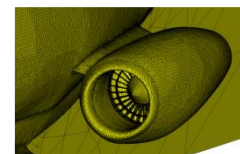
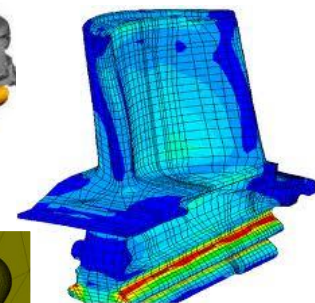
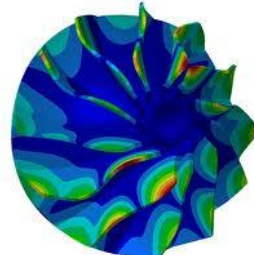
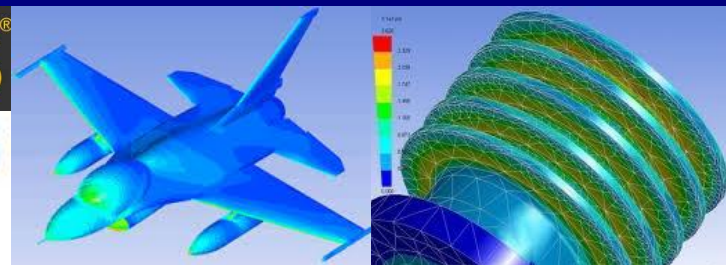




# Finite Element Method in Computer Simulation and Engineering



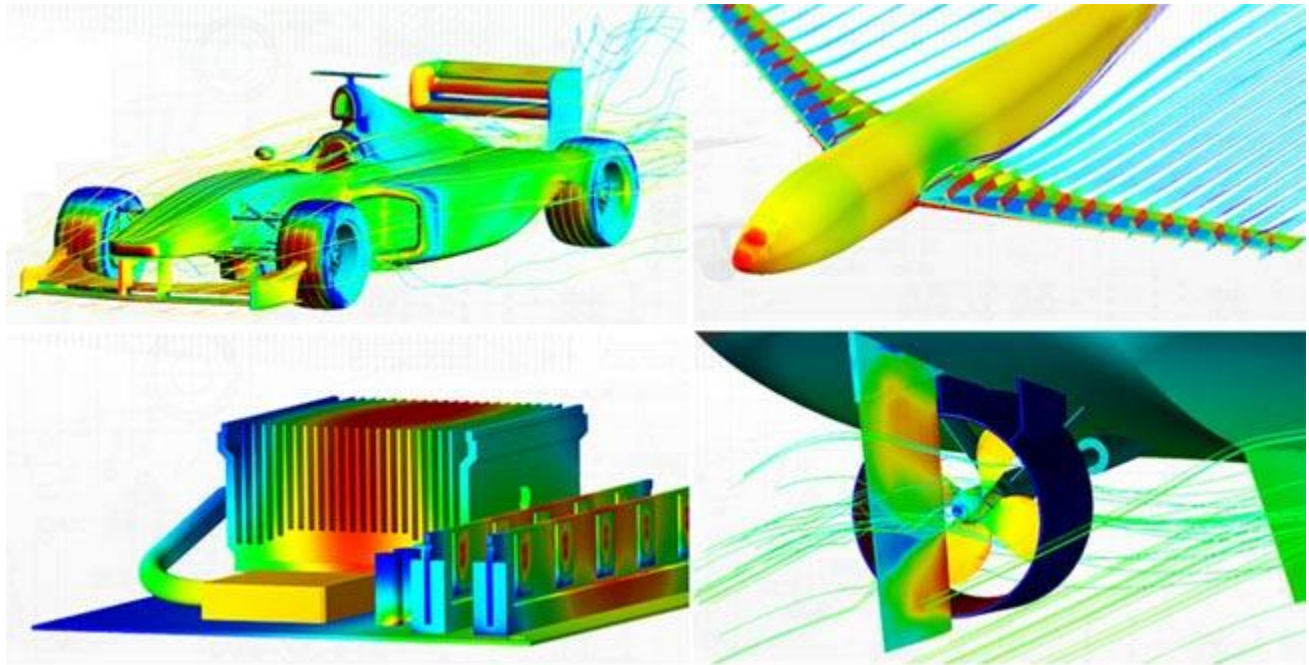
**Dr. Anna A. Nasedkina**

[nasedkina@math.sfedu.ru](mailto:nasedkina@math.sfedu.ru)

Institute of Mathematics,  
Mechanics and Computer Science  
Southern Federal University

# Outline of the lecture

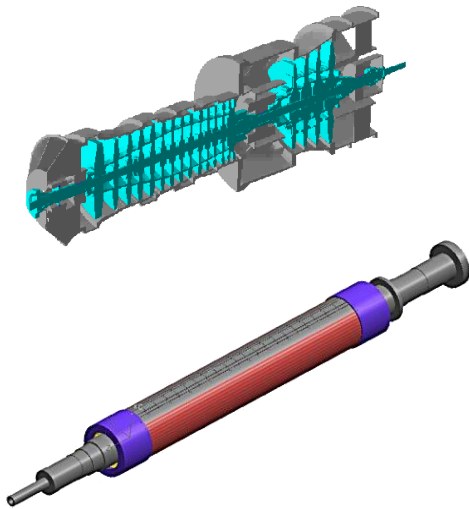
- **Computer Aided Engineering and Finite Element Analysis**
- **Main concepts of Finite Element Method**
- **Overview of Finite Element Software packages**



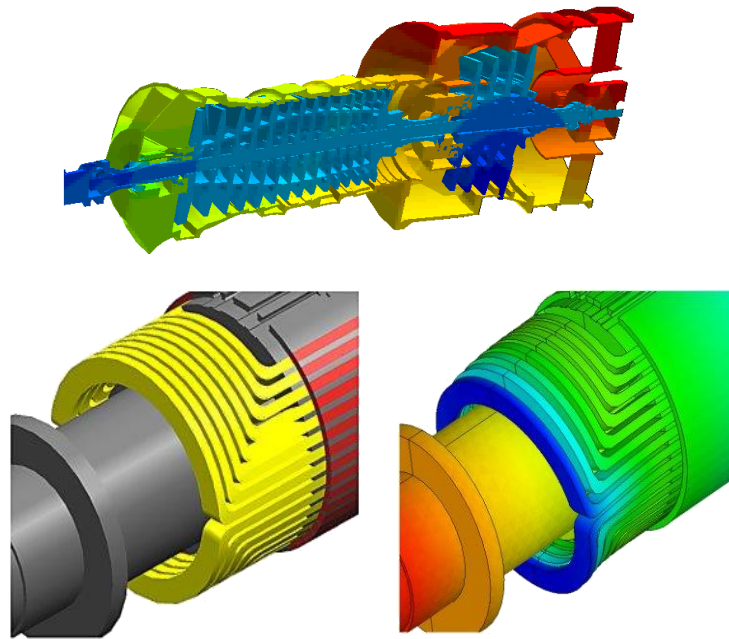
# Computer Aided Engineering and Finite Element Analysis

# CAD/CAE/CAM

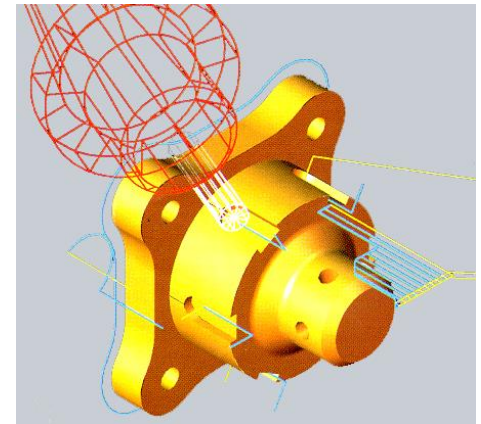
**CAD –  
Computer Aided  
Design**



**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

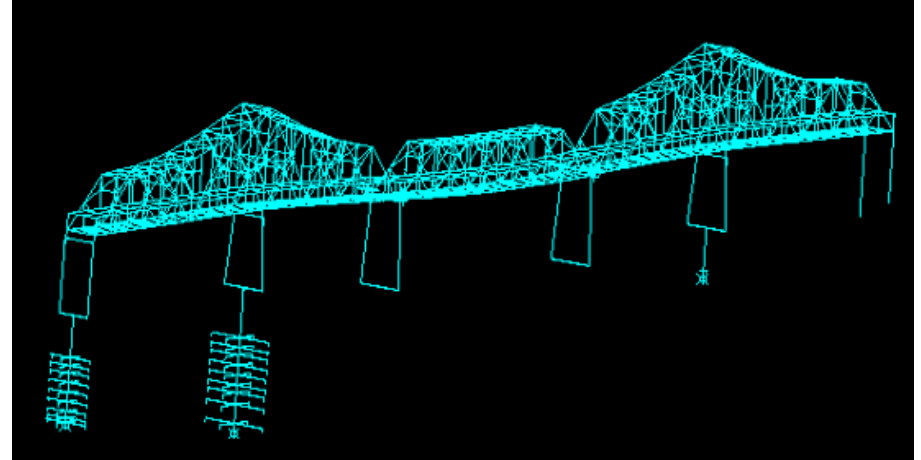


**CAD system: CATIA**

**CAE system: ELFINI**

both by Dassault Systemes (France)

# Role of simulation in Engineering: CAE example

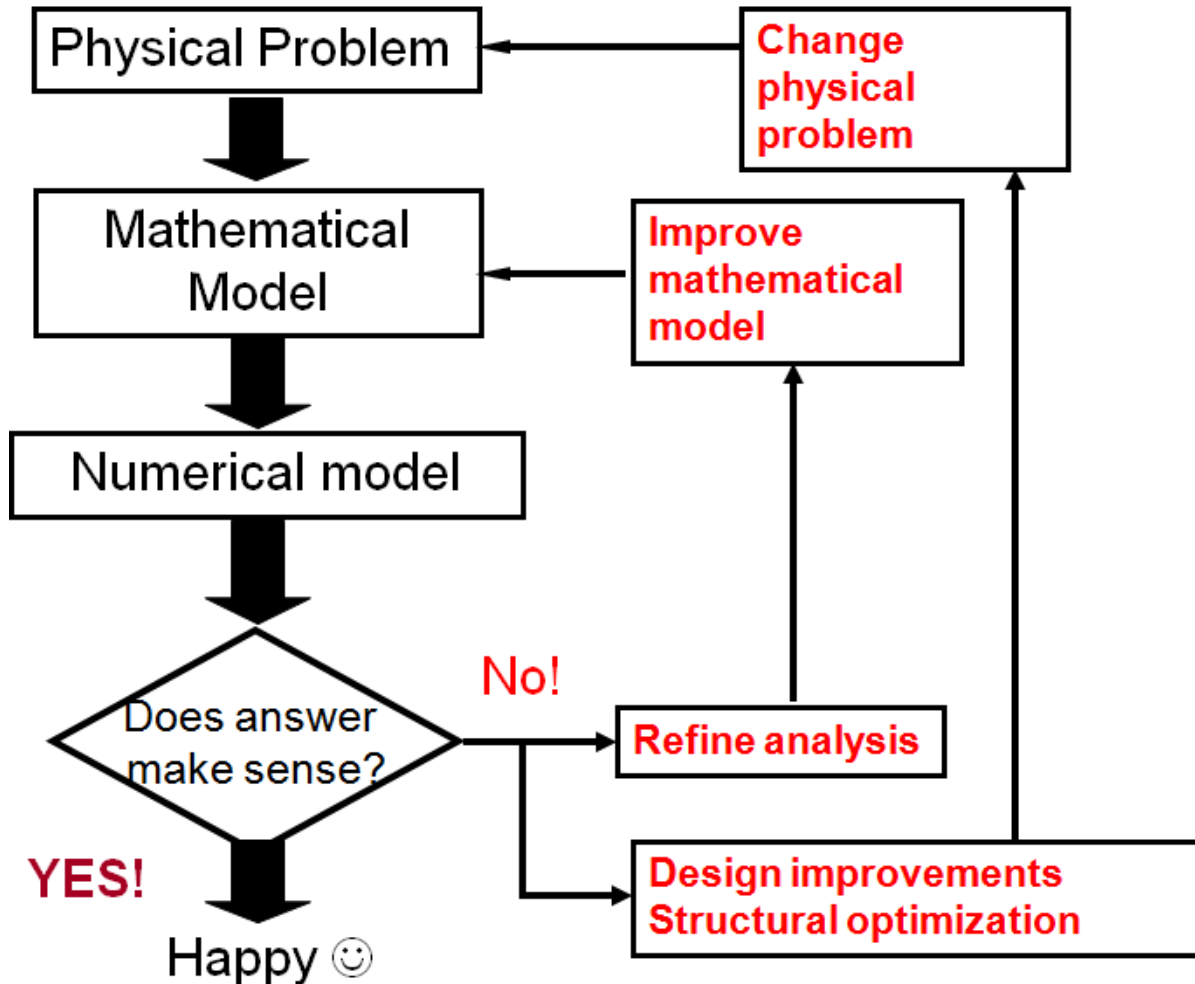


## San Francisco Oakland Bay Bridge

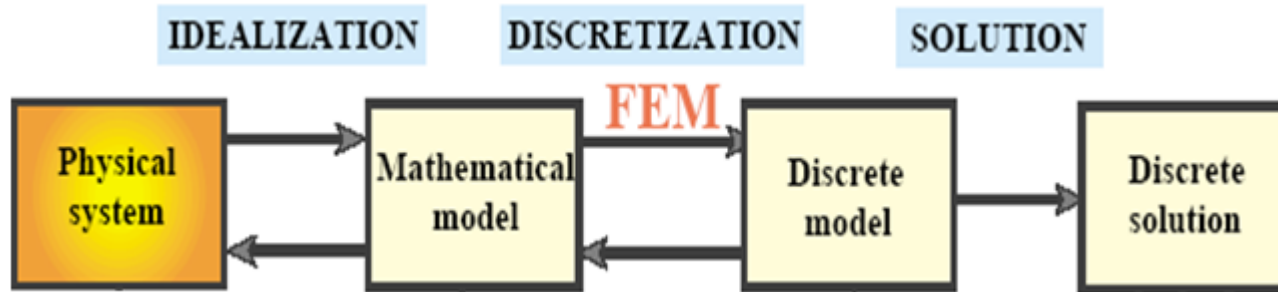
- Seismic analysis of the bridge after the 1989 Loma Prieta earthquake
- Finite Element model of a section of the bridge subjected to an earthquake load
- **CAE system: ADINA** (USA, located in MA)



# Flow diagram of computer simulation process



# From physical system to mathematical modeling

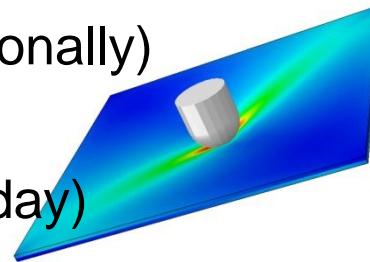


- **Idealization:** mathematical model, which is an abstraction of physical reality. It is governed by partial differential equations. Modeling can be explicit and implicit.
- **Discretization:** numerical method, for example Finite Element Method.
- **Solution:** linear system solution algorithms, error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



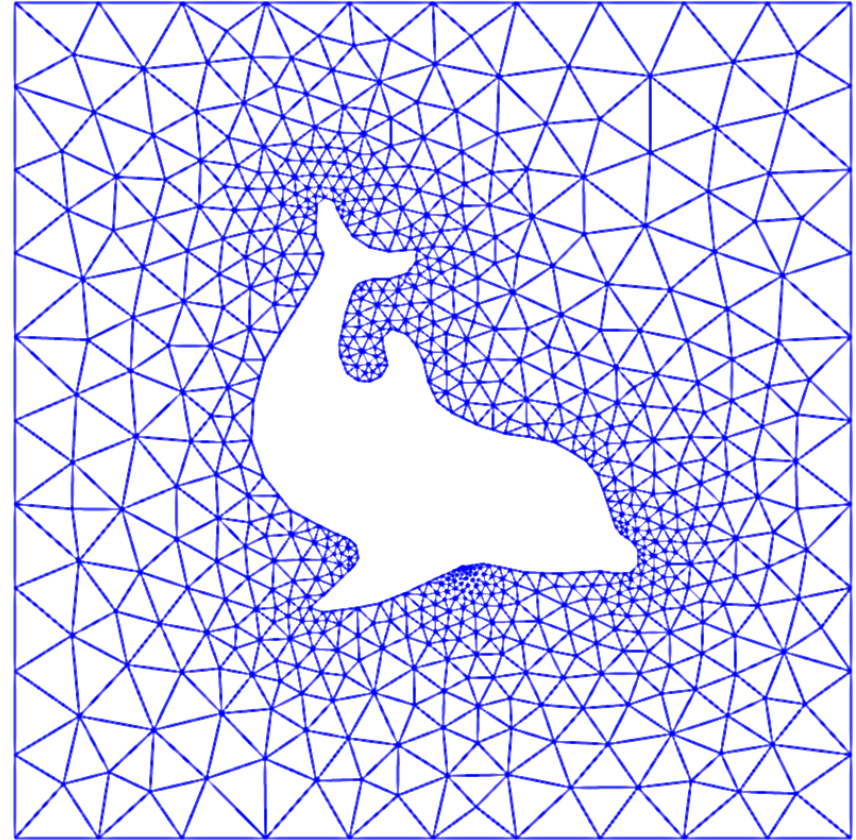
## FEA can be used in

- 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: generalization of the classical variational (Ritz) and weighted-residual (Galerkin, least-squares, etc.) methods.

Each differential equation representing 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 Swansea), and Richard Gallagher (Cornell University).

# Historical background: CAD and CAE programs

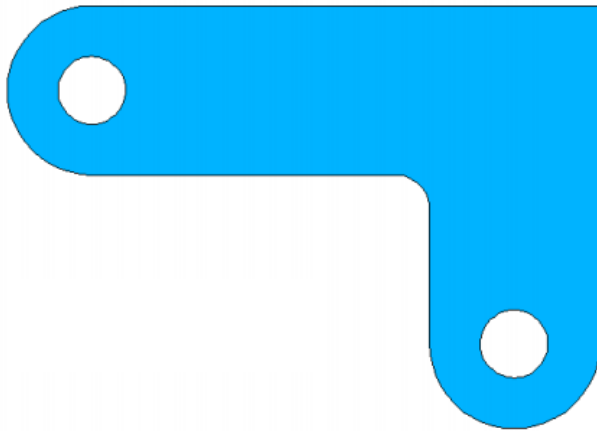
- In the early 1960s, the MacNeal-Schwendle Corporation (MSC) developed a general purpose FEA code for NASA. When the NASA contract was complete, MSC continued development of its own version called MSC/NASTRAN
- Around the time MSC/NASTRAN was released, ANSYS, MARC, and SAP were introduced.
- By the 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



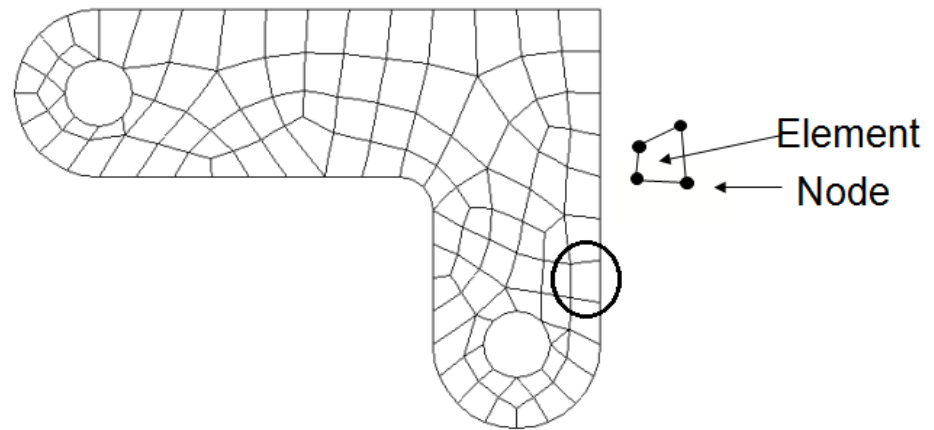
# Main idea of Finite Element Method

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

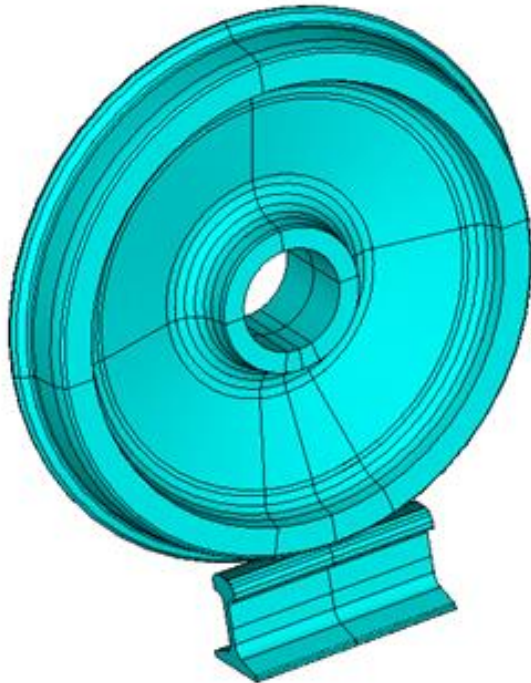


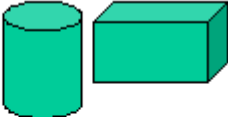



- Finite element model of a bracket



# From Solid modeling...

- Solid model of rail-wheel contact

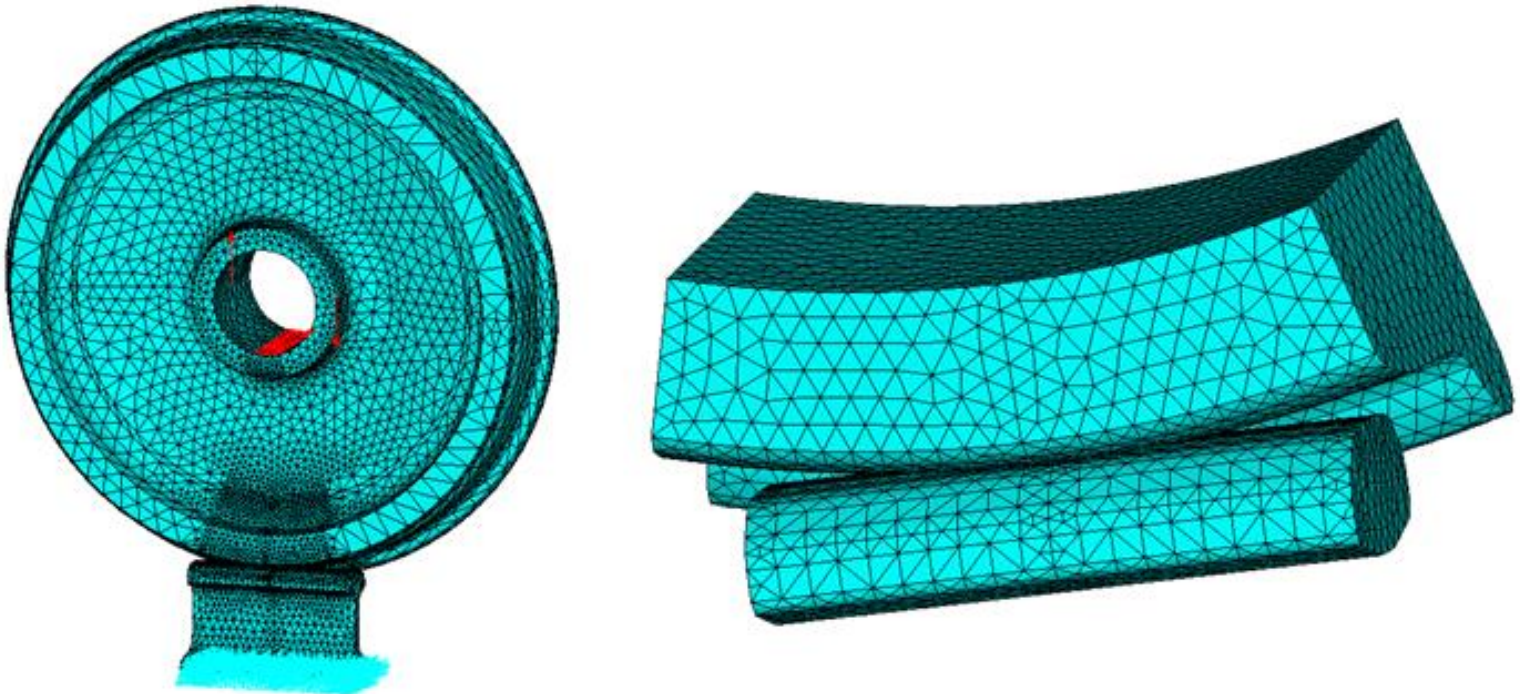


Volumes	
Areas	
Lines	
Keypoints	

- 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

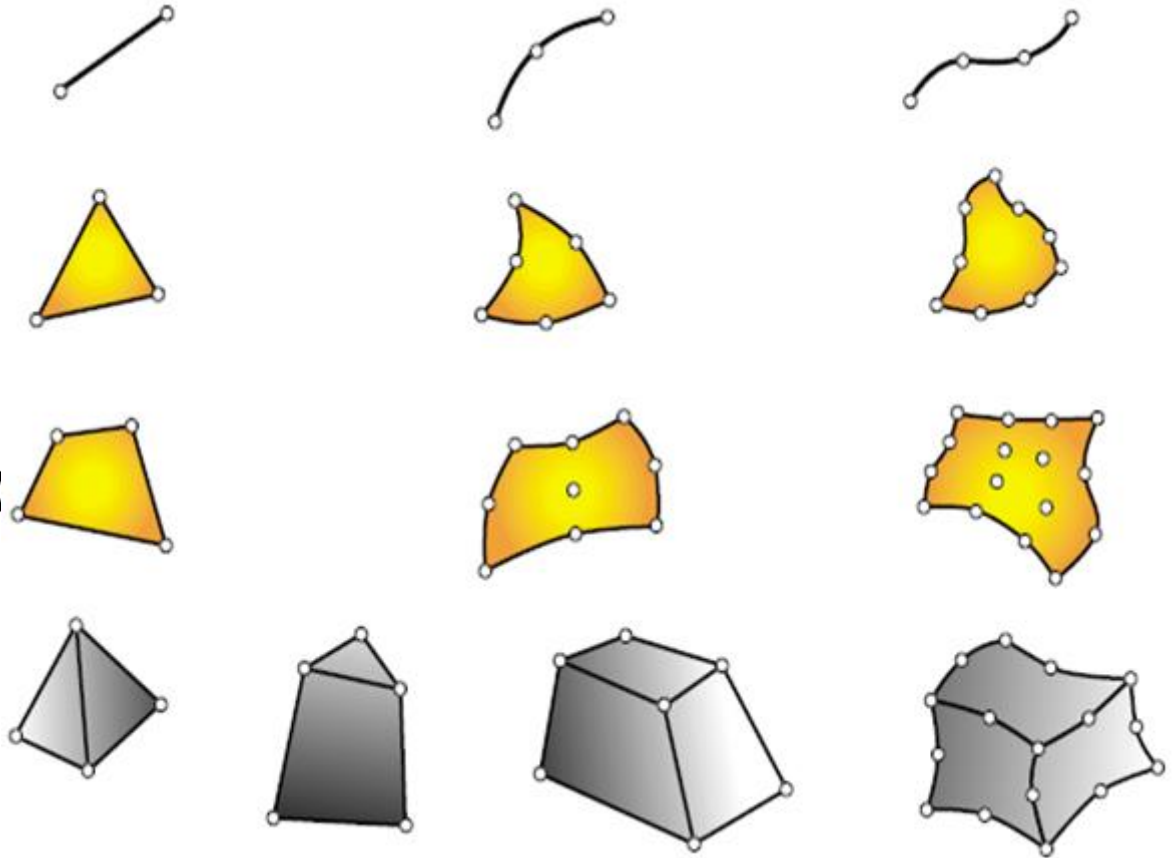
- Geometry (shape)

(triangle, quadrilateral,  
pyramid, prism,  
tetrahedral, hexahedral)

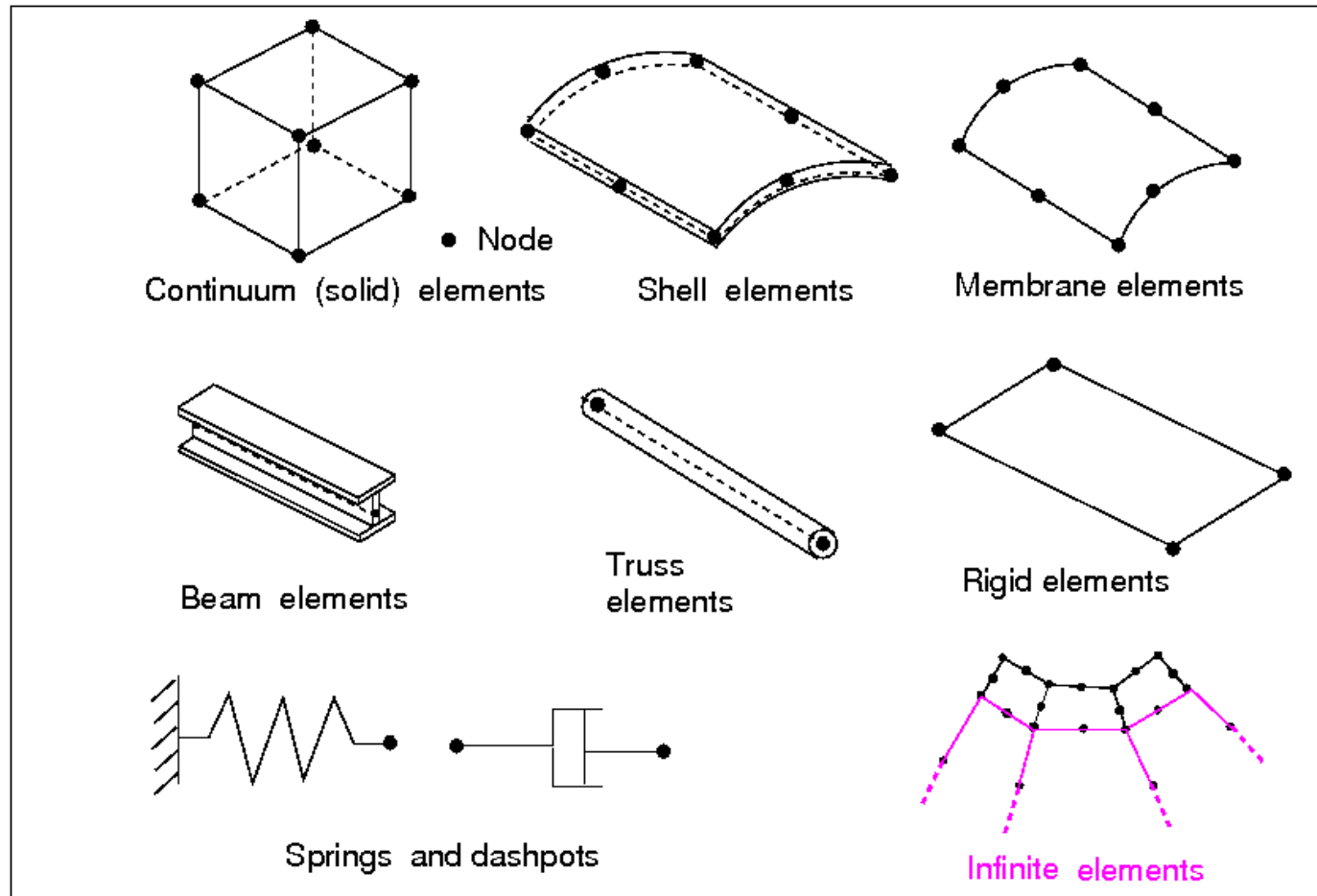
- Degrees of Freedom

(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)

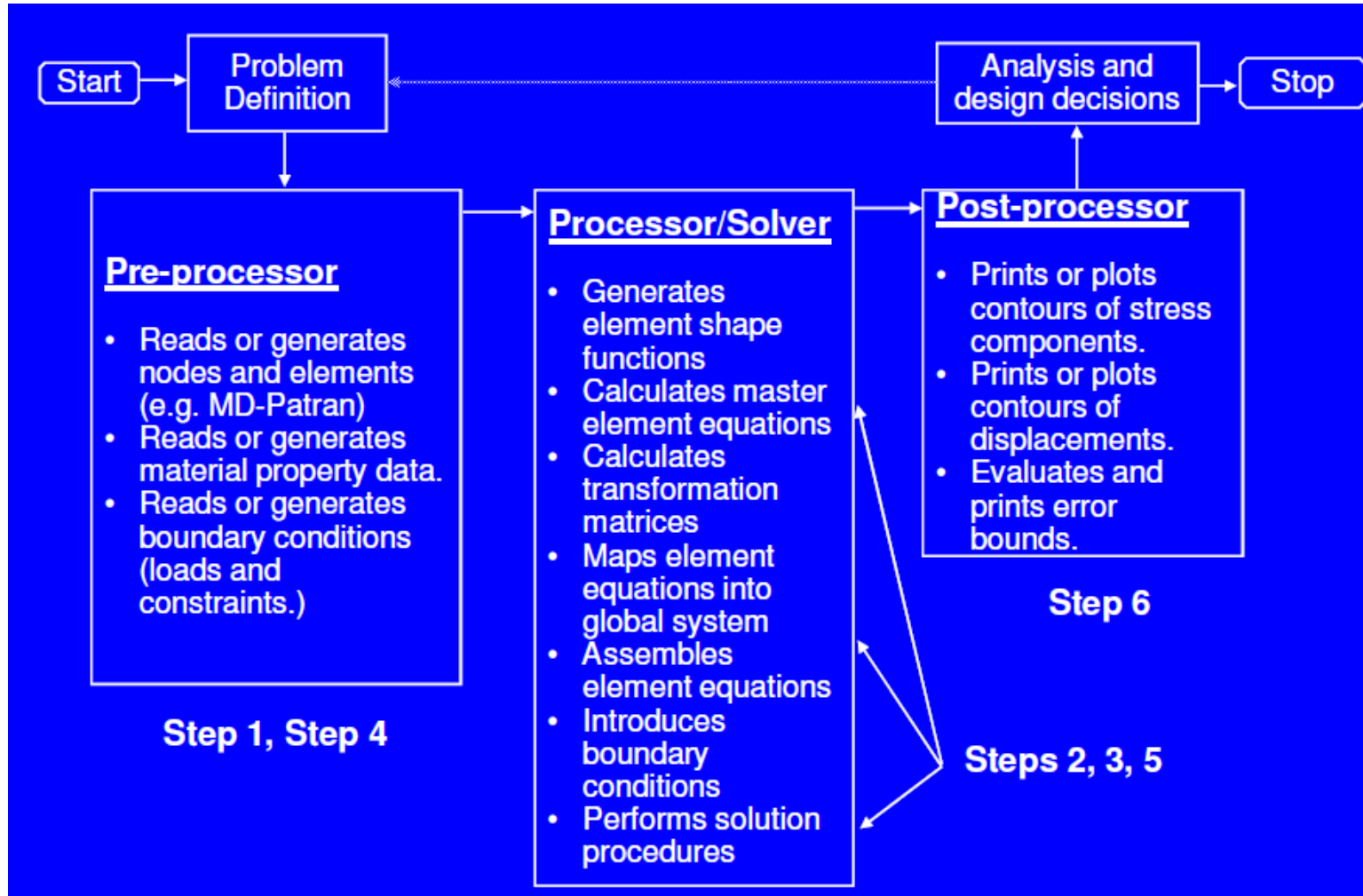
## ■ Finite difference method (FDM) - approximates the derivatives in the differential equation using difference equations

- 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

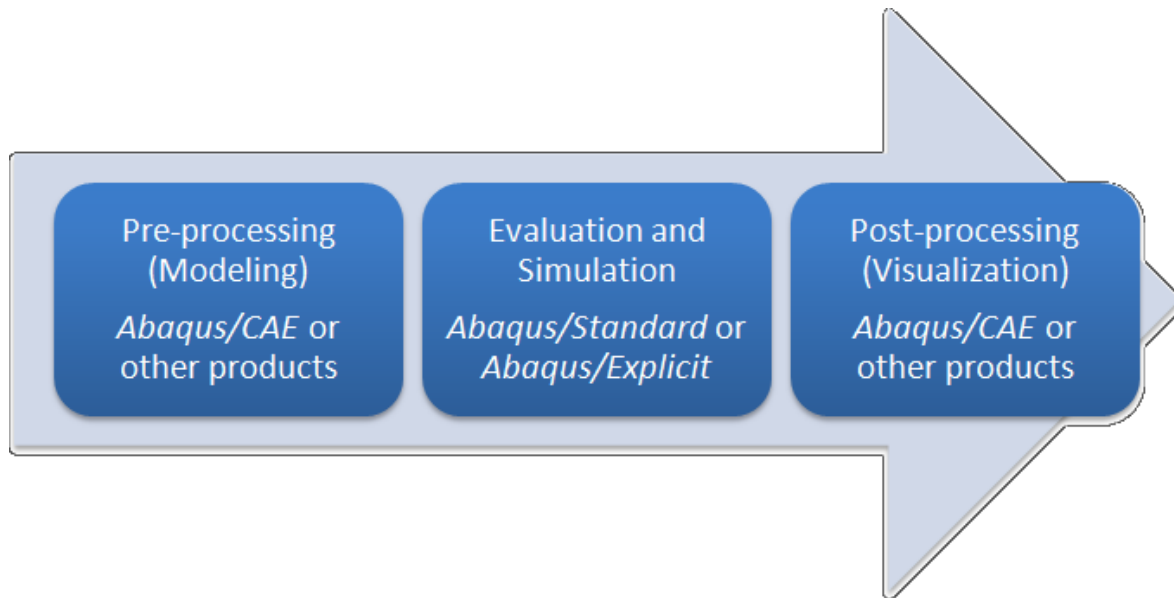
- *Name and logo from abacus calculation tool*
- Original author: Dassault Systemes
- Developer: ABAQUS Inc.
- Originally released in: 1978
- Operating system : Microsoft Windows, Linux
- License: proprietary commercial software
- Website: [www.simulia.com](http://www.simulia.com)



# 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



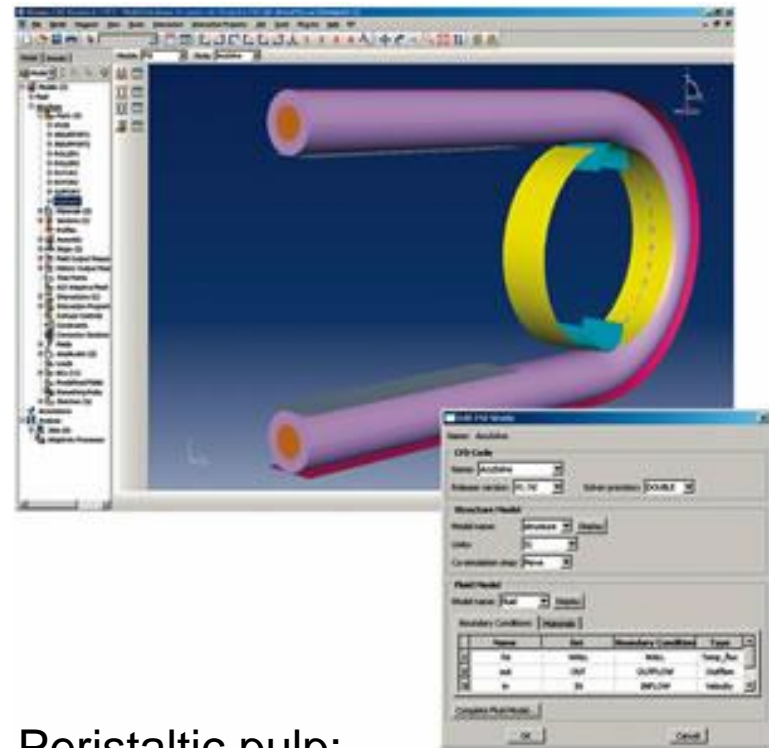
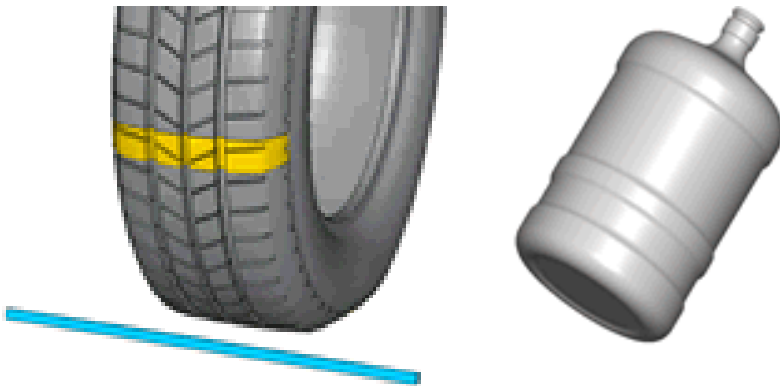
- Abaqus was initially designed to address **highly nonlinear** complex problems
- Good collection of multiphysics capabilities: acoustic-structural, piezoelectric, and structural-pore capabilities, etc.
  - Capabilities for both static and dynamic problems
  - The ability to model very large shape changes 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 hydroplaning and drop of container with fluid



Peristaltic pump:  
Fluid-structure interaction

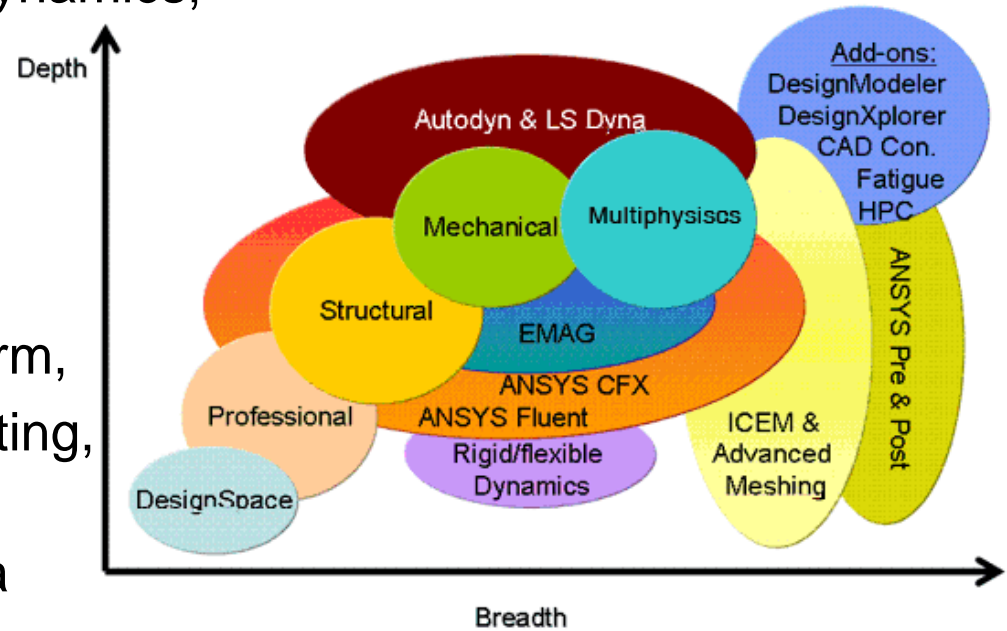
# ANSYS

- ❑ *ANSYS – ANalysis SYStem*
- ❑ Developer: ANSYS Inc.
- ❑ Company founded by:  
Dr. John A. Swanson as  
Swanson Analysis Systems, Inc. SASI
- ❑ Company founded in: 1970, Canonsburg, Pennsylvania
- ❑ Operating system: Microsoft Windows, Linux and IBM AIX
- ❑ License: proprietary commercial software
- ❑ Website: [www.ansys.com](http://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

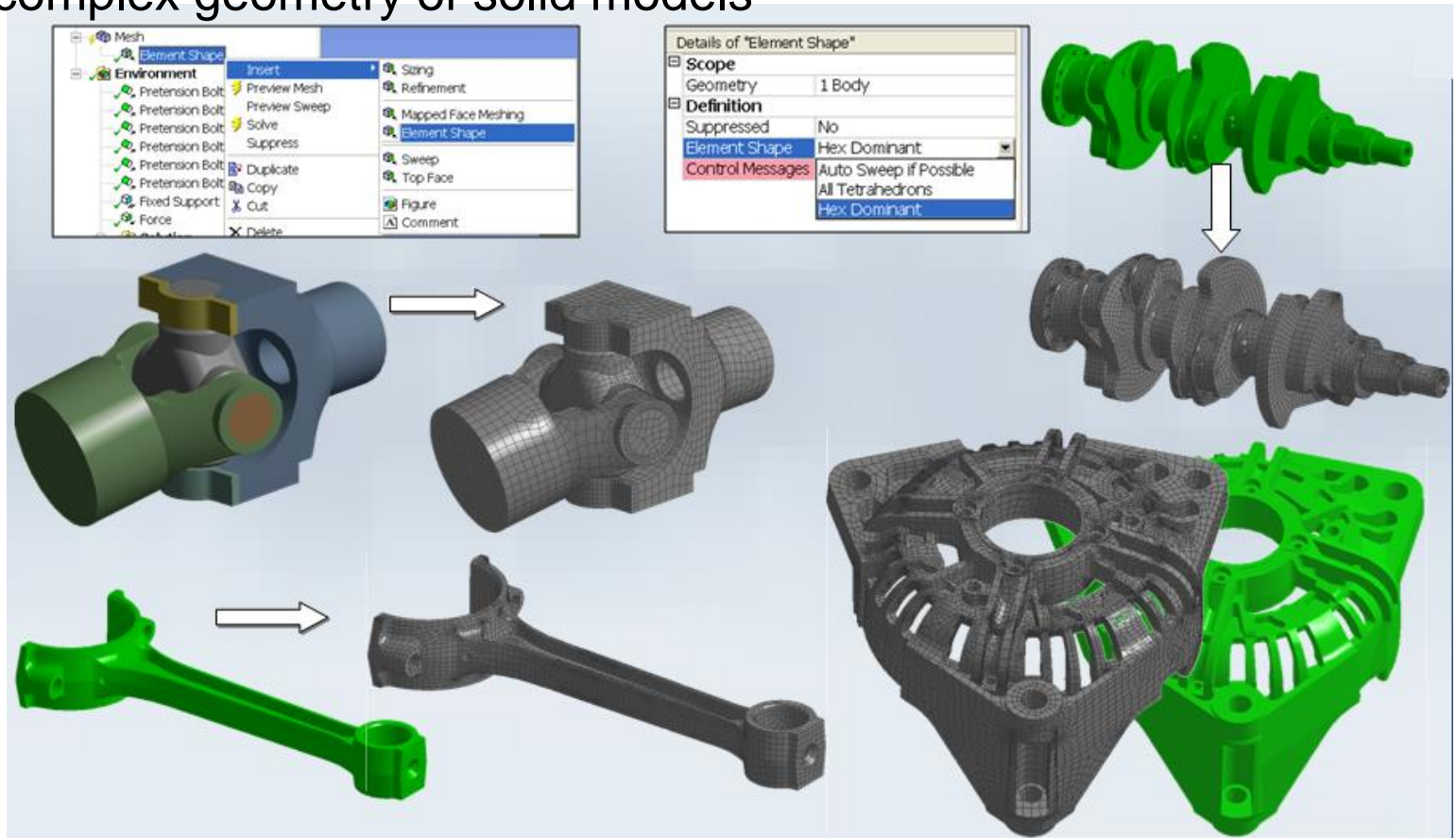


- Largely used in industry:
  - Aerospace & Defense
  - Automotive
  - Construction
  - Consumer Goods
  - Electronics & Semiconductor
  - Energy
  - Healthcare
  - Industrial Equipment & Rotating Machinery
  - Materials & Chemical Processing
  
- Academic and research institutions

# ANSYS: applications (continue)



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

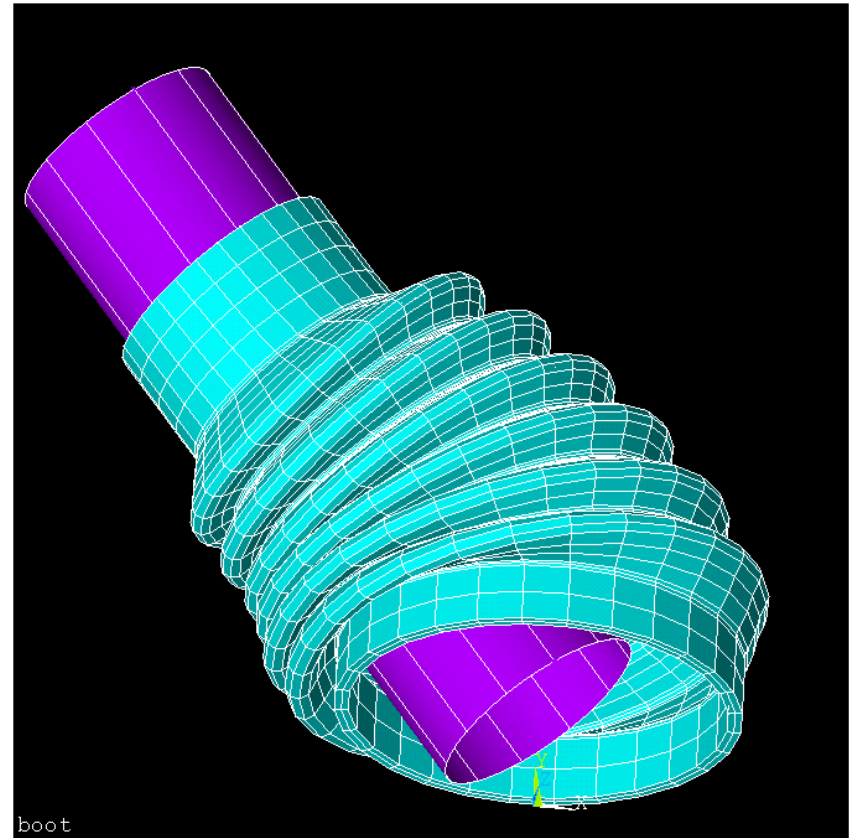


# ANSYS: applications (continue)



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

- *Nonlinear geometry* (large deformations and displacements)
- *Nonlinear material* (rubber)
- *Status change* (contact)



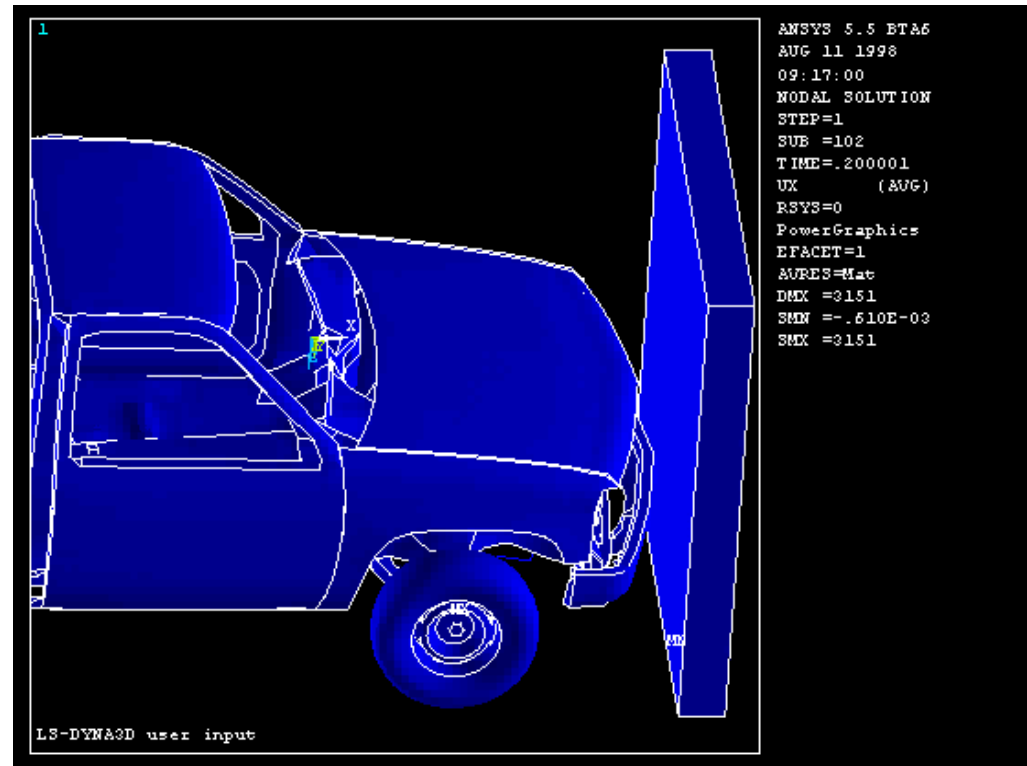
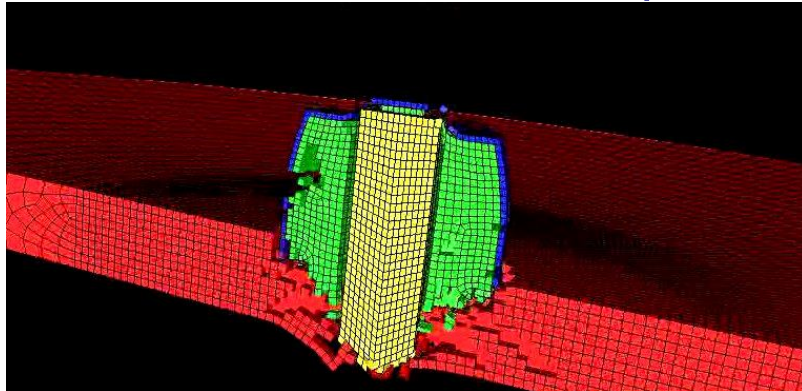
# ANSYS: applications (continue)



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
- 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
- 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
- Operating system: cross-platform
- License: proprietary end-user license agreement
- Website: [www.comsol.com](http://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



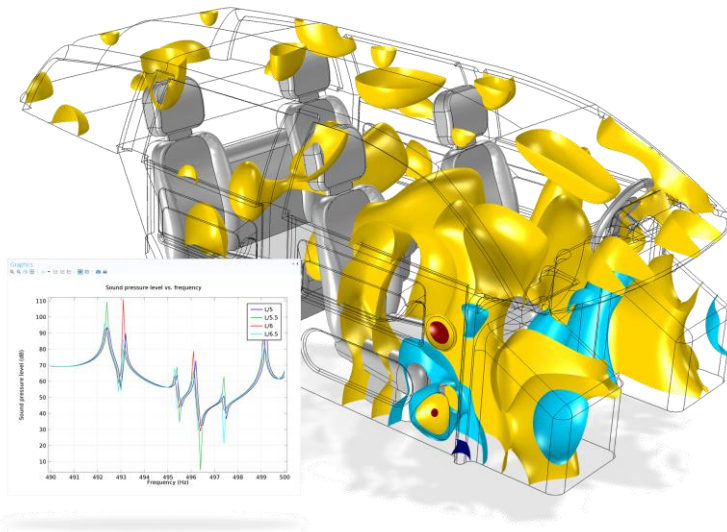
- **COMSOL Multiphysics**
- **COMSOL Server**
  
- **Add-on Products**
  - Electrical
  - Structural & Acoustics
  - Fluid & Heat
  - Chemical
  - Multipurpose
  - Interfacing (LiveLink for MATLAB and others)

# COMSOL: applications



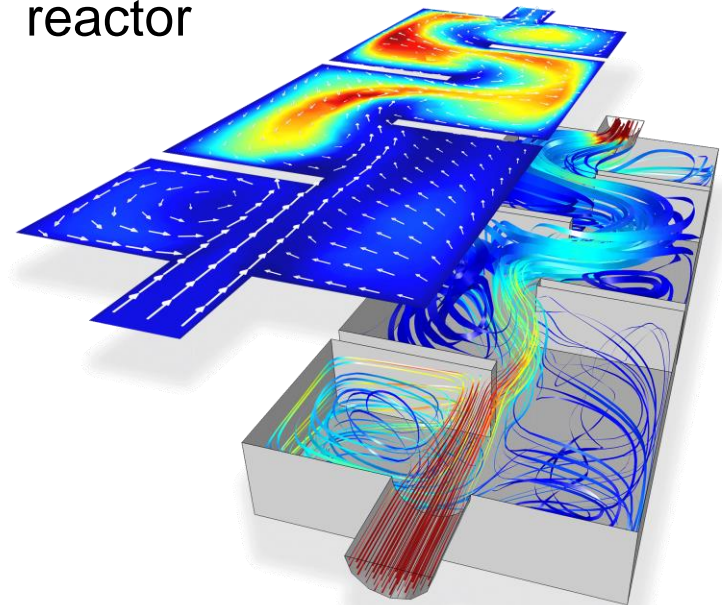
## Acoustics Module

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



## PDE Solutions Inc

Problem Solving Environments Since 1995

- 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 Professional Version 6.00z10/W/32 3D
File Controls View Stop Edit Help
borg7

TITLE 'borgbox'
COORDINATES cartesian3
VARIABLES vx vy vz p
SELECT regrid=off

DEFINITIONS
pipe = 5 box = 10
rad = 2 rads = 0.7*rad
ars = rads*sin(pi/4)

leftbar = cylinder ((-box,0,0), (-box,1,0), rad)
rightbar = cylinder ((box,0,0), (box,1,0), rad)
frontbar = cylinder ((0,-box,0), (1,-box,0), rad)
backbar = cylinder ((0,box,0), (1,box,0), rad)
pipebar = cylinder ((0,0,0), (1,0,0), rad)

INITIAL VALUES
vx=0 vy=0 vz=0 p=Pin*((box-x)/(2*box))

EQUATIONS
vx dens*(vx*dx(vx) + vy*dy(vx) + vz*dz(vx)) + dx(p) -visc*div(grad(vx)) = 0
vy dens*(vx*dx(vy) + vy*dy(vy) + vz*dz(vy)) + dy(p) -visc*div(grad(vy)) = 0
vz dens*(vx*dx(vz) + vy*dy(vz) + vz*dz(vz)) + dz(p) -visc*div(grad(vz)) = 0
p div(grad(p)) = PENALTY*div_v

EXTRUSION
SURFACE '1' z = z1
LAYER 'botbar'
SURFACE '2' z = z2
LAYER 'middle'
SURFACE '3' z = z3
LAYER 'topbar'
SURFACE '4' z = z4
```

FlexPDE Professional Version 6.00z10/W/32 3D

borg7

Elapsed Time 4:30  
Grid 40400  
Nodes 23662  
Cells 161600  
Unknowns 232472  
Mem(K) 1  
RMS Error 1  
Max Error 1

Lanczos@Blockv  
Nodes/Cells

log Convergence(2)

log Error

Mesh

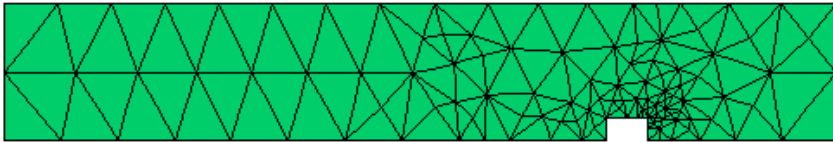
3D Plot: borg7: G181 P2 Ndep=4262 CUB=2362 RMS E= 1.0  
2D Plot: borg7: G181 P2 Ndep=4262 CUB=2362 RMS E= 1.0  
2D Plot: borg7: G181 P2 Ndep=4262 CUB=2362 RMS E= 1.0  
2D Plot: borg7: G181 P2 Ndep=4262 CUB=2362 RMS E= 1.0

# FlexPDE: applications

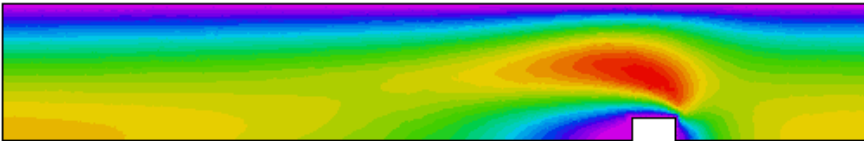
- ❑ Some industrial companies
- ❑ Academic and research institutions

## Viscous Flow in a 2D channel

Adaptively refined mesh:



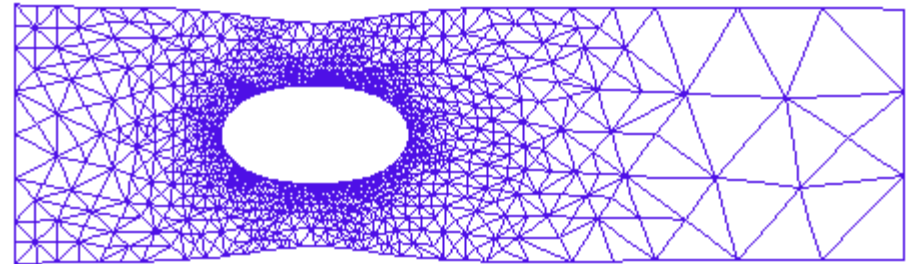
Fluid Speed:



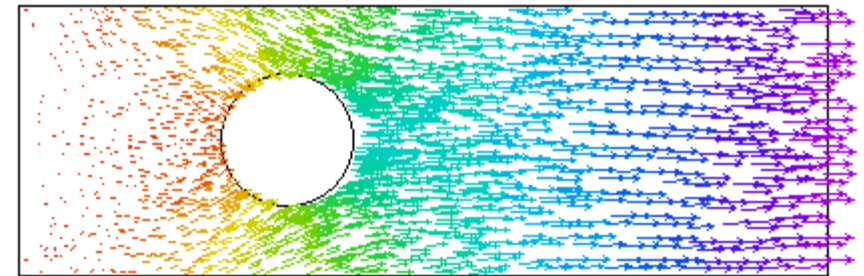
Pressure:



## Stress analyses



The final adaptively refined grid

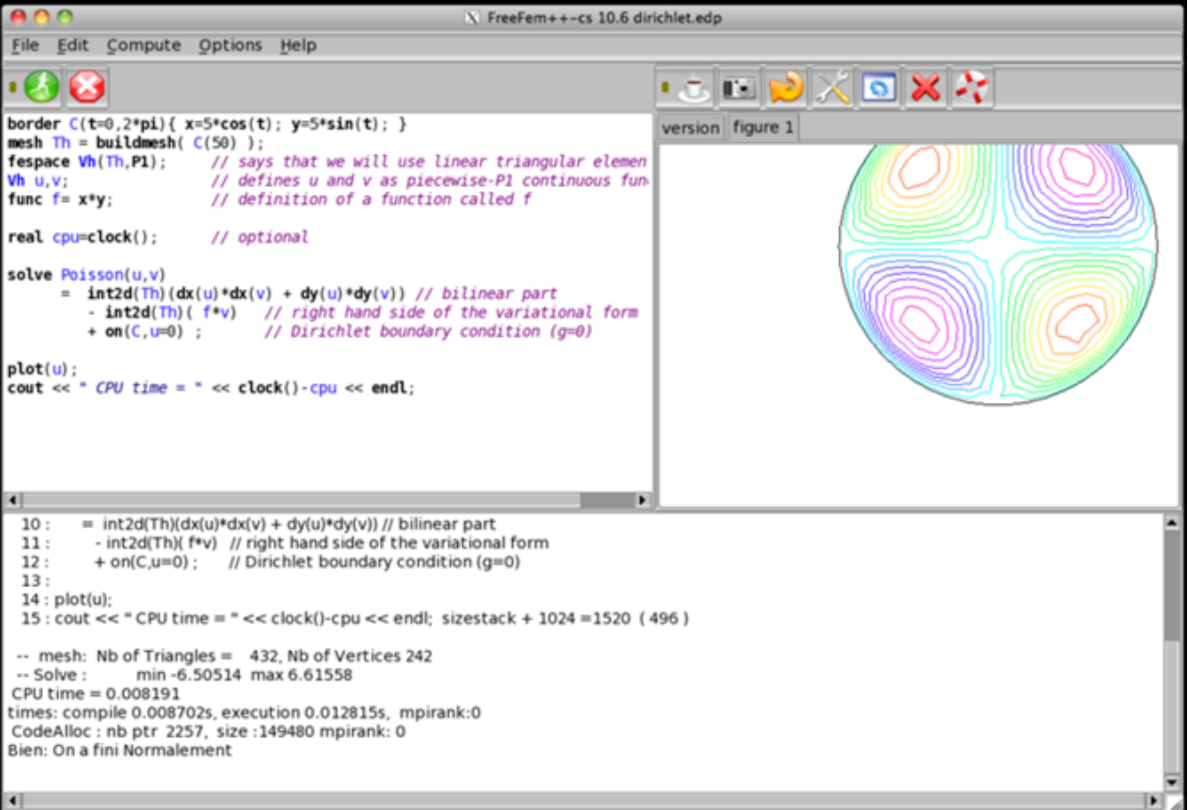


The vector displacement Field

# 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



```
FreeFem++-cs 10.6 dirichlet.edp
File Edit Compute Options Help

border C(t=0,2*pi){ x=5*cos(t); y=5*sin(t); }
mesh Th = buildmesh( C(50) );
fespace Vh(Th,P1); // says that we will use linear triangular elemen
Vh u,v; // defines u and v as piecewise-P1 continuous fun
func f = x*y; // definition of a function called f

real cpu=clock(); // optional

solve Poisson(u,v)
= int2d(Th)(dx(u)*dx(v) + dy(u)*dy(v)) // bilinear part
- int2d(Th)( f*v) // right hand side of the variational form
+ on(C,u=0); // Dirichlet boundary condition (g=0)

plot(u);
cout << " CPU time = " << clock()-cpu << endl;

10 : = int2d(Th)(dx(u)*dx(v) + dy(u)*dy(v)) // bilinear part
11 : - int2d(Th)( f*v) // right hand side of the variational form
12 : + on(C,u=0); // Dirichlet boundary condition (g=0)
13 :
14 : plot(u);
15 : cout << " CPU time = " << clock()-cpu << endl; sizestack + 1024 = 1520 ( 496 )

-- mesh: Nb of Triangles = 432, Nb of Vertices 242
-- Solve : min -6.50514 max 6.61558
CPU time = 0.008191
times: compile 0.008702s, execution 0.012815s, mpirank:0
CodeAlloc : nb ptr 2257, size :149480 mpirank: 0
Bien: On a fini Normalement
```