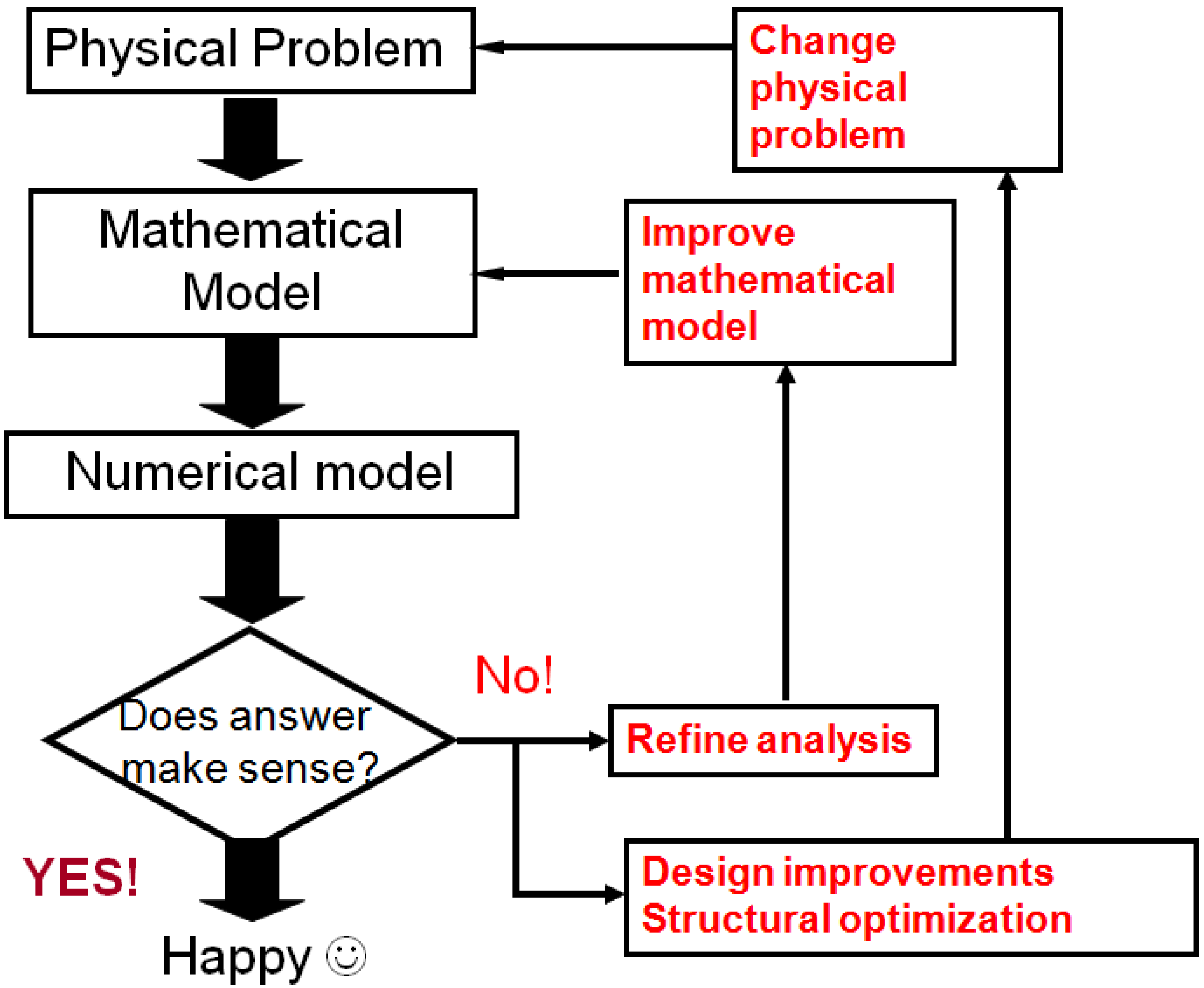
Role of simulation in Engineering: CAE example



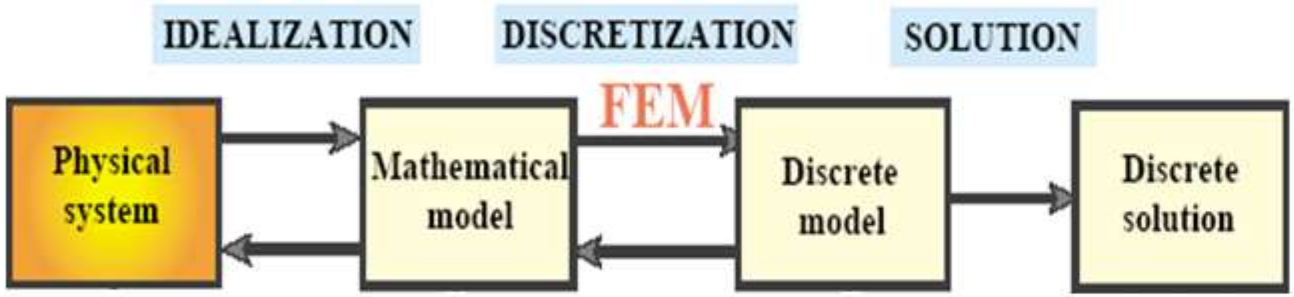
San Francisco Oakland Bay Bridge

- Seismic analysis of the bridge after the 1989 Loma Prieta earthquake
- Finite Element model of a section of the bridge subjected to an earthquake load
- **CAE system: ADINA** (USA, located in MA)

Flow diagram of computer simulation process



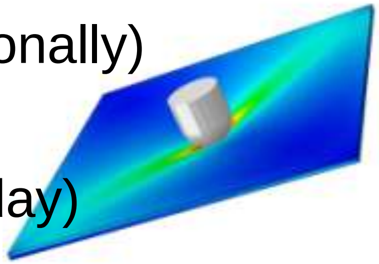
From physical system to mathematical modeling



- **Idealization**: mathematical model, which is an abstraction of physical reality. It is governed by partial differential equations. Modeling can be explicit and implicit.
- **Discretization**: numerical method, for example Finite Element Method.
- **Solution**: linear system solution algorithms, error estimation and convergence analysis.

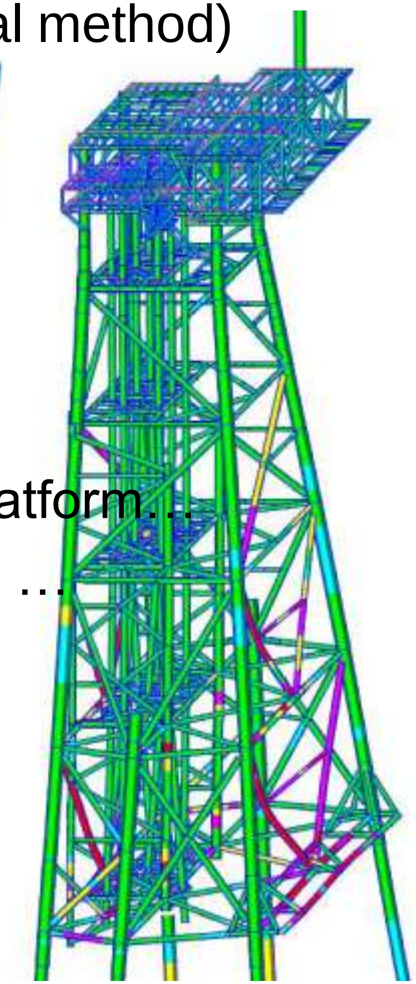
What is Finite Element Analysis (FEA)?

- Simulation by Finite Element Method (as numerical method)
- Branch of Solid Mechanics (traditionally)
- Commonly used method for solving multiphysics problems (today)



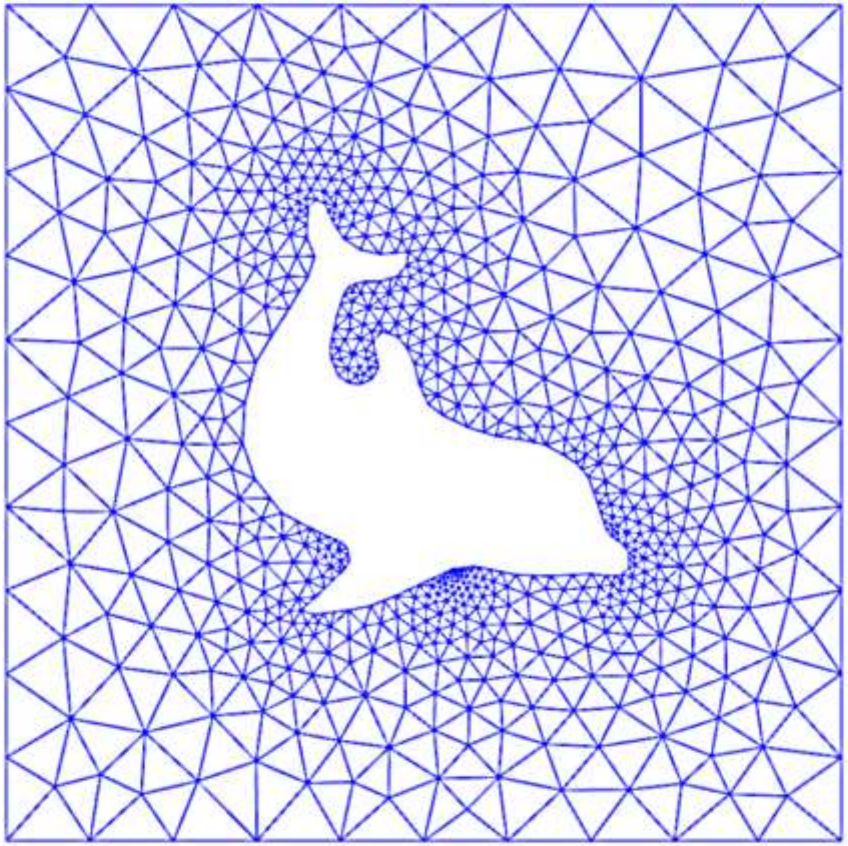
FEA can be used in

- Structural analysis: a cantilever, a bridge, an oil platform…
- Solid mechanics: a gear, a automotive power train …
- Dynamics: earthquake, bullet impact…
- Thermal analysis: thermoplastic polymers
- Electrical analysis: electrical signal propagation…
- Biomaterials: human organs and tissues…
- and many other areas



Examples of FEA application areas

- Mechanical/Aerospace/Civil/Automotive Engineering
- Structural/Stress analyses (static/transient, linear/nonlinear)
- Modal analyses, Harmonic analyses
- Heat transfer
- Fluid flow
- Acoustics
- Aerodynamics
- Soil mechanics, Rock mechanics, Fracture mechanics
- Biomechanics
- Creep and plasticity
- Electro-magnetic fields
- Coupled-field analyses



Main concepts of Finite Element Method

What is Finite Element Method?

There are two interpretations

- Physical Interpretation

  The continuous physical model is divided into finite pieces called elements and laws of nature are applied on the generic element. The results are then recombined to represent the continuum.

- Mathematical Interpretation

  Numerical method: <u>generalization of the classical variational (Ritz) and weighted-residual (Galerkin, least-squares, etc.) methods.</u>

  Each differetional equation reppresenting the physical system is converted into a variational form, which is approximated by the linear combination of a finite set of trial functions.

Why is Finite Element Method widely used?

Most of the real problems:

- are defined on domains that are geometrically complex
- may have different boundary conditions on different portions of the boundary.

Therefore, it is usually impossible (or difficult) to:

- find a solution analytically (hence, one should use numerical methods),
- generate approximation functions required in the traditional variational methods.

An answer to these problems is a **finite-element approach.**

Historical background: FEM theory

- In 1870s Lord John William Strutt **Rayleigh**, developed a method for predicting natural frequencies of simple structures. It assumed a deformed shape for a structure and then quantified this shape by *minimizing the potential energy* of the structure.
- In 1909 Walter **Ritz** ended this into a method, now known as the *Rayleigh-Ritz method*, for approximating the energy functional by known functions with unknown coefficients.
- In 1915 **Galerkin** published an article, in which he put forward an idea of an approximate method for differential equations, in particular boundary value problems. He had applied his method to a big number of pivot and plate analysis problems.
- Some time before I.G.**Bubnov** developed a similar approach for the variational problem solution, which he interpreted as a variant of Ritz method algorithm.

Historical background: FEM theory (continued)

- 1940s – in aerospace engineering: idea of *representing continuum structure by* system of *discrete elements*
- In 1941 Hrennikoff discretized the continuum domain by framework method, representing elastic body by a system of bars and beams
- In 1943, Richard Courant used *a variational formulation* to approximate PDEs by linear interpolation over triangular element
- For Boeing project, Turner, Clough and others used structural triangular elements in plane stress analyses, they also formulated *stiffness matrix for a triangular element*
- In 1960 Dr. Ray Clough introduced the term "*finite element*"
- 1960s and 70s - developments of J.H. Argyris (University of Stuttgart), R.W. Clough (UC Berkeley), O.C. Zienkiewicz (University of Swanse)a, and Richard Gallagher (Cornell University).

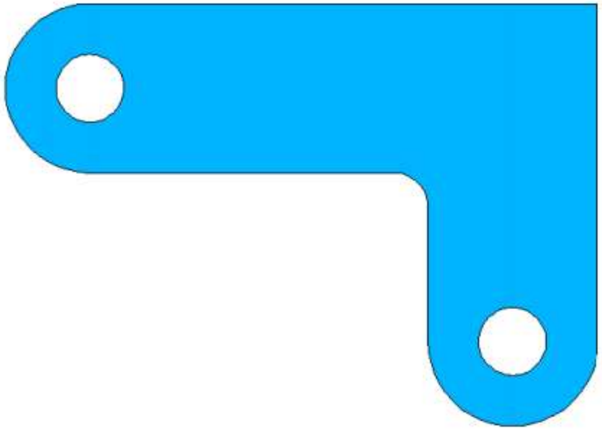
Historical background: CAD and CAE programs

- In the early 1960s, the MacNeal-Schwendle Corporation (MSC) developed a general purpose FEA code for NASA. When the NASA contract was complete, MSC continued development of its own version called MSC/NASTRAN
- Around the time MSC/NASTRAN was released, ANSYS, MARC, and SAP were introduced.
- By the 1970s, Computer-aided design, or CAD, was introduced.
- In the 1980s,CAD progressed from a 2D drafting tool to a 3D surfacing tool. Design engineers began to seriously consider incorporating FEA into the general product design process.
- 1990s – FEM technology has become actually being “hidden” inside CAE packages

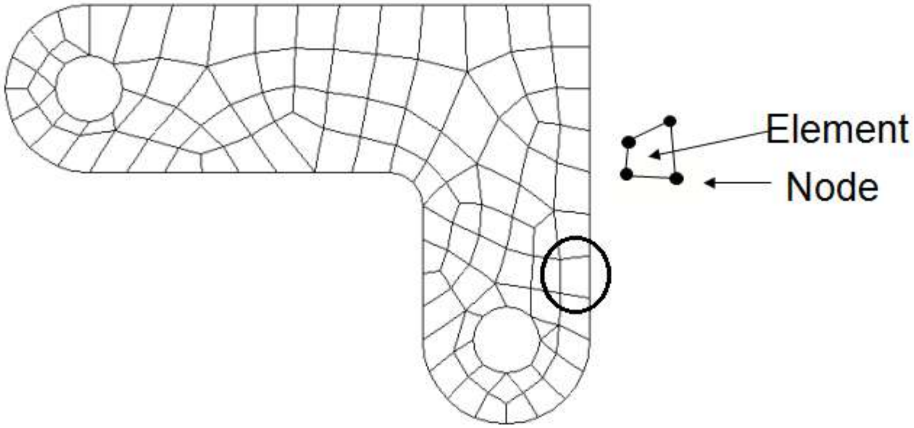
Main idea of Finite Element Method

A given domain can be viewed as an assemblage of simple geometric shapes, called **finite elements, for which it is possible to** systematically generate the **approximation functions** (synonyms: shape functions, interpolation functions).

- Solid model of a bracket

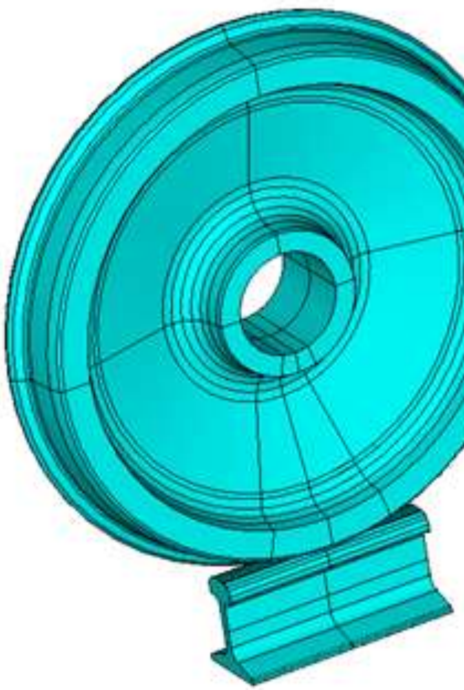


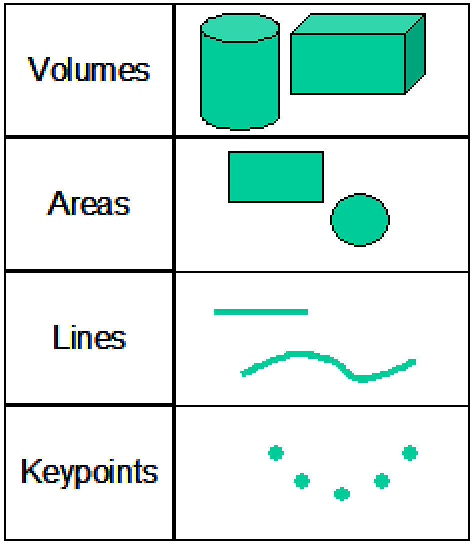
- Finite element model of a bracket



From Solid modeling…

- Solid model of rail-wheel contact

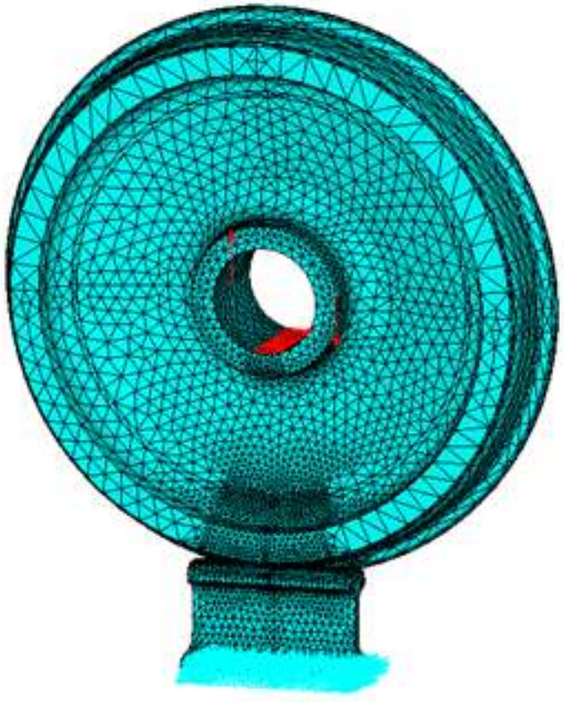




- Solid modeling is a process of creating solid models in CAD system
- Solid model is defined by volumes, areas, lines and keypoints

…To finite element modeling

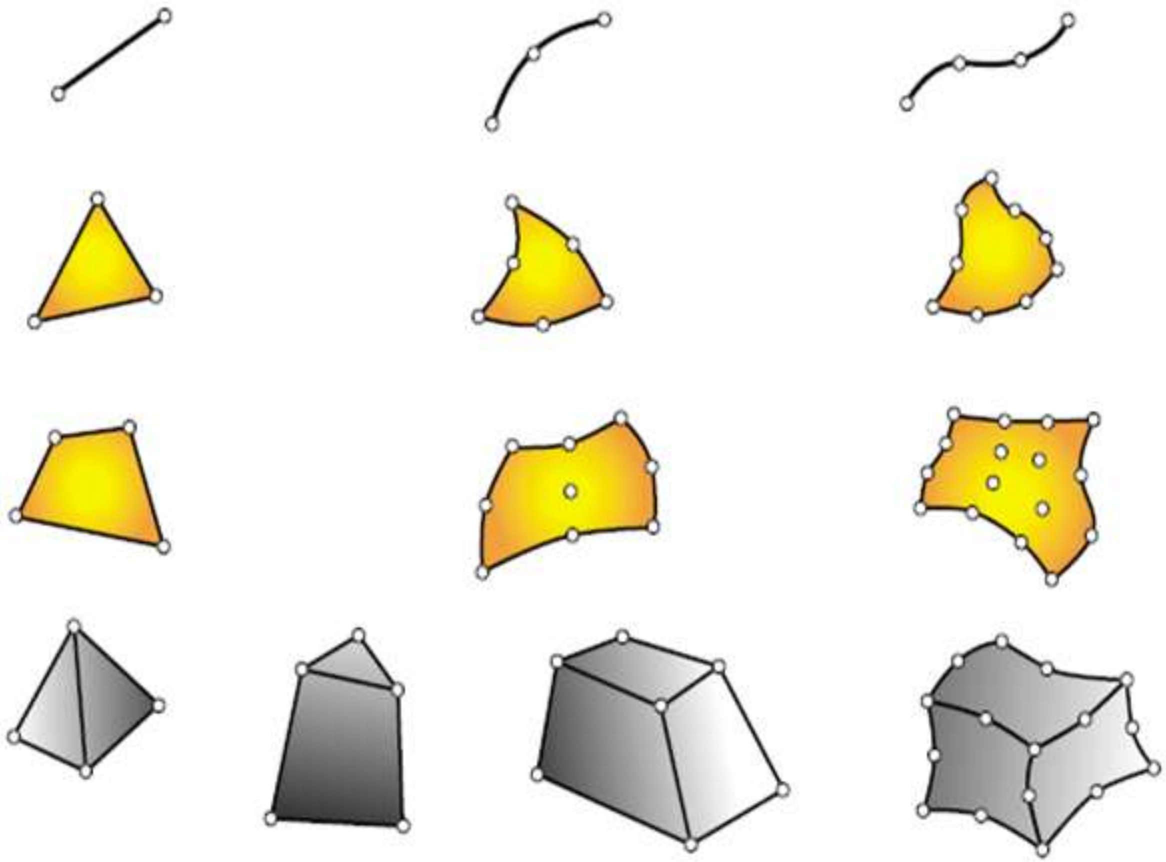
- Finite element model of rail-wheel contact (showing imposed boundary conditions)



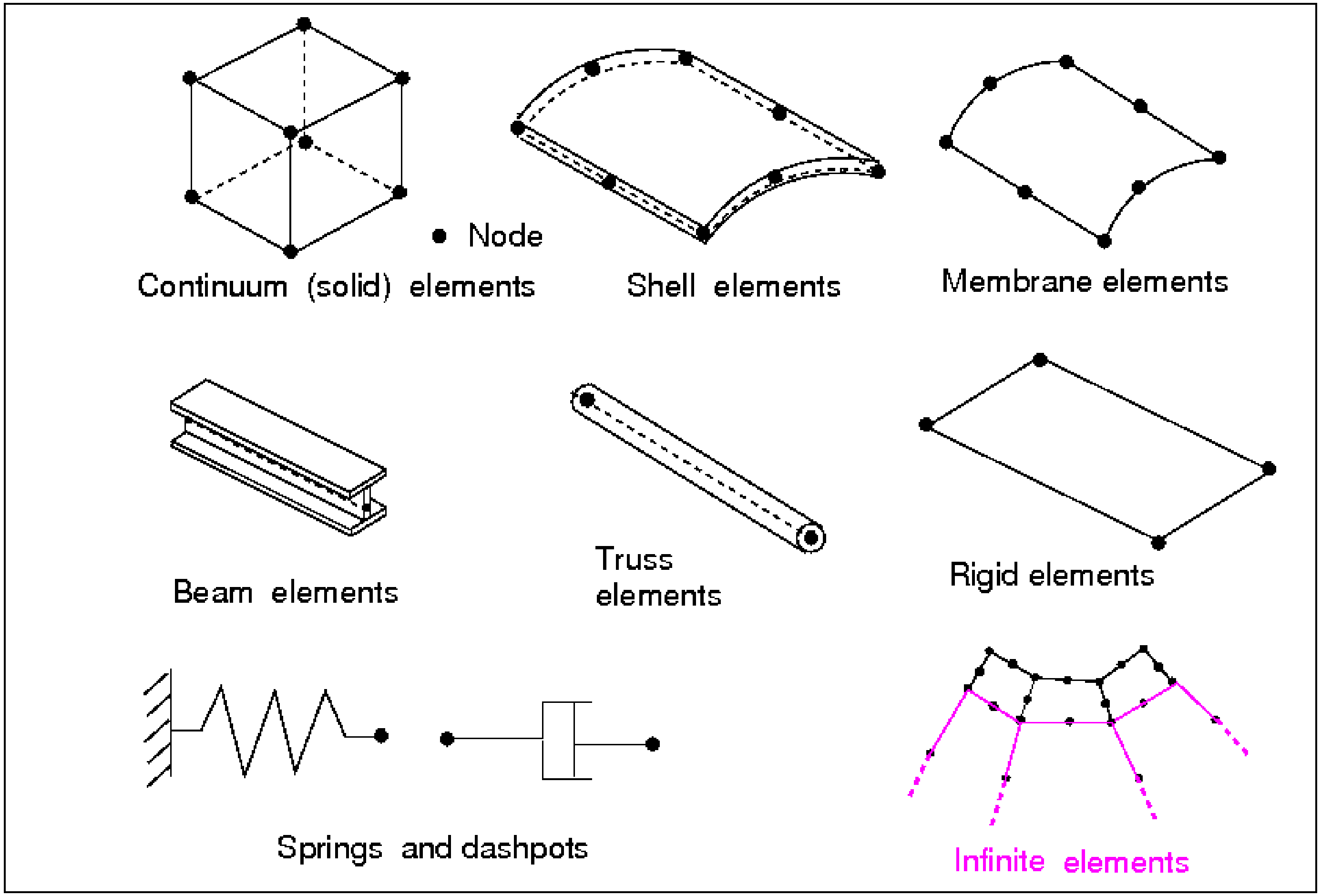
Finite elements at glance

Elements are defined by the following properties:

- Dimensionality
(1D –line, 2D - plane, 3D - solid)
- Nodal Points
- Geometry (shape)
(triangle, quadrilateral, pyramid, prism, tetrahedral, hexahedral)
- Degrees of Freedom
(temperature, displacements, etc.)
- Nodal Forces



Element types and names



Comparison of FEM with other numerical methods

- **Finite element method (FEM)**
  - The problem is discretized, physical meaning is conserved on elements
  - FEM uses exact operators but approximates the *solution basis functions.* Also, FEM solves a problem on the *interiors* of grid cells (and optionally on the gridpoints as well)
- **Finite difference method (FDM)** - approximates the derivatives in the differential equation using difference equations
  - The solution is discretized, loss of physical meaning
  - FDM approximates an *operator* (e.g., the derivative) and solves a problem on a set of *points* (the grid)

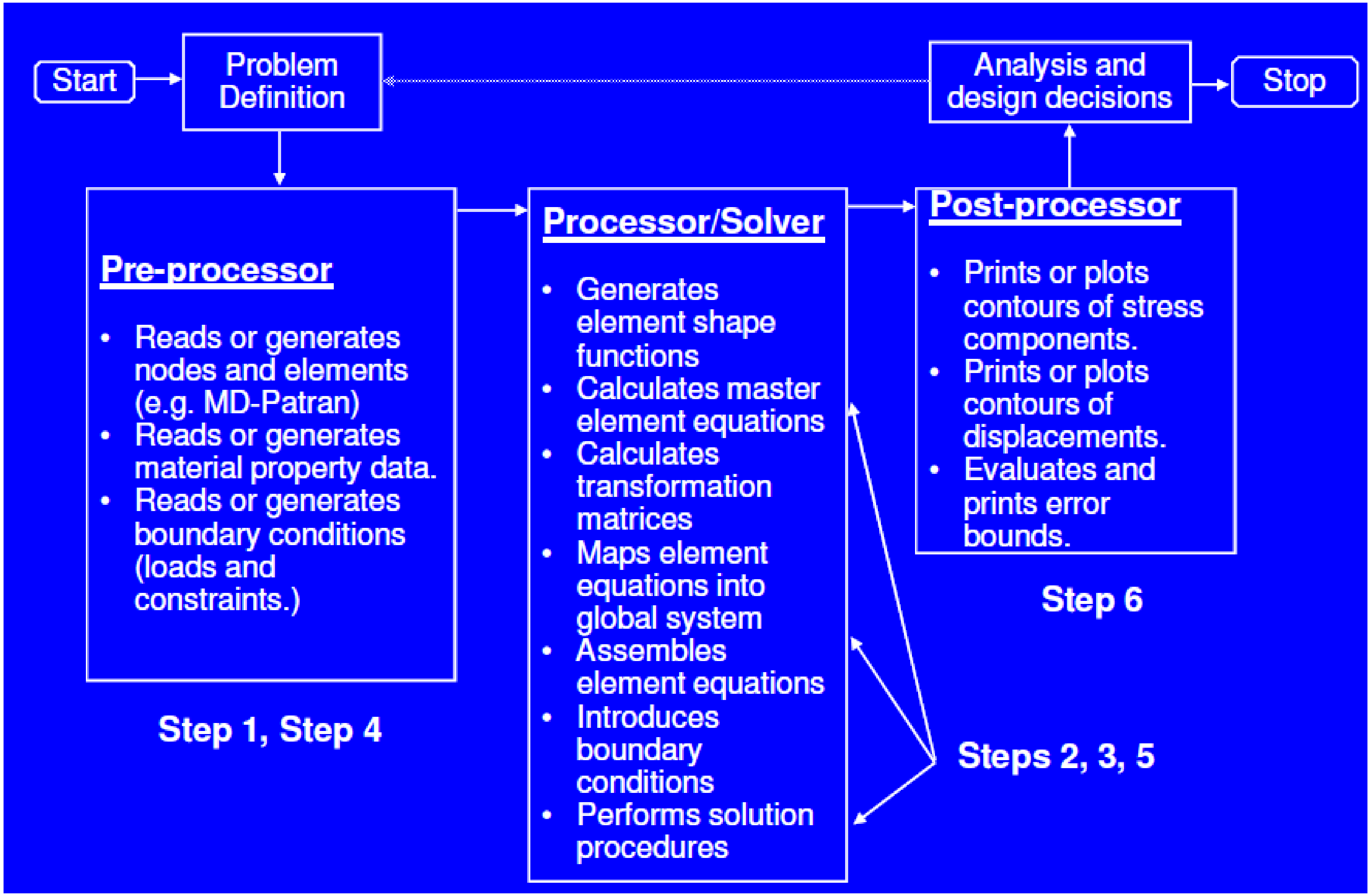
  + Works well for two-dimensional regions with boundaries parallel to the coordinate axes

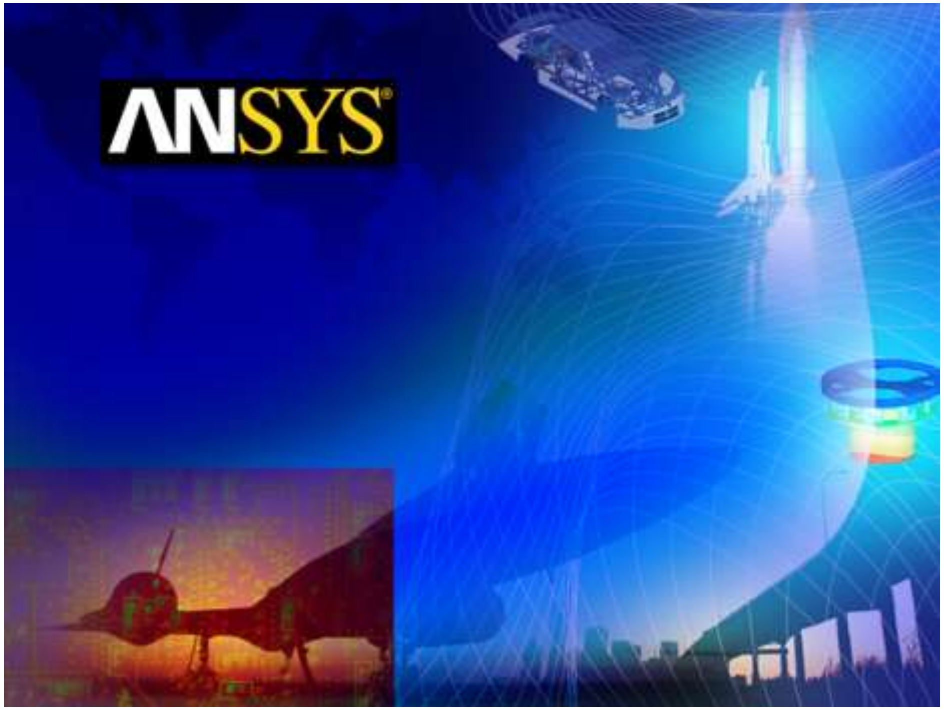
  - Cumbersome when regions have curved boundaries

Steps in Finite Element Analysis

- **Step 1 – Discretization (mesh generation):** The problem domain is discretized into a collection of non-overlapping regions of simple shapes (**elements**) connected to each other through special points (**nodes**)
- **Step 2 - Develop Element Equations (weak formulation)** using the physics of the problem, and typically Galerkin's Method or variational principles.
- **Step 3 - Assembly: The element equations for each element** in the FEM mesh are assembled into a set of global equations that model the properties of the entire system.
- **Step 4 - Application of Boundary Conditions: Solution** cannot be obtained unless boundary conditions are applied. They reflect the known values for certain primary unknowns. Imposing the boundary conditions modifies the global equations.
- **Step 5 - Solve for Primary Unknowns: The modified global** equations are solved for the primary unknowns at the nodes.
- **Step 6 - Calculate Derived Variables** using the nodal values of the primary variables.

Process flow diagram of a typical FEA





Overview of Finite Element Software Packages

Some examples of Finite Element Software

Commercial (CAE programs):

- **ABAQUS FEA** (from Dassault Systèmes)
- **ANSYS** (from ANSYS Inc.)
- **COMSOL Multiphysics**
- ADINA (from ADINA R&D Inc.)
- MSC. Software (NASTRAN, MARC, PATRAN, DYTRAN)
- CosmosWorks
- **FlexPDE** (from PDE Solutions Inc.)

Free limited versions of commercial FEM software:

- ABAQUS SE (Student Edition)
- ANSYS Student
- FlexPDE Student Version

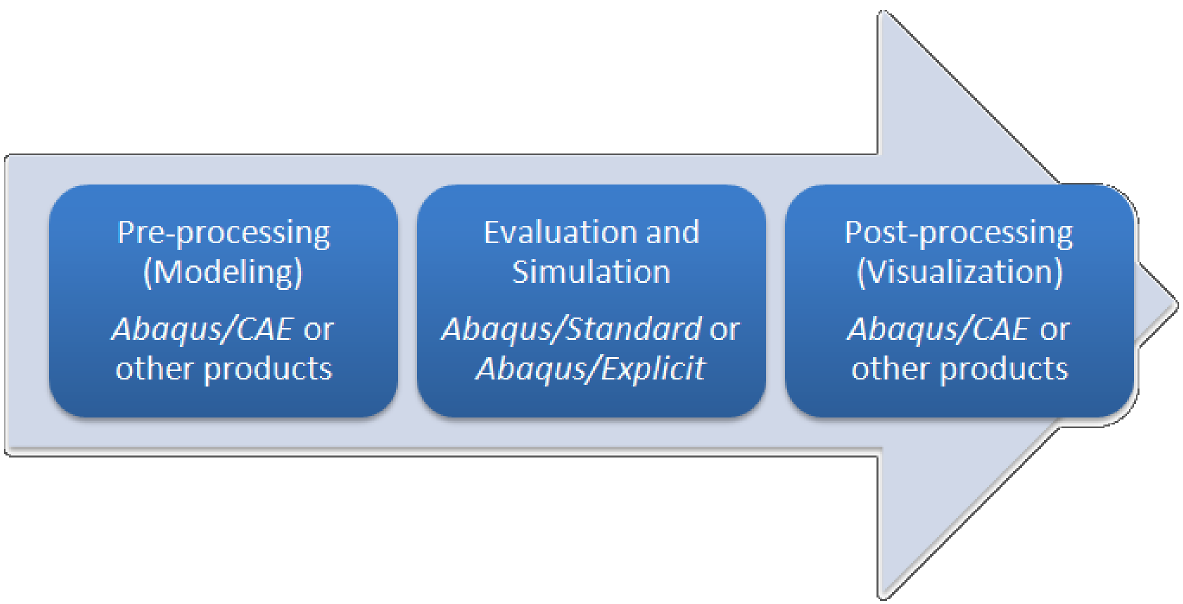
Free: FreeFEM++, ELMER, LISA and many more

ABAQUS FEA

- **SIMULIA Abaqus FEA**
  - *Name and logo from abacus calculation tool*
  - Original author: Dassault Systemes
  - Developer: ABAQUS Inc.
  - Originally released in: 1978
  - Last stable release: ABAQUS 6.14-AP / November 2014
  - Operating system : Microsoft Windows, Linux
  - License: proprietary commercial software
  - Website: www.simulia.com

ABAQUS FEA: products

- Four core products:
  - Abaqus/CAE, or "Complete Abaqus Environment“
  - Abaqus/CFD, a Computational Fluid Dynamics software application
  - Abaqus/Standard, a general-purpose Finite-Element analyzer
  - Abaqus/Explicit, a special-purpose Finite-Element analyzer



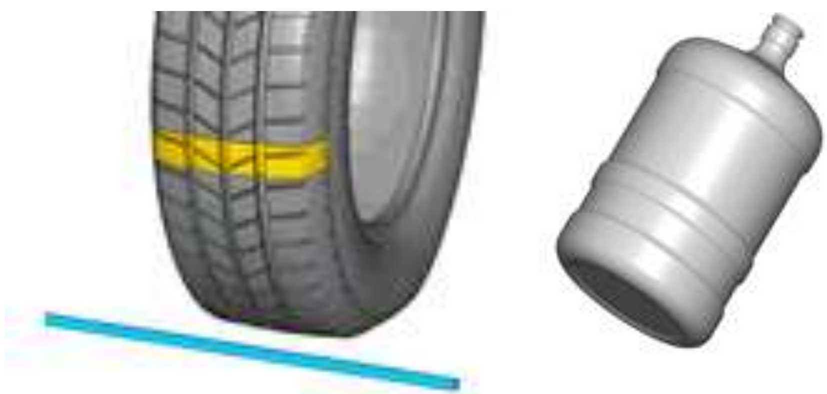
ABAQUS FEA: features

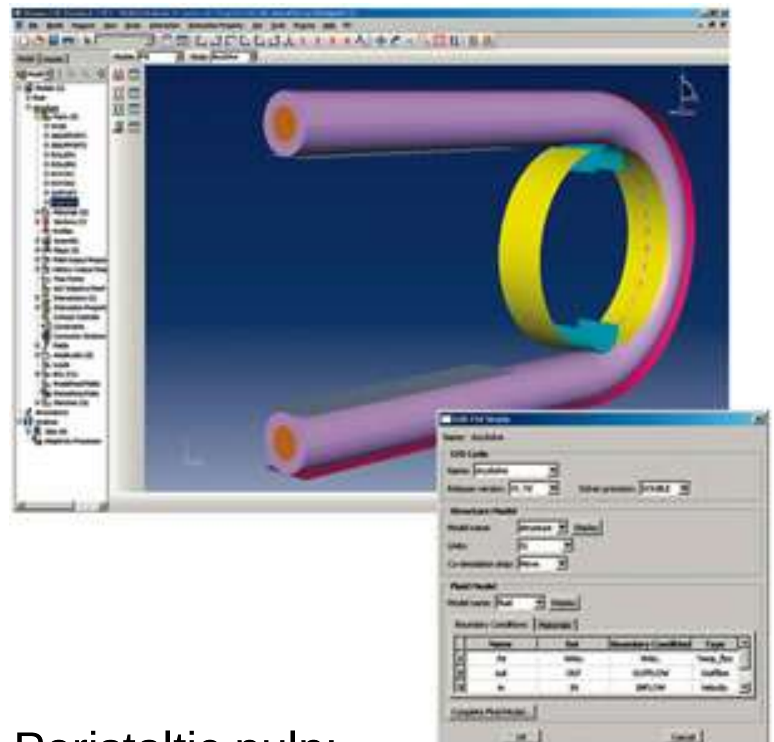
- Abaqus was initially designed to address **highly nonlinear** complex problems
- Good collection of multiphysics capabilities: acoustic-structural, piezoelectric, and structural-pore capabilities, etc.
  - Capabilities for both static and dynamic problems
  - The ability to model very large shape changes in solids, in both two and three dimensions
  - A very extensive element library, including a full set of continuum elements, beam elements, shell and plate elements, among others.
  - A sophisticated capability to model contact between solids
  - An advanced material library, including the usual elastic and elastic – plastic solids; models for foams, concrete, soils, piezoelectric materials, and many others.

ABAQUS FEA: applications

- Automotive, aerospace, and industrial products industries
- Academic and research institutions

Coupled Eulerian-Lagrangian (CEL) analysis in Abaqus Explicit: tire hydroplanning and drop of container with fluid





Peristaltic pulp:
Fluid-structure interaction

ANSYS

- *ANSYS – ANalyses SYStem*
- Developer: ANSYS Inc.
- Company founded by:
  Dr. John A. Swanson as
  Swanson Analysis Systems, Inc. SASI
- Company founded in: 1970, Canonsburg, Pennsylvania
- Last stable release: ANSYS 17.0 / January 2016;
- Operating system: Microsoft Windows, Linux and IBM AIX
- License: proprietary commercial software
- Website: www.ansys.com

ANSYS: products

- Simulation Technology:

  Structural Mechanics, Multiphysics, Fluid Dynamics, Explicit Dynamics, Electromagnetics,

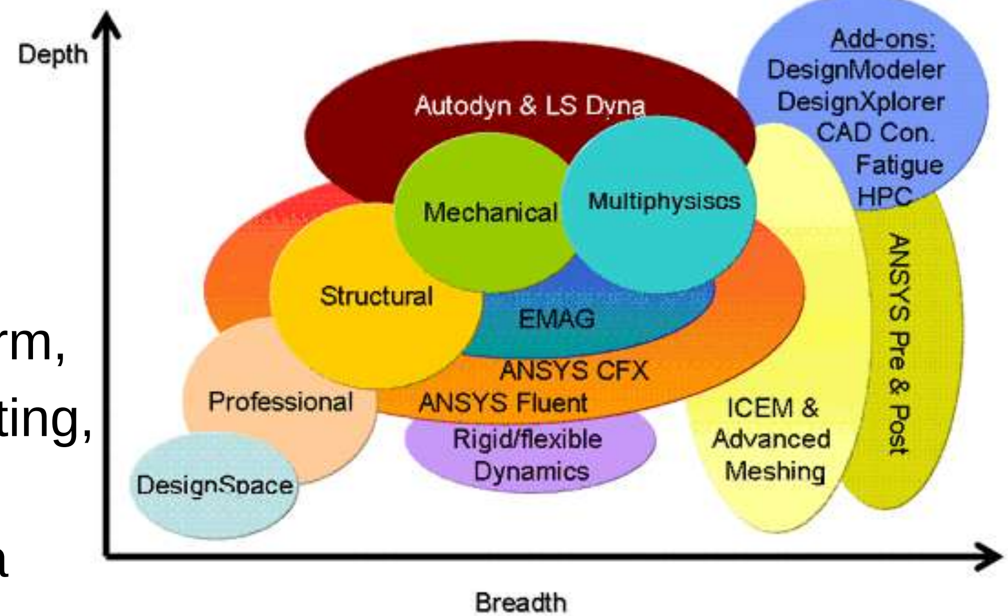
  Hydrodynamics (AQWA).

- Workflow Technology:

  ANSYS Workbench Platform,

  High-Performance Computing, Geometry Interfaces,

  Simulation Process & Data Management.



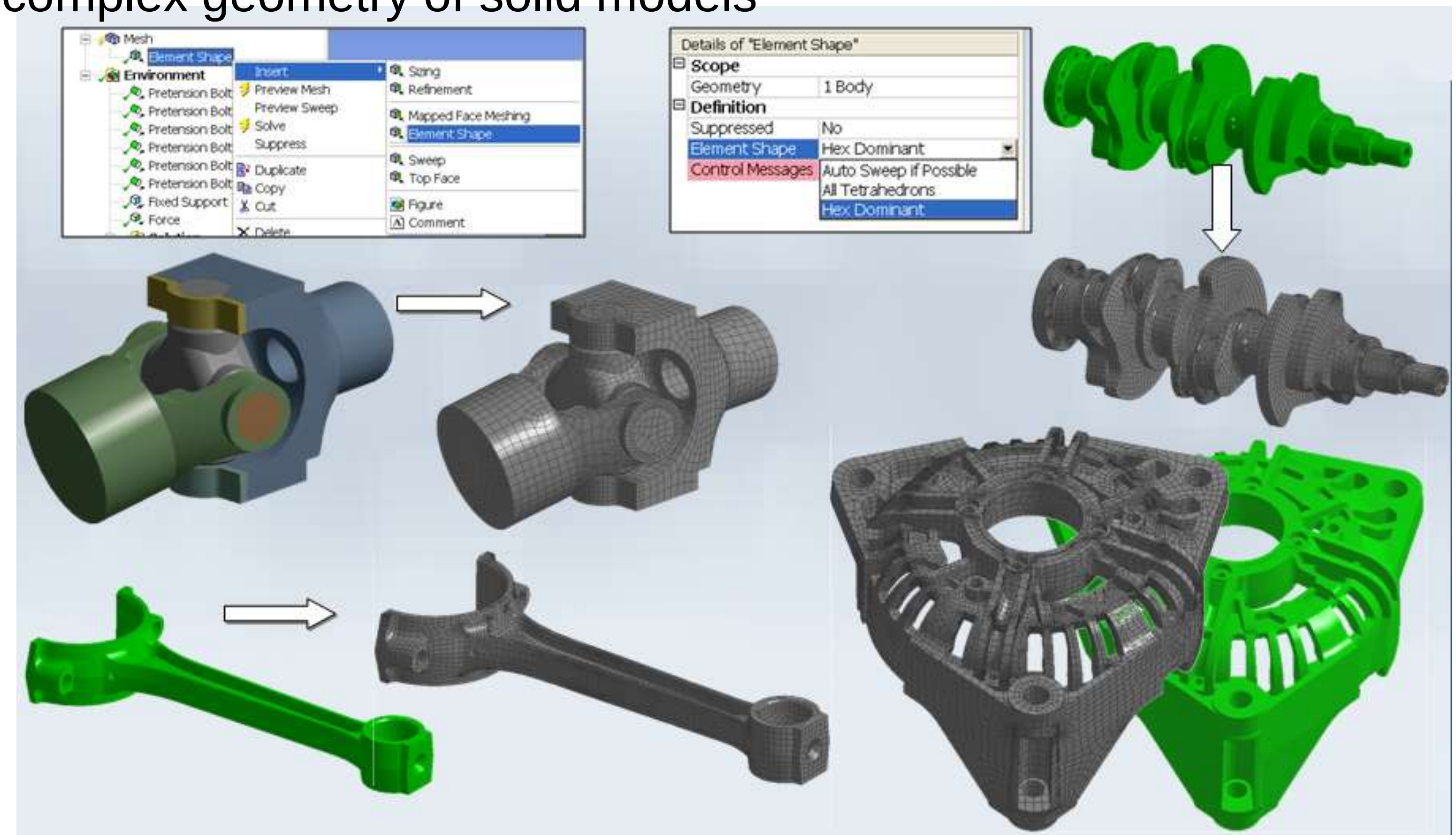
ANSYS: applications

- Largely used in industry:
  - Aerospace & Defense
  - Automotive
  - Construction
  - Consumer Goods
  - Electronics & Semiconductor
  - Energy
  - Healthcare
  - Industrial Equipment & Rotating Machinery
  - Materials & Chemical Processing

- Academic and research institutions

ANSYS: applications (continue)

Automatic meshing: shows how ANSYS can handle complex geometry of solid models



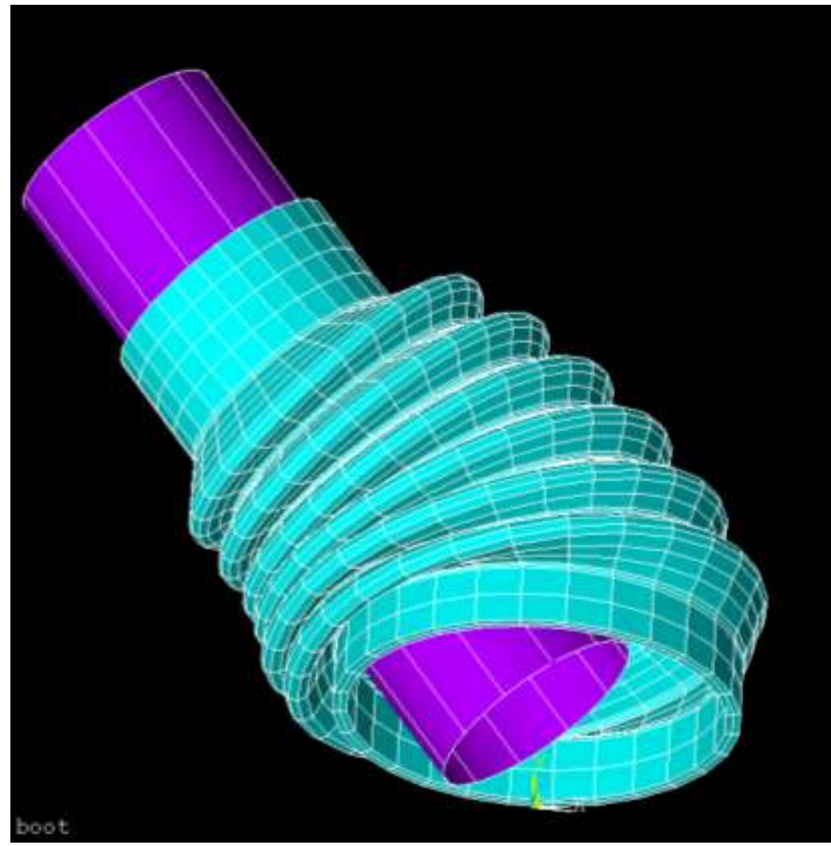
ANSYS: applications (continue)

Rubber sleeve: shows how ANSYS can handle three types of nonlinearities

• *Nonlinear geometry* (large deformations and displacements)

•*Nonlinear material* (rubber)

• *Status change* (contact)

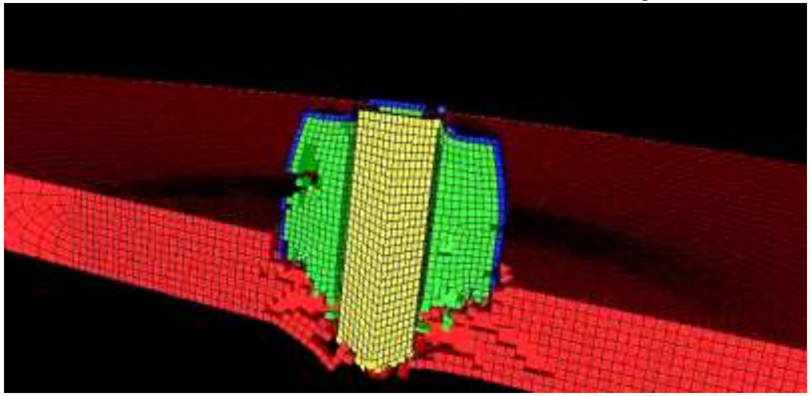


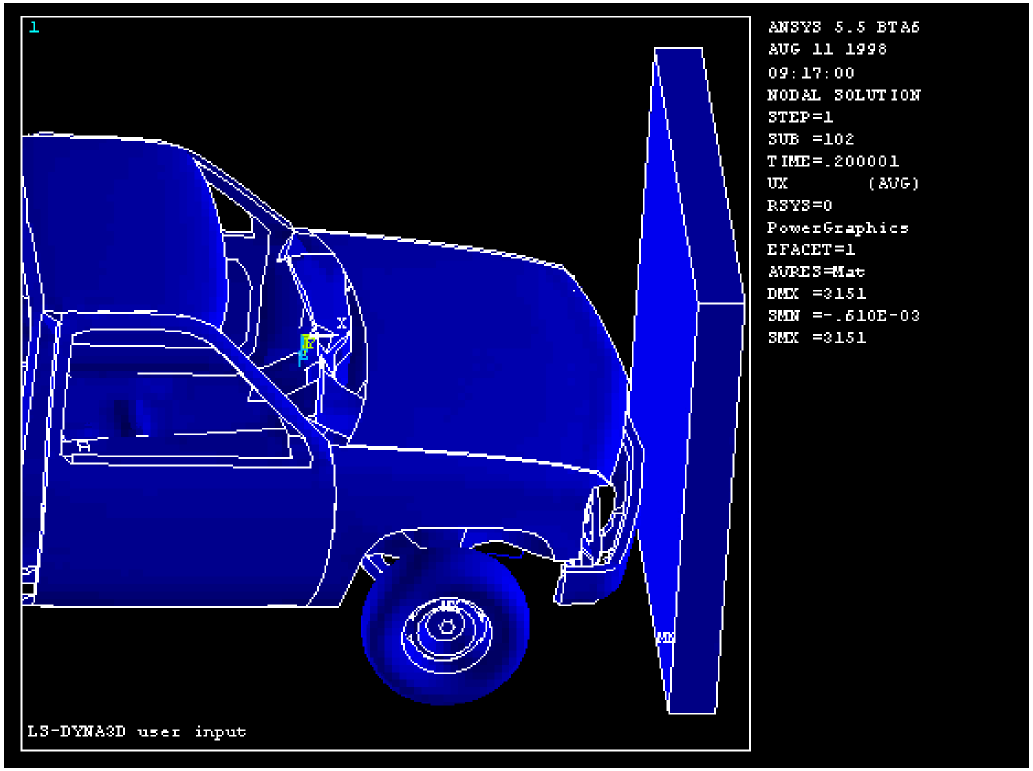
ANSYS: applications (continue)

Car crash test: shows how ANSYS can simulate impact, failure, crash

•LS-DYNA module is used to model very large deformations

Destruction of the bullet tip





ANSYS versus ABAQUS

- Both are excellent FE packages, have similar feature sets related to non-linear modeling, offer good support

ANSYS

- Used more by Industry
- Widely used in Russia (in industry as well as in universities); distributed in Russia by CADFEM
- Easy to use (especially ANSYS Workbench)
- ANSYS APDL programming language to create input file

ABAQUS

- Used more by Academia
- Less known in Russia; Distributor in Russia: Tesis
- Less user friendly, requires a deeper understanding of mechanics and finite elements
- Scripting in Python

COMSOL

- Developer: COMSOL
- Company founded in: 1986, Stockholm, Sweden
- Last stable release: 5.2a / June 2016;
- Operating system: cross-platform
- License: proprietary end-user license agreement
- Website: www.comsol.com
- The PDEs can be entered directly or using the weak form
- Early versions (before 2005) of COMSOL Multiphysics were called FEMLAB

COMSOL: products

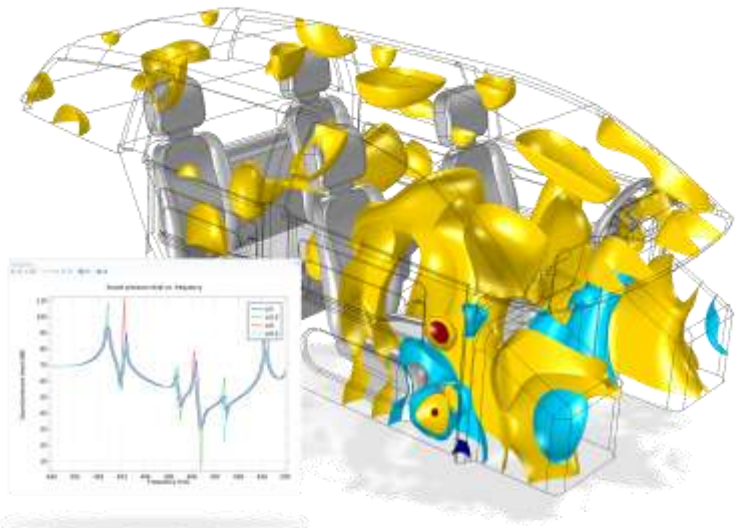
- **COMSOL Multiphysics**
- **COMSOL Server**

- **Add-on Products**
  - Electrical
  - Structural & Acoustics
  - Fluid & Heat
  - Chemical
  - Multipurpose
  - Interfacing (LiveLink for MATLAB and others)

COMSOL: applications

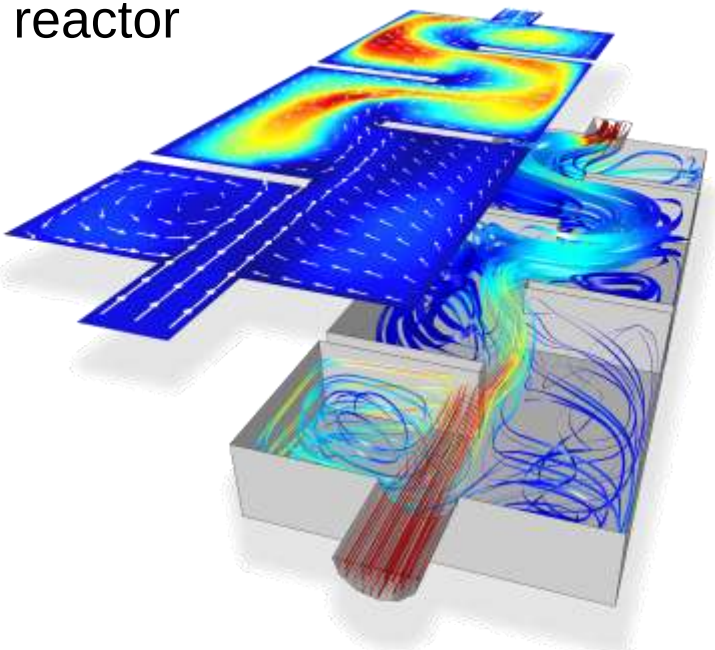
Acoustics Module

Simulation of the acoustics inside a sedan



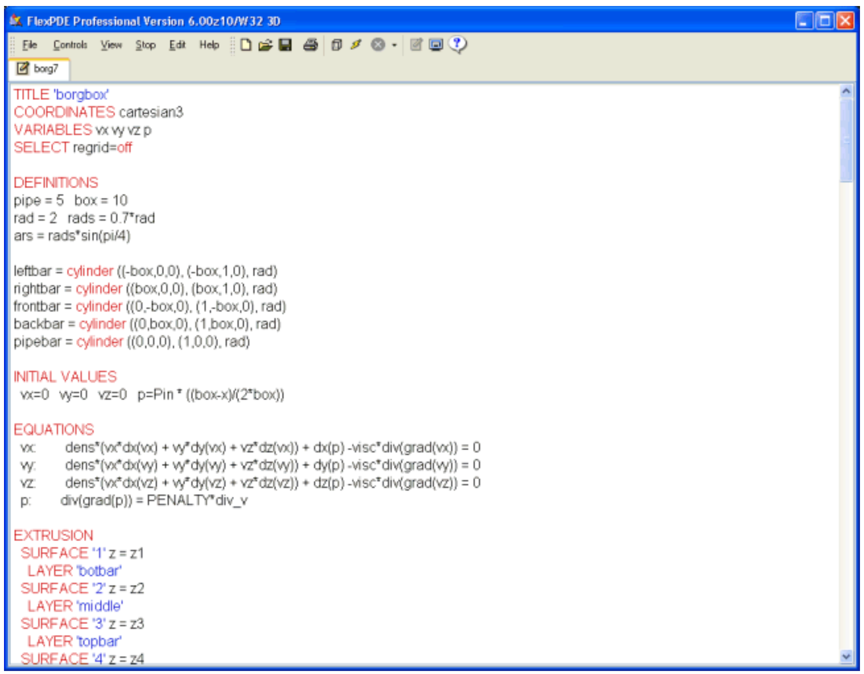
CFD Module

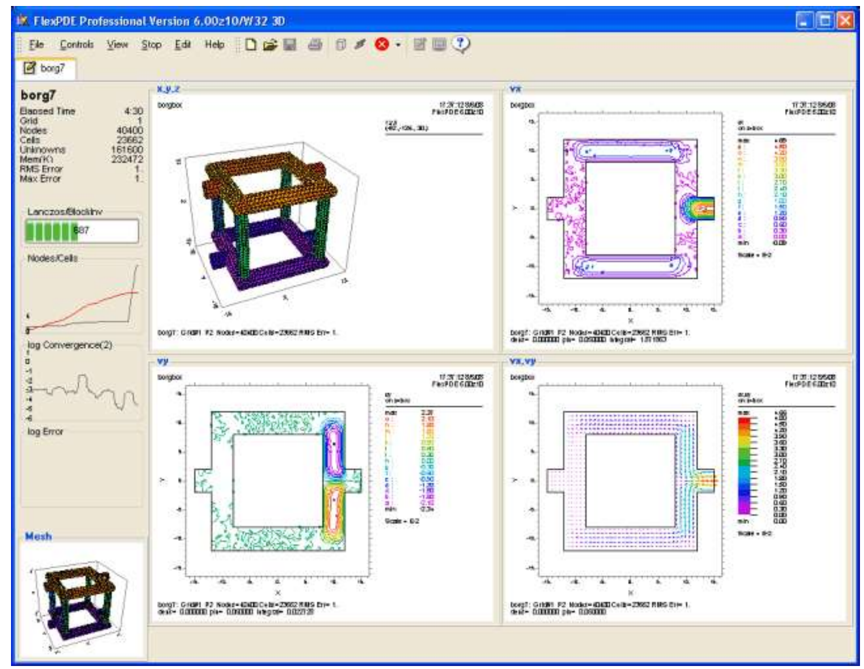
Comparison of the flow field in a 2D approximation with the 3D model of a baffled, turbulent reactor



FlexPDE

• FlexPDE is a general-purpose finite element software for obtaining numerical solutions to partial differential equations in 2 or 3 dimensions. FlexPDE can solve steady-state or time-dependent problems; eigenvalue analysis; and free boundary problems. Last release: FlexPDE 6.39



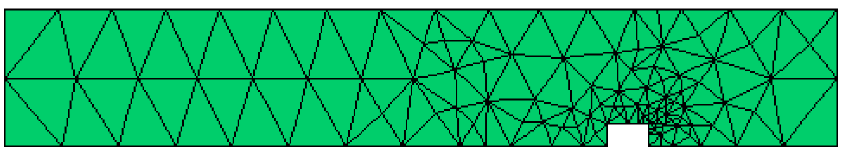


FlexPDE: applications

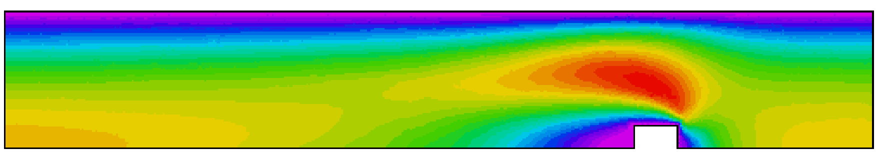
- ❑ Some industrial companies
- ❑ Academic and research institutions

Viscous Flow in a 2D channel

Adaptively refined mesh:

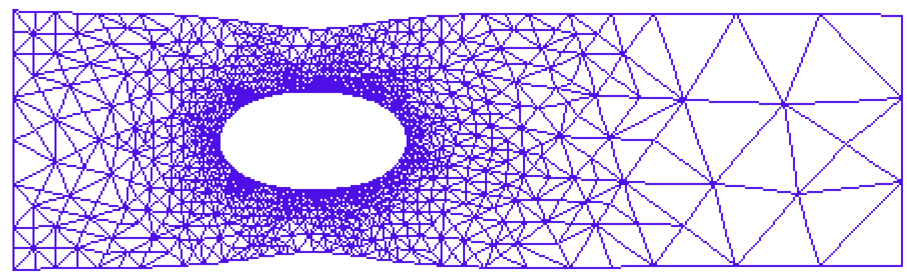


Fluid Speed:

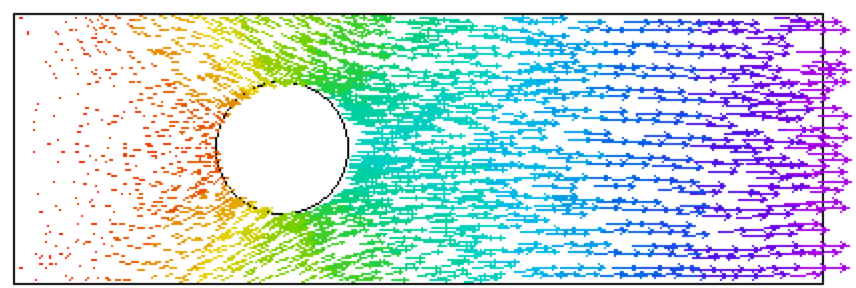


Pressure:

Stress analyses



The final adaptively refined grid



The vector displacement Field

FreeFEM++

• FreeFem++ is a programming language and a software focused in solving partial differential equations by FEM; equations need to be entered in weak form

• written in C++
• developed and maintained by Université Pierre et Marie Curie and Laboratoire Jacques-Louis Lions
• runs on GNU/Linux, Solaris, OS X and MS Windows systems
• free software