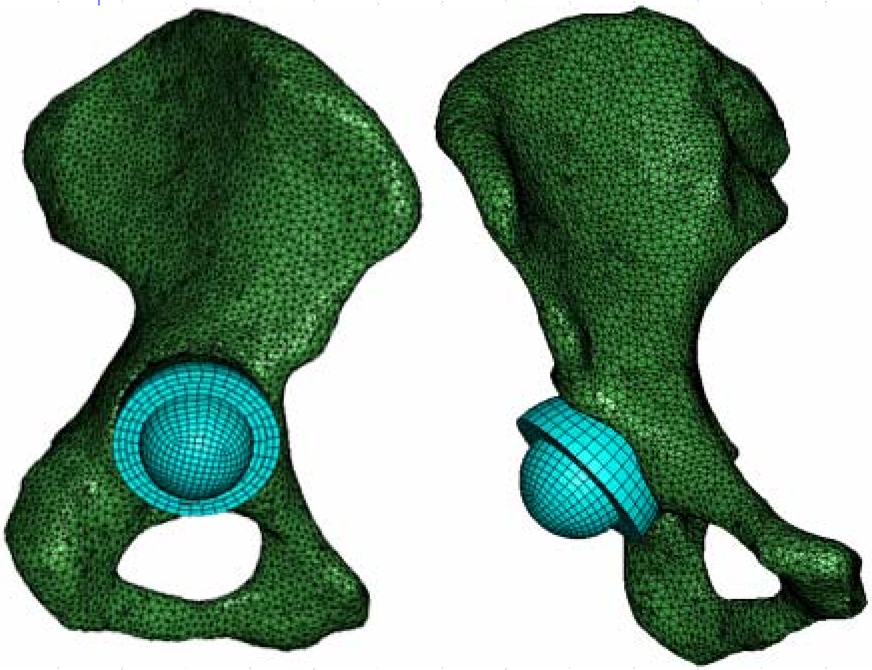
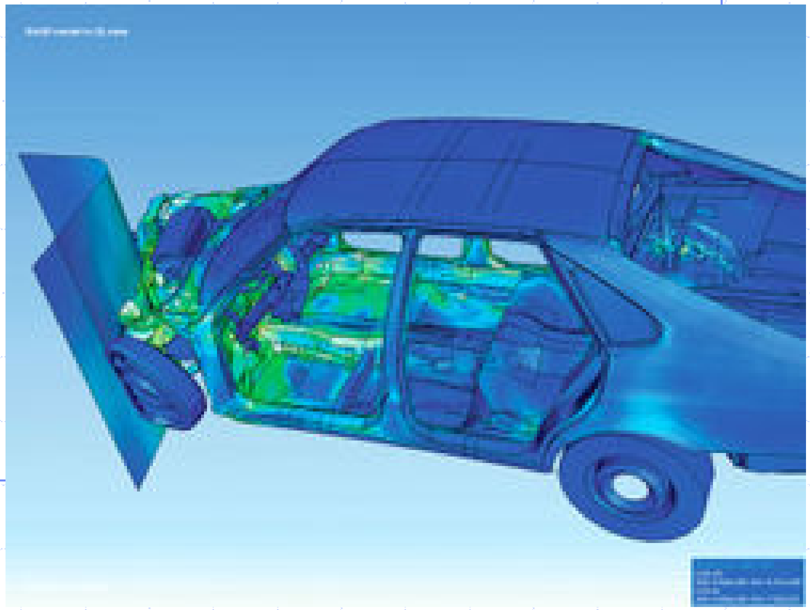
Finite Element Analysis (FEA)



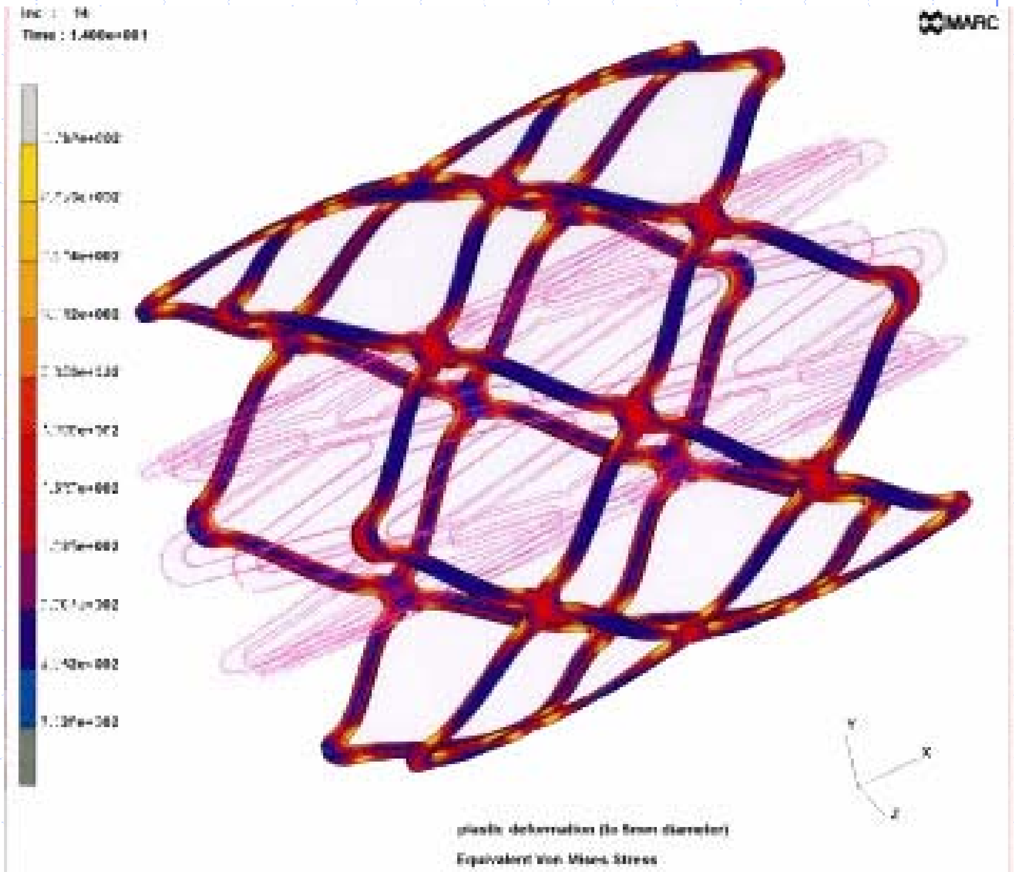


What Is FEA?

- Computer model
- Allows numerical solution of equations of complicated geometries
  - divided into many small regions called "finite elements“
- Assemblage of all elements' equations produces a large matrix
- Simulation

Use of FEA

- Structure
- Solid mechanics
- Dynamics
- Thermal analysis
- Electrical analysis
- Biomaterials



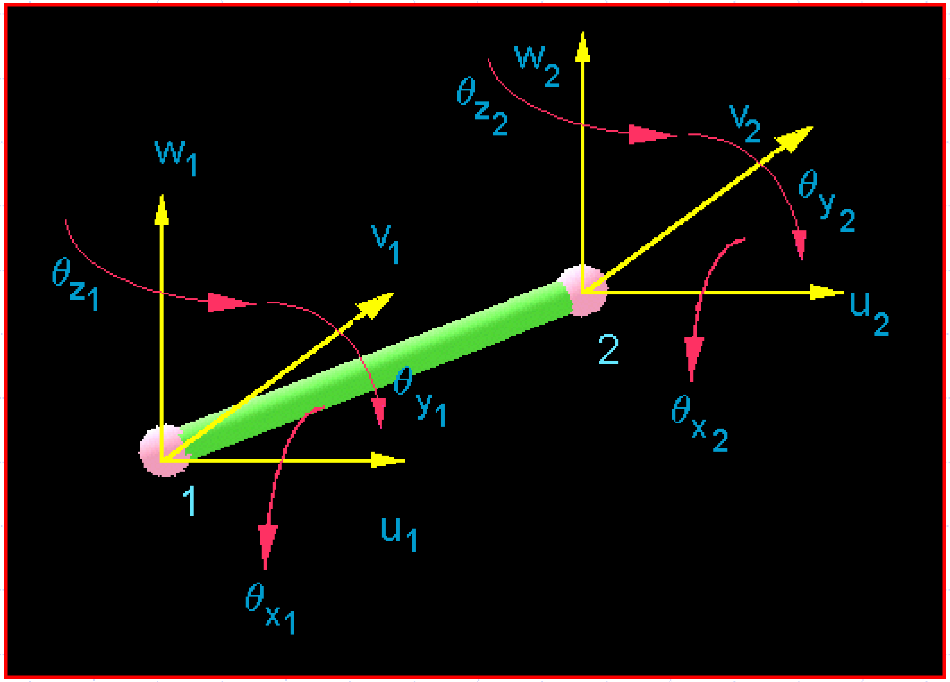
Analysis of coronary stents

Why The FEA?

- Greater flexibility to model complex geometries than
- Used in solving structural, mechanical, heat transfer, fluid dynamics, etc.
- Much cheaper than if each sample was actually built and tested

How Does It Work?

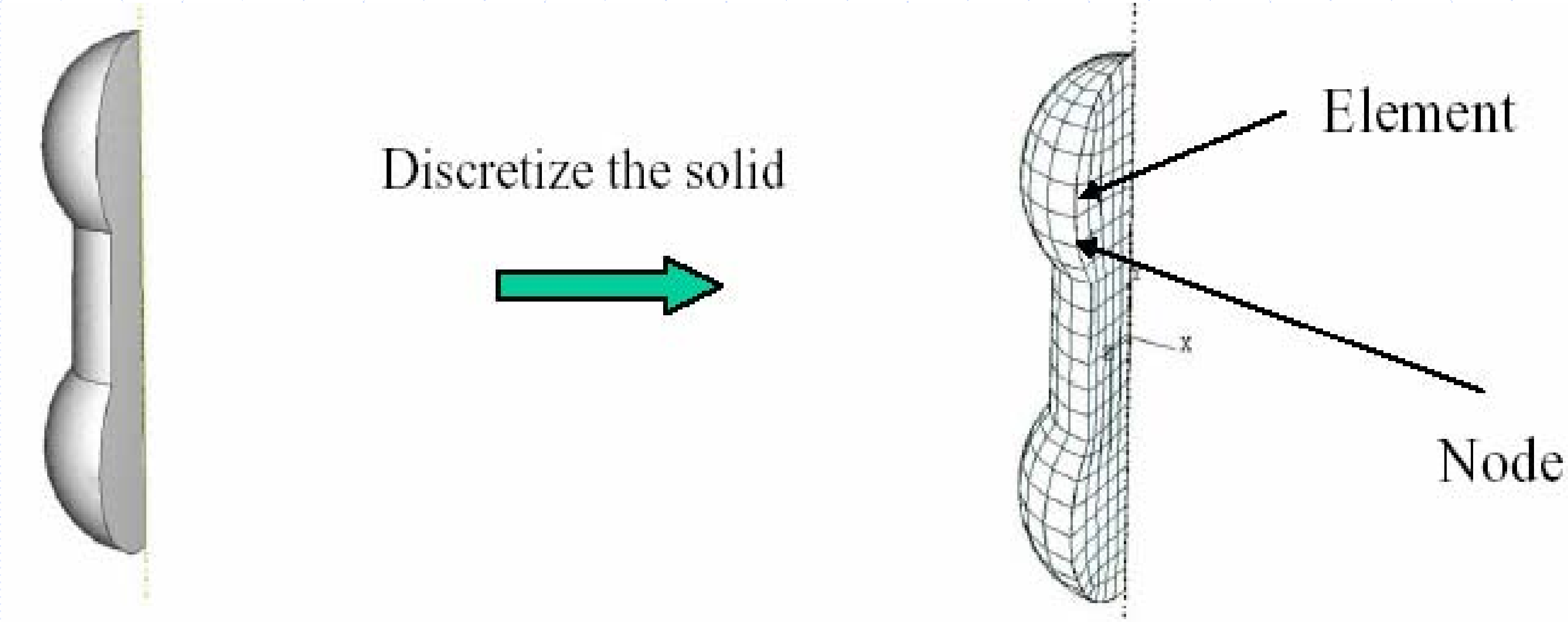
- Uses system of points called nodes which make a grid called a mesh
- Mesh is programmed to contain the material and structural properties.
- Defines how the structure will react to certain loading conditions



Preprocessing

- Construct a FE model of the structure to be analyzed
- Can be in either 1D, 2D or 3D form
- Primary objective is to replicate important parameters & features of the real model
  - utilize pre-existing digital blueprints, design files, CAD models
- Preprocessing Link

- Once created, a meshing procedure is used to define and break the model up into small elements



A Quarter of Object

Mesh of the 3D Solid

Analysis

- Series of computational procedures involving applied forces, and the properties of the elements which produce a model solution
- Effects such as:
  - Deformations, strains, and stresses
    - Caused by applied structural loading conditions

Use a simple function to approximate the displacements in each element

→

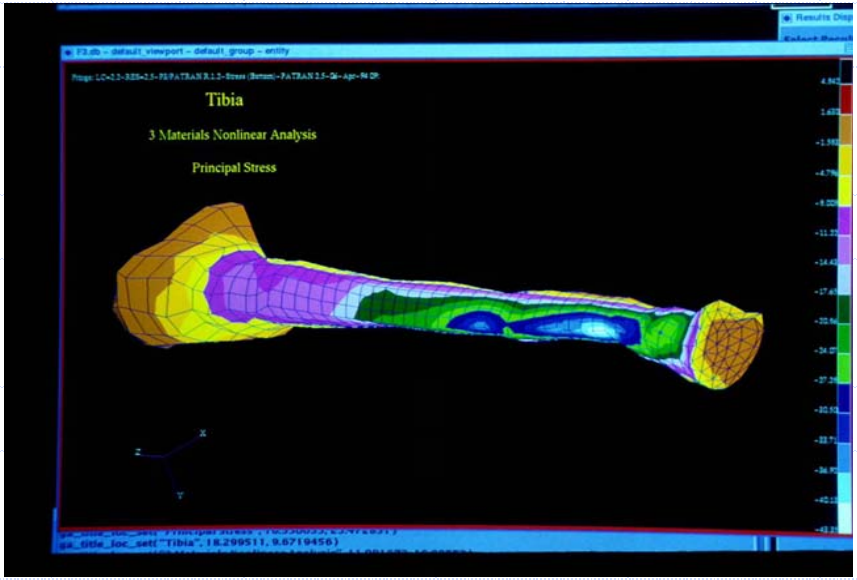
Formulate a set of linear equations with displacements at each node as unknowns

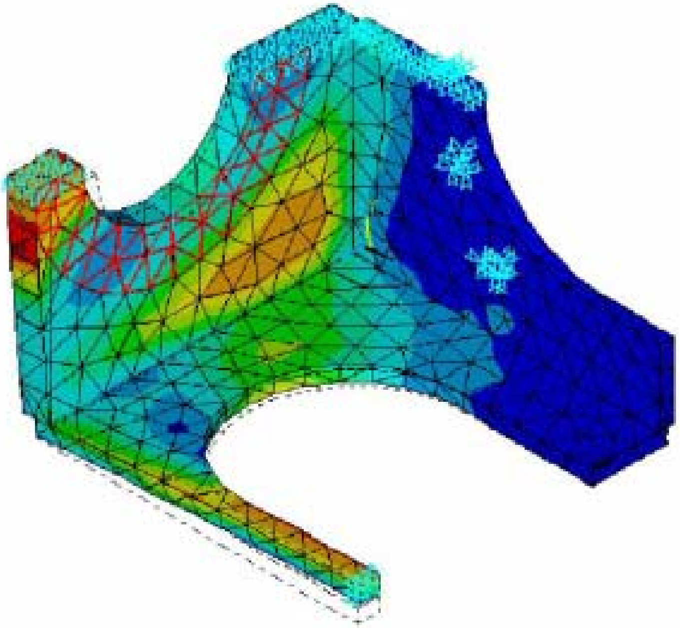
→

Solve linear equations

Variables Within The System

- Mass, volume, temperature
- Strain energy, stress strain
- Force, displacement, velocity, acceleration
- Synthetic (User defined)





Input

Boundary Conditions (prescribed force, prescribed displacement, etc.)

↓

Output

Stresses, Strains, Displacements, at each material point (X_1, X_2, X_3)

Postprocessing

New Developments in FEA

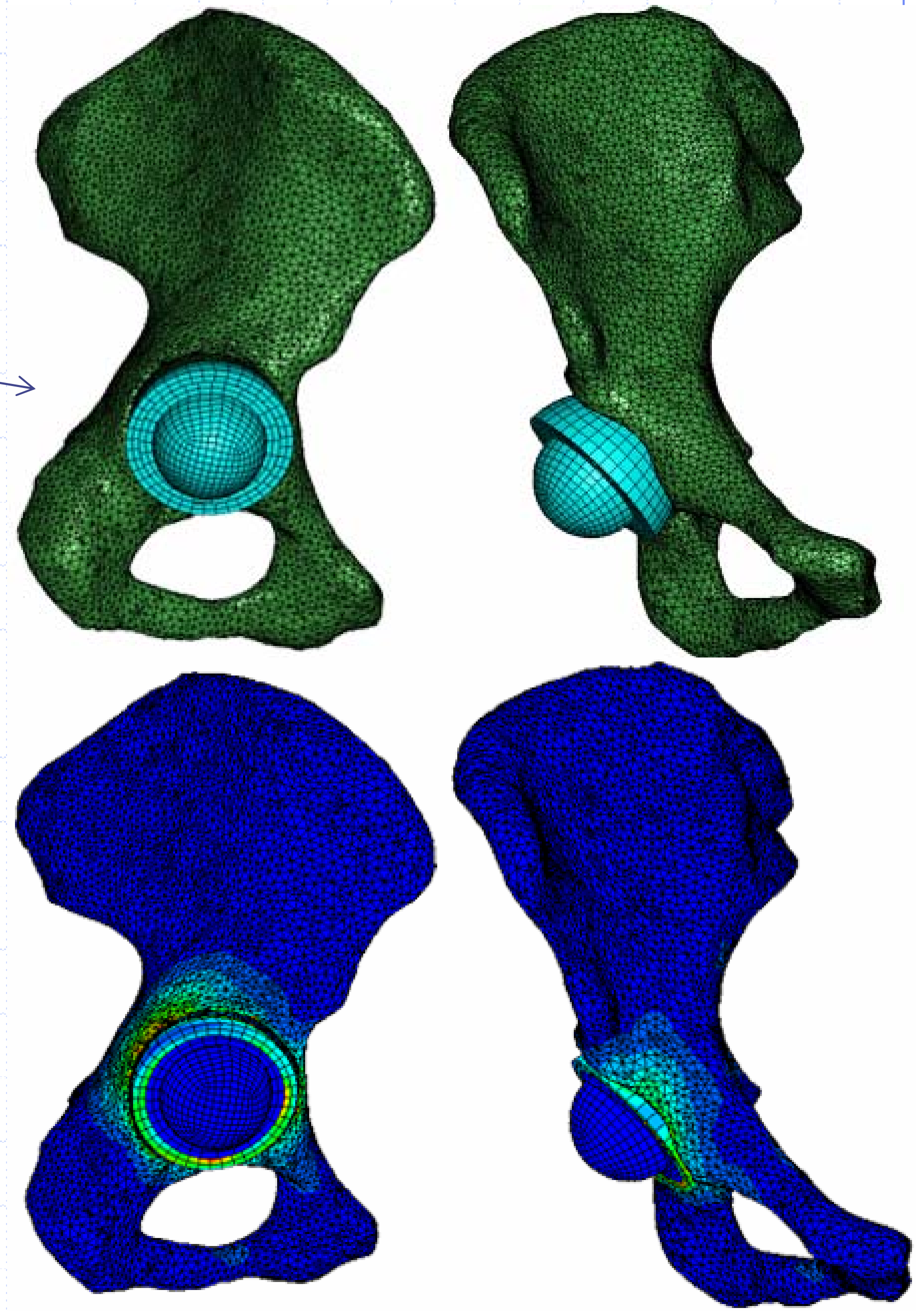
- Integrating FEA into CAD design software
- Self-adaptive analysis
- Analysis of problem of huge size
- Multi-scale analysis
- Multi-physics analysis

hip endoprosthesis

http://www.endolab.de/computer/computersimulation_e.htm

Case Study

- Incorrect acetabular cup
  - 2mm larger than reamed hip
- Correct
  - Significant decrease in stress



http://web.missouri.edu/~smithdoug/complab/resproj/cb_fea_hipcup.html

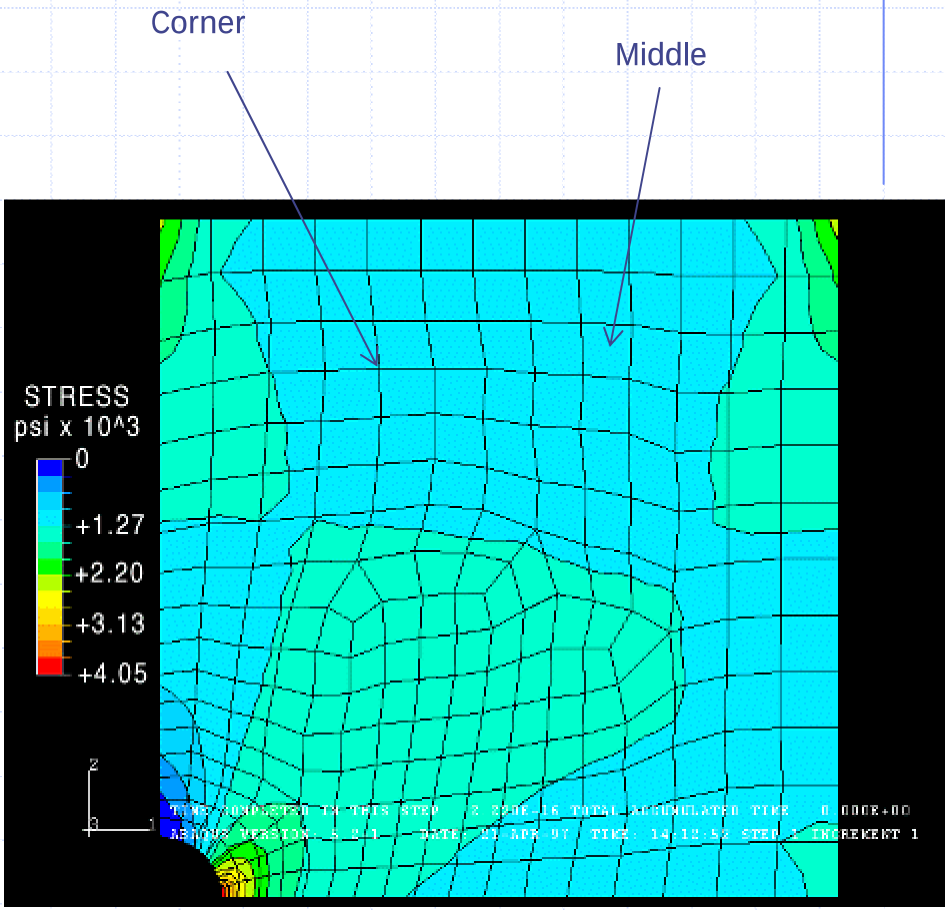
Helpful Examples

- http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/frames.html
  - Look under 3. Numerical Approach
  - Example in ‘Theory’ Link
- http://www.colorado.edu/MCEN/MCEN4173/chap_01.pdf
- http://www.endolab.de/computer/computersimulation_e.htm

The End

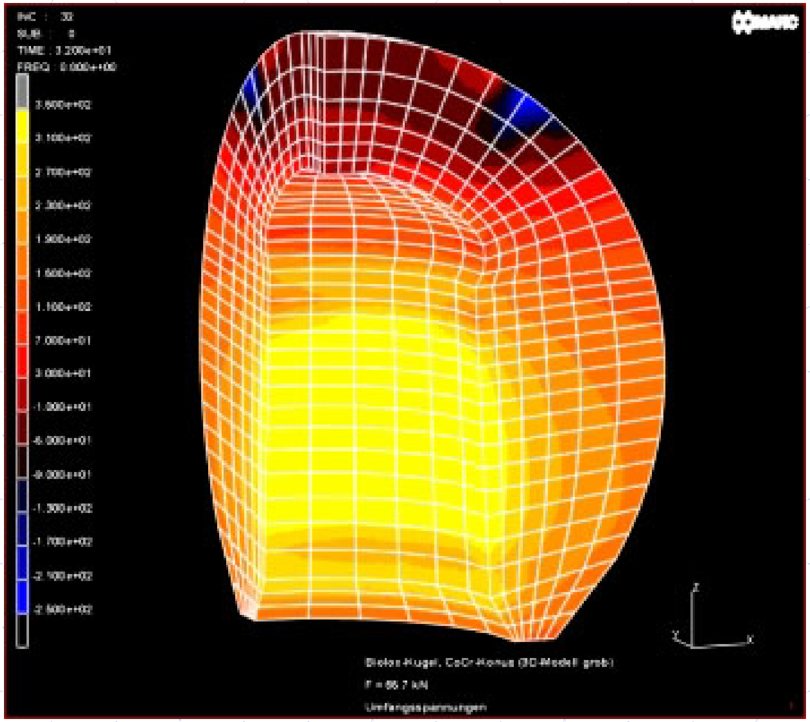
Nodes

- Points at which elements are jointed
- Nodes are the locations where values are to be approximated
- Can be either at the corner or middle of element



Nodes

- Certain density throughout material
- Bounded by sets of nodes, and define localized mass and stiffness properties of the model



Analysis of a ceramic ball for hip replacement

Loading Conditions

- Point, pressure, thermal, gravity, and centrifugal static loads
- Thermal loads from solution of heat transfer analysis
- Enforced displacements
- Heat flux and convection
- Point, pressure and gravity dynamic loads