عنوان درس:

طراحی به کمک کامپیوتر پیشرفته (کاربرد روش های عددی در تحلیل فرآیندهای تولید)

۱- معرفی و توانایی های نرم افزار ABAQUS

۲- معرفی و استفاده از راهنمای نرم افزار ABAQUS

۳- روش های حل مسایل اجزای محدود ضمنی (Implicit) و صریح (Explicit)

۴- تعریف تماس، تئوری و کاربرد

۵- حل مثال های از فرآیندهای مکانیکی و شکل دهی

۶- توانایی های نرم افزار در تجدید شبکه، Restart، پس پردازش نرم افزار

۷- حل مثال های از فرآیندهای ترمومکانیکی، مکانیک آسیب، شکست

۸- زیر برنامه ها در نرم افزار

۹- تدوین زیر برنامه در نرم افزار

Introduction

Physical Problem

↓

Question regarding the problem

...how large are the deformations?

...how much is the heat transfer?

Mathematical model

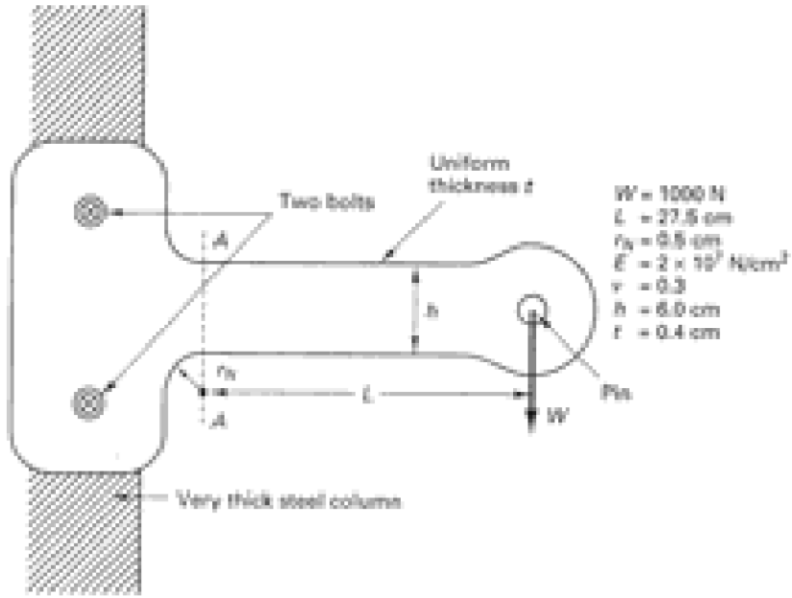
(Governed by differential Equations)

Assumptions regarding

- Geometry
- Kinematics
- Material law
- Loading
- Boundary conditions
- Etc.

Physical problem

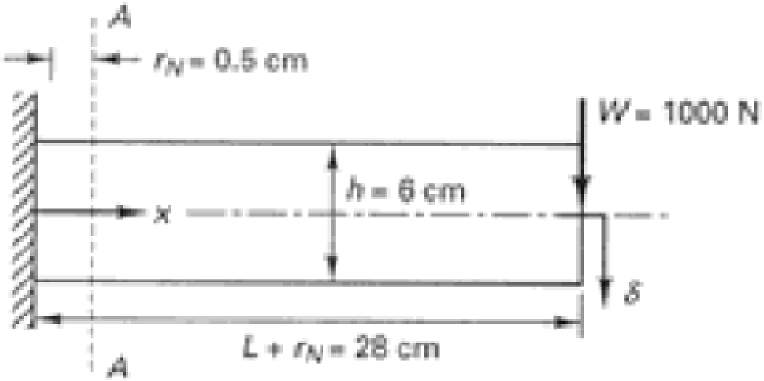
Example: A bracket



Questions:

1. What is the bending moment at section AA?
2. What is the deflection at the pin?

Mathematical model 1: beam



Moment at section AA: $M = WL$

$$= 27{,}500 \text{ N cm}$$

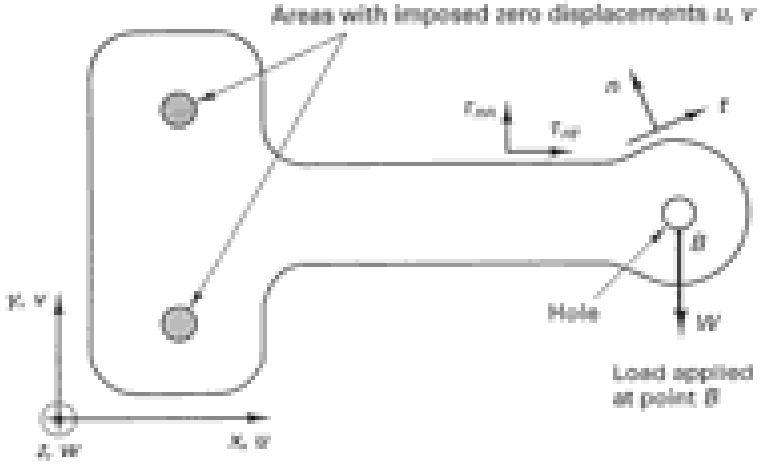
Deflection at load: $$\delta_{\text{at load W}} = \frac{1}{3}\frac{W(L+r_N)^3}{EI} + \frac{W(L+r_N)}{\frac{5}{6}AG}$$

$$= 0.053 \text{ cm}$$

How **reliable** is this model?

How **effective** is this model?

Mathematical model 2: plane stress



Equilibrium equations (see Example 4.2)

$$\left.\begin{aligned}\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} = 0 \\ \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} = 0\end{aligned}\right\} \text{ in domain of bracket}$$

$\tau_{nn} = 0,\ \tau_{nt} = 0$ on surfaces except at point B and at imposed zero displacements

Stress-strain relation (see Table 4.3):

$$\begin{bmatrix}\tau_{xx}\\ \tau_{yy}\\ \tau_{xy}\end{bmatrix} = \frac{E}{1-\nu^2}\begin{bmatrix}1 & \nu & 0\\ \nu & 1 & 0\\ 0 & 0 & (1-\nu)/2\end{bmatrix}\begin{bmatrix}\epsilon_{xx}\\ \epsilon_{yy}\\ \gamma_{xy}\end{bmatrix}$$

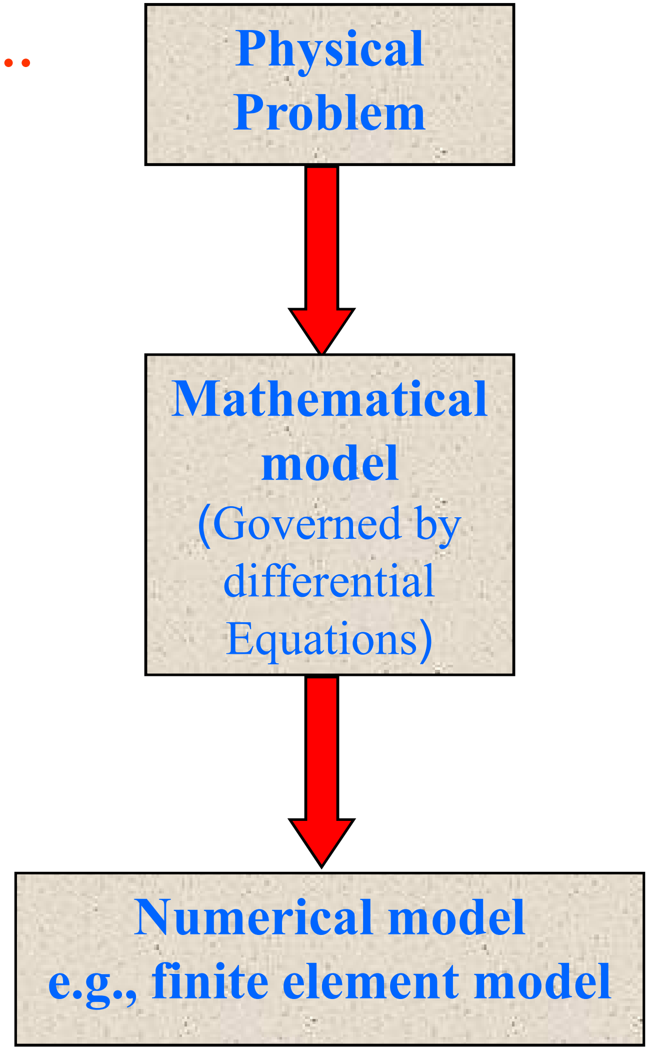
E = Young's modulus, ν = Poisson's ratio

Strain-displacement relations (see Section 4.2):

$$\epsilon_{xx} = \frac{\partial u}{\partial x}; \qquad \epsilon_{yy} = \frac{\partial v}{\partial y}; \qquad \gamma_{xy} = \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}$$

Difficult to solve by hand!

..General scenario..

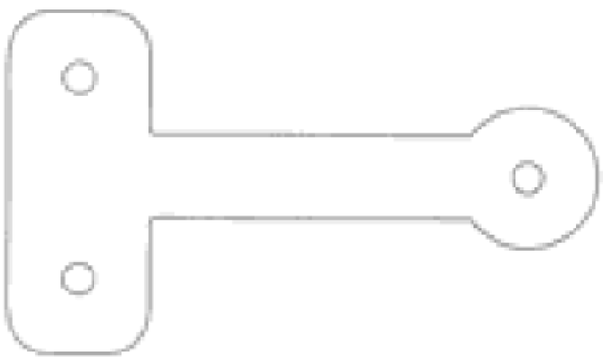


..General scenario..

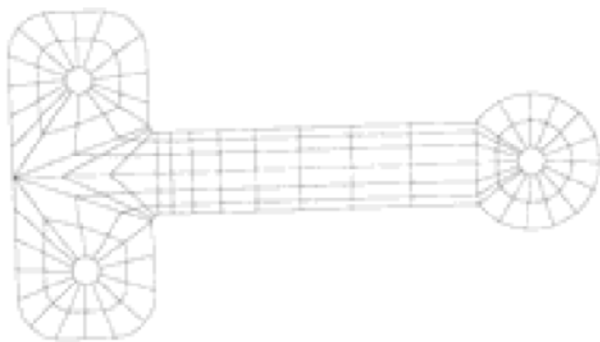
Finite element analysis

PREPROCESSING

1. Create a geometric model
2. Develop the finite element model



Solid model



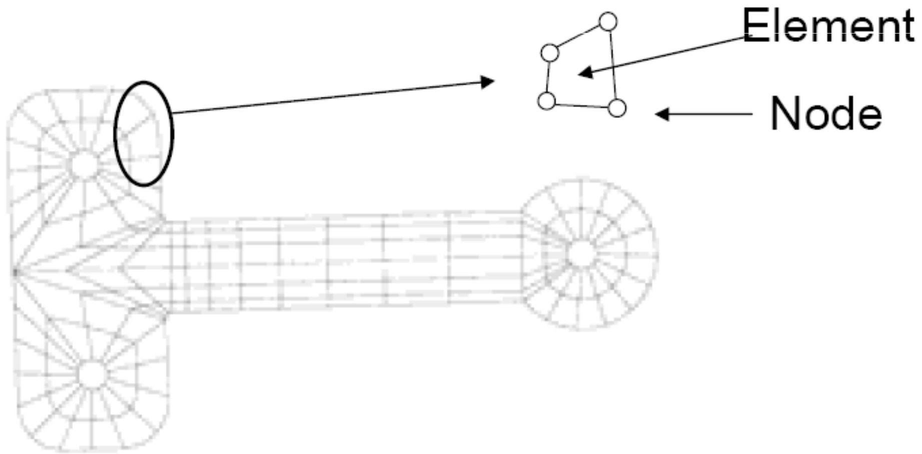
Finite element model

..General scenario..

Finite element analysis

FEM analysis scheme

Step 1: Divide the problem domain into non overlapping regions ("elements") connected to each other through special points ("nodes")



Finite element model

..General scenario..

Finite element analysis

FEM analysis scheme

Step 2: Describe the behavior of each element

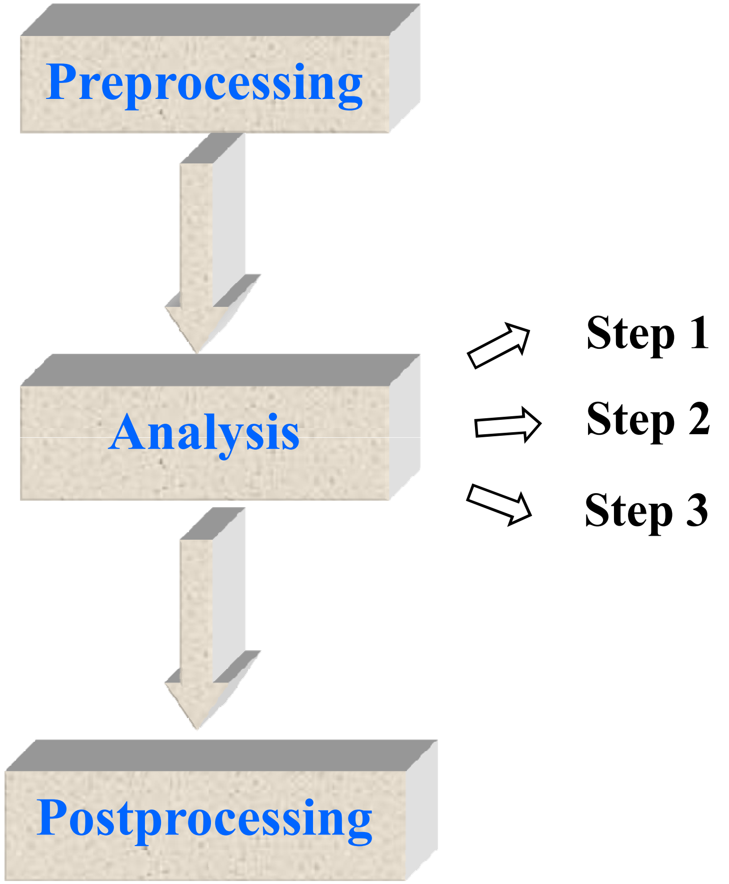
Step 3: Describe the behavior of the entire body by putting together the behavior of each of the elements (this is a process known as "assembly")

POSTPROCESSING

Compute moment at section AA

..General scenario..

Finite element analysis



Example: A bracket

Mathematical model 2: plane stress

FEM solution to mathematical model 2 (plane stress)

Moment at section AA M=27,500 N cm

Deflection at load $\delta_{\text{at load W}} = 0.064\ \text{cm}$

Conclusion: With respect to the questions we posed, the beam model is reliable if the required bending moment is to be predicted within 1% and the deflection is to be predicted within 20%. The beam model is also highly effective since it can be solved easily (by hand).

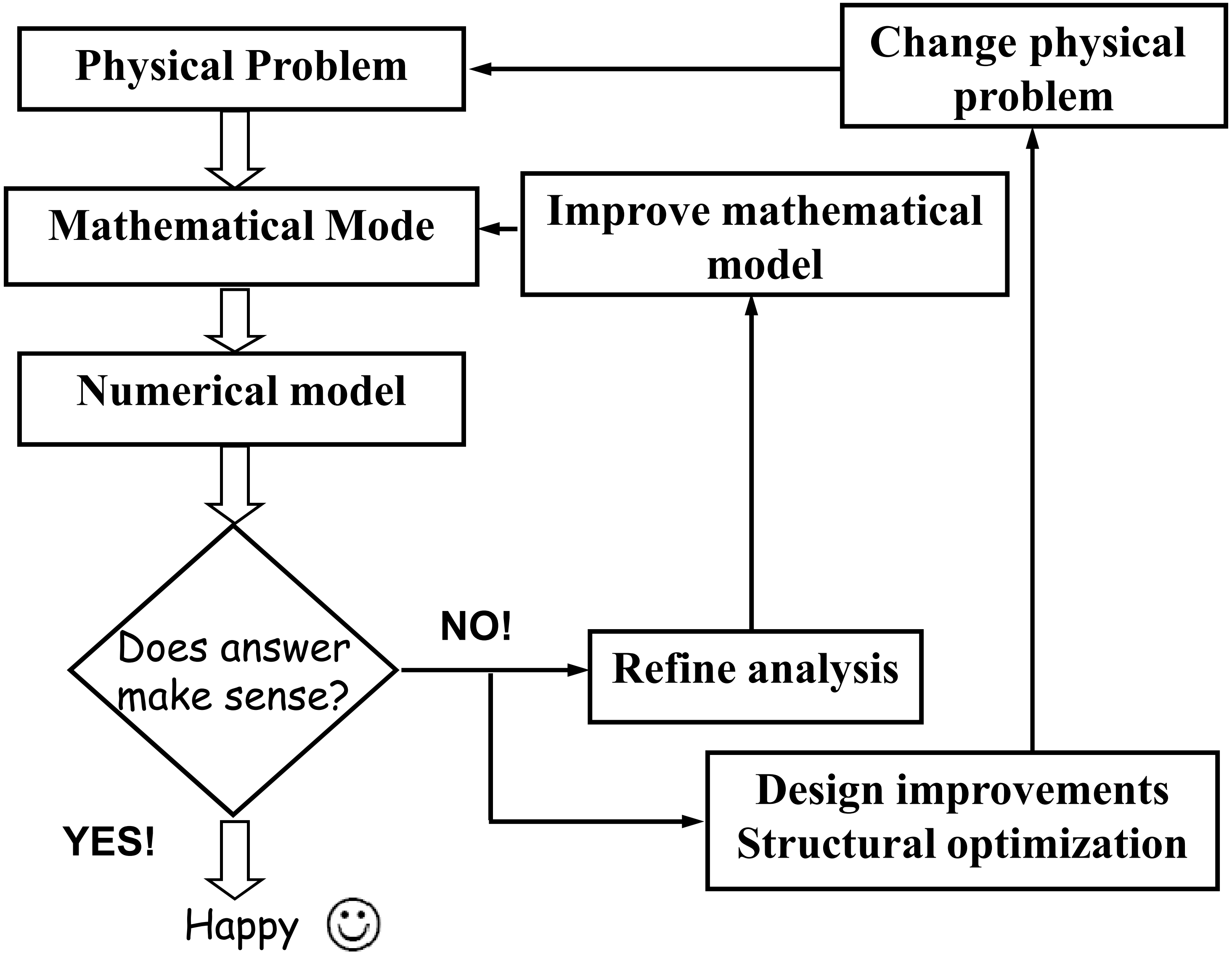
What if we asked: what is the maximum stress in the bracket? would the beam model be of any use?

Example: A bracket

Summary

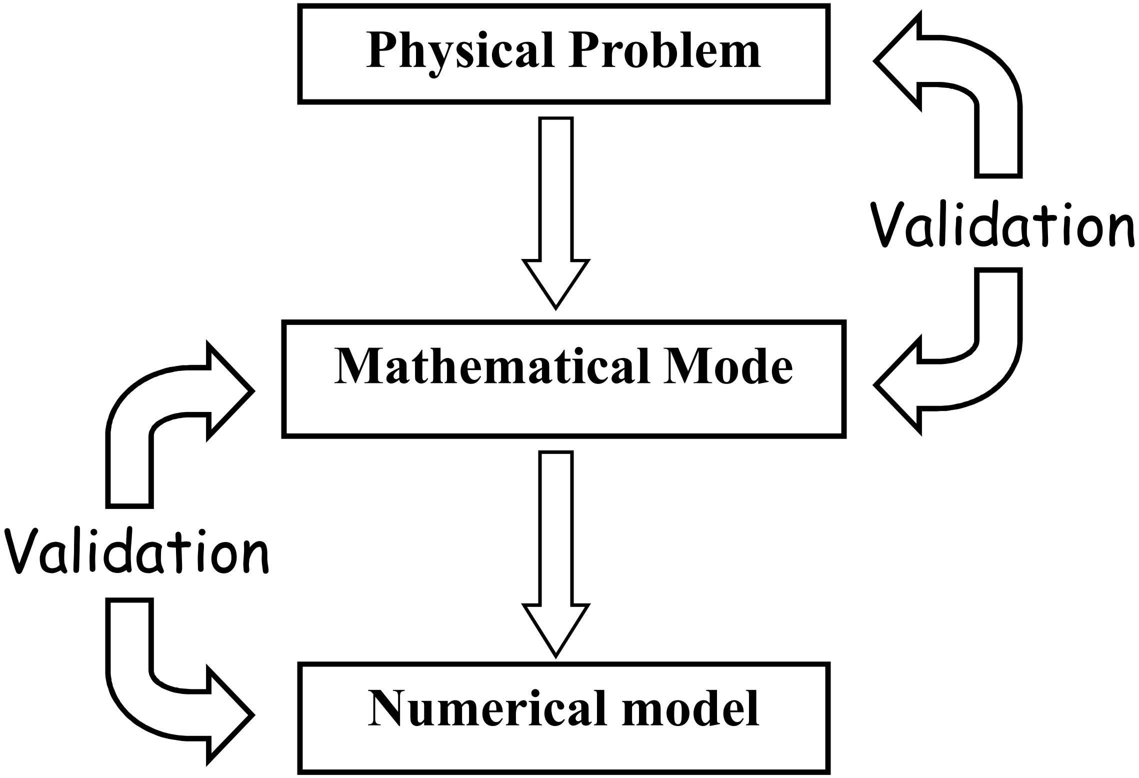
1. The selection of the mathematical model depends on the response to be predicted.
2. The most effective mathematical model is the one that delivers the answers to the questions in reliable manner with least effort.
3. The numerical solution is only as accurate as the mathematical model.

Modeling a physical problem

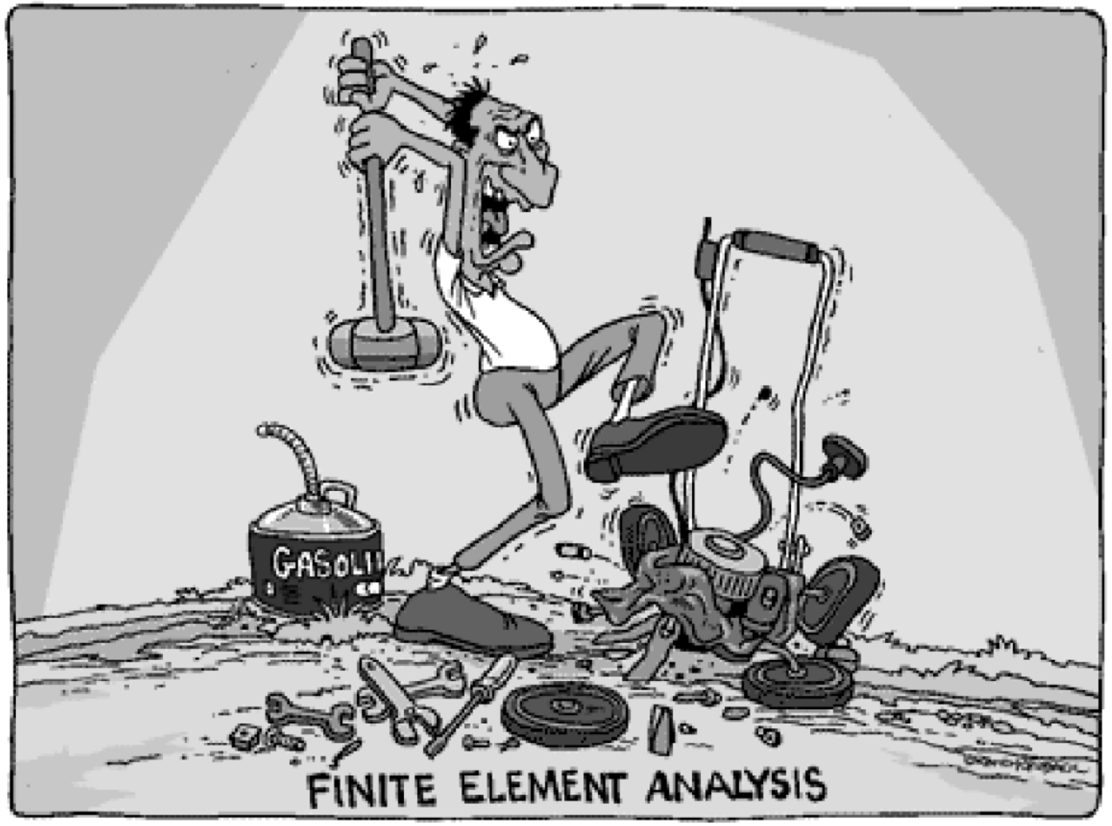


Modeling a physical problem

Verification and validation

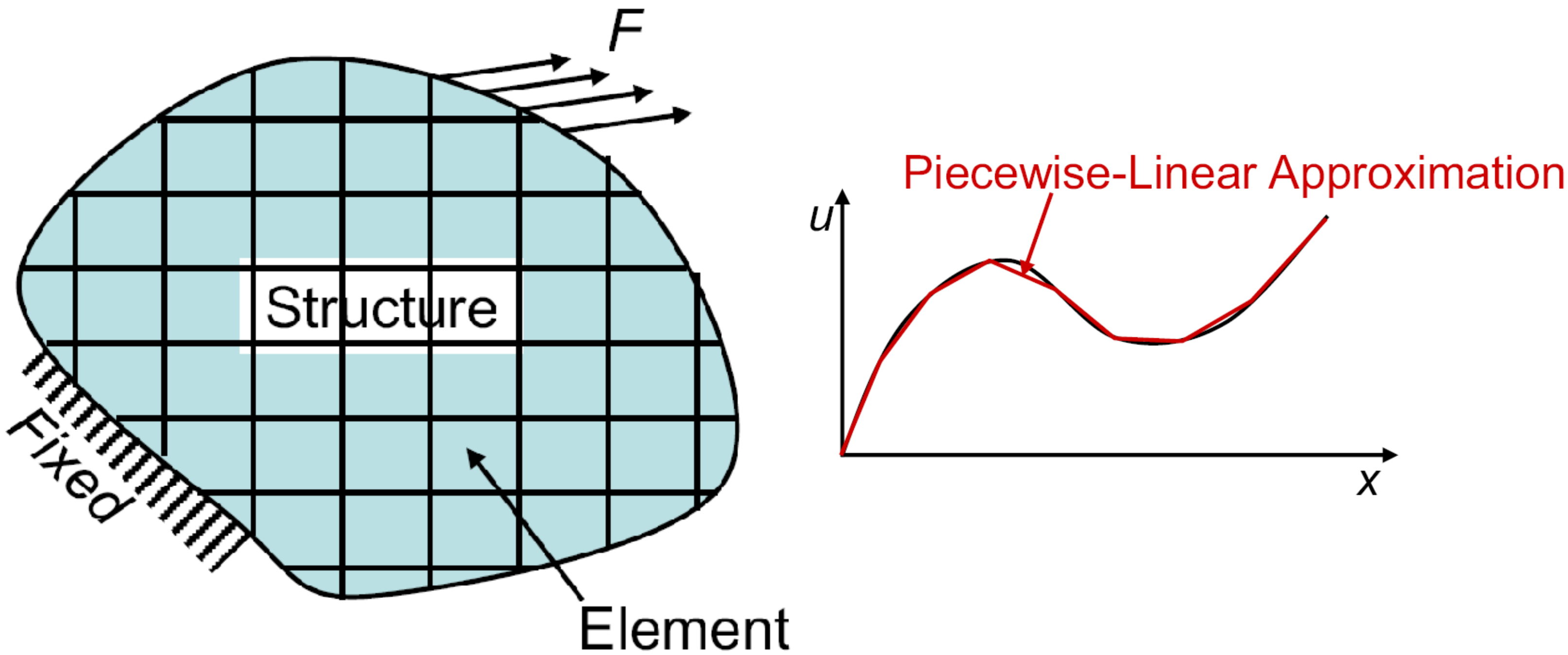


Finite Element Analysis



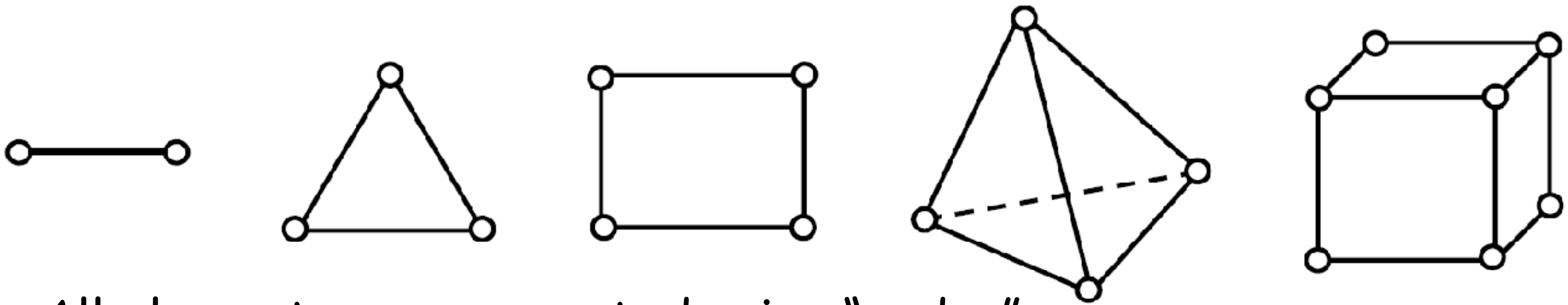
What is the finite element method (FEM)?

- A technique for obtaining approximate solutions to boundary value problems.
- Partition of the domain into a set of simple shapes (element)
- Approximate the solution using piecewise polynomials within the element

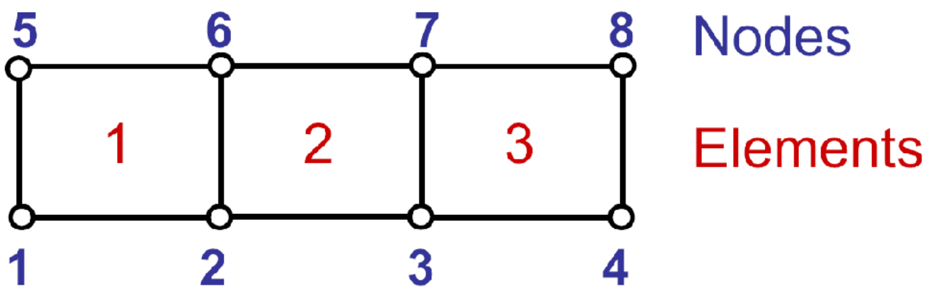


- How to discretize the domain?

– Using simple shapes (element)



– All elements are connected using "nodes".



– Solution at Element 1 is described using the values at Nodes 1, 2, 6, and 5 (Interpolation).

- Finite element analysis solves for Nodal Solutions.

• How to calculate nodal solutions?

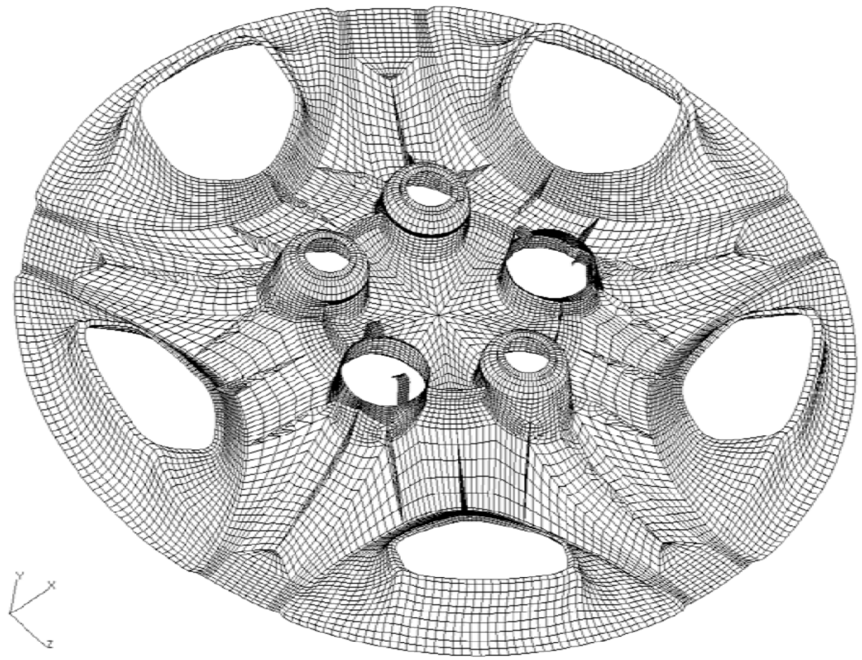
- Construct a huge simultaneous system of equations and solve for nodal solutions.

- Different physical problems have different matrices and vectors.

$$\begin{bmatrix} K_{11} & K_{12} & \cdots & K_{1n} \\ K_{21} & K_{22} & \cdots & K_{2n} \\ \vdots & \vdots & \ddots & \vdots \\ K_{n1} & K_{n2} & \cdots & K_{nn} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_n \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_n \end{Bmatrix}$$

EXAMPLE: FINITE ELEMENTS

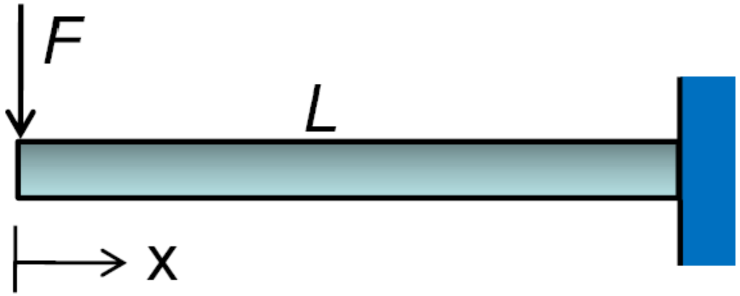
- Plastic Wheel Cover Model
- – 30,595 Nodes, 22,811 Elements
- – Matrix size is larger than 150,000 x150,000.



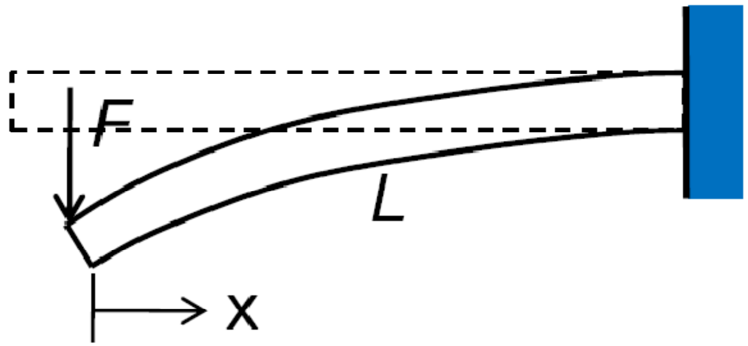
Introduction to Nonlinear Finite Element Analysis Procedures

An Example: Bending of A Cantilever Beam

- Linear Problem
  (Infinitesimal Deformation)



- Nonlinear Problem
  (Large Deformation)



. Moment is a function of deformation

Linear Problem:

- Infinitesimal deformation: $\varepsilon_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$

  Undeformed coord.

- Linear stress-strain relation: $\sigma = \mathbf{C} : \varepsilon$

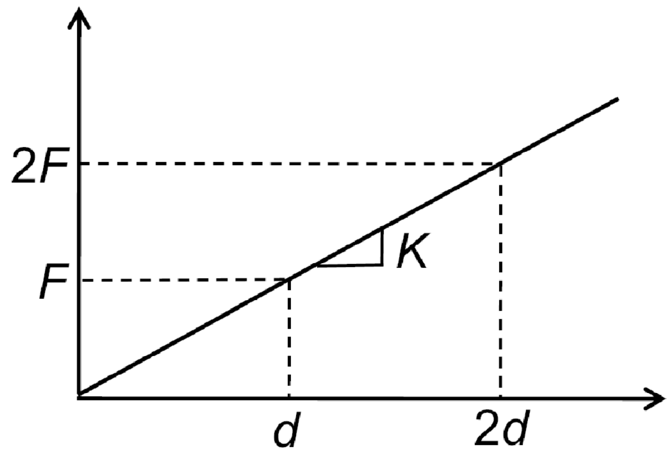
  Constant

- Constant displacement BCs
- Constant applied forces

Solution Procedure

$$\mathbf{K}.\mathbf{d} = \mathbf{F} \; or \; P(\mathbf{d}) = \mathbf{F}$$

- Stiffness matrix $\mathbf{K}$ is constant



If the load is doubled, displacement is doubled, too

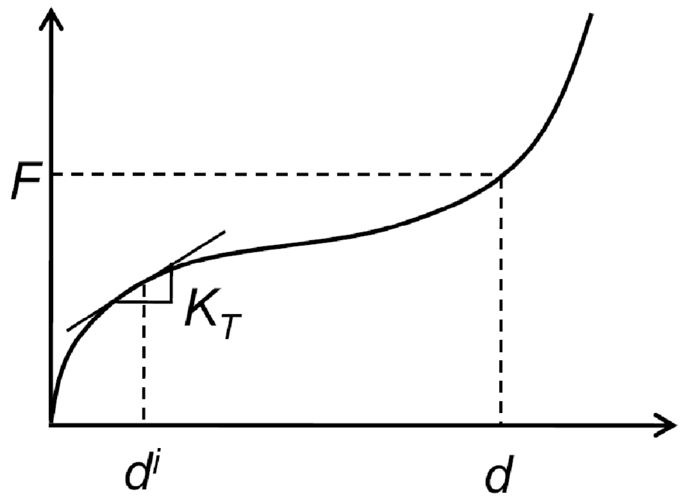
Nonlinear Problem:

- Everything except for linear problems!
- Geometric nonlinearity: nonlinear strain-displacement relation
- Material nonlinearity: nonlinear constitutive relation
- Kinematic nonlinearity: Non-constant displacement BCs, contact
- Force nonlinearity: follow-up loads

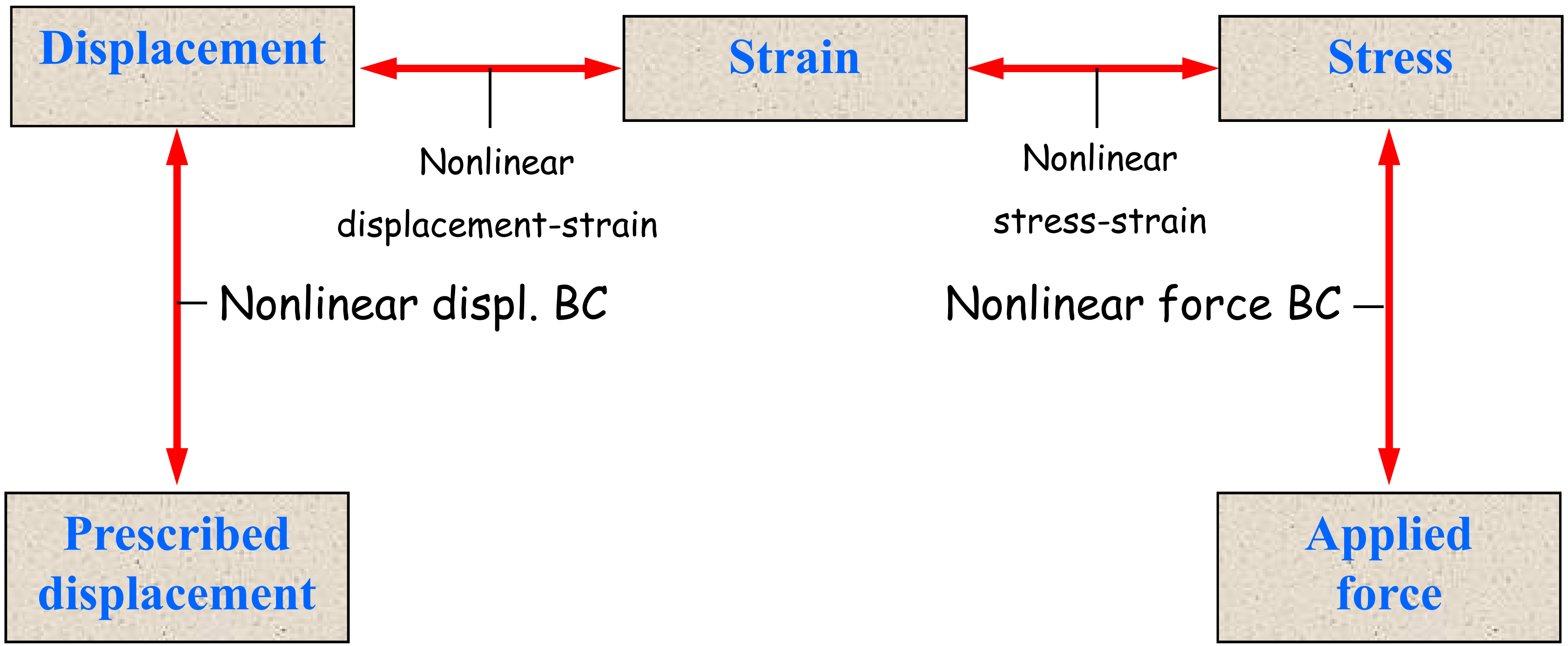
Solution Procedure

- How to find d for a given $\mathbf{F}$?

Incremental Solution Procedure



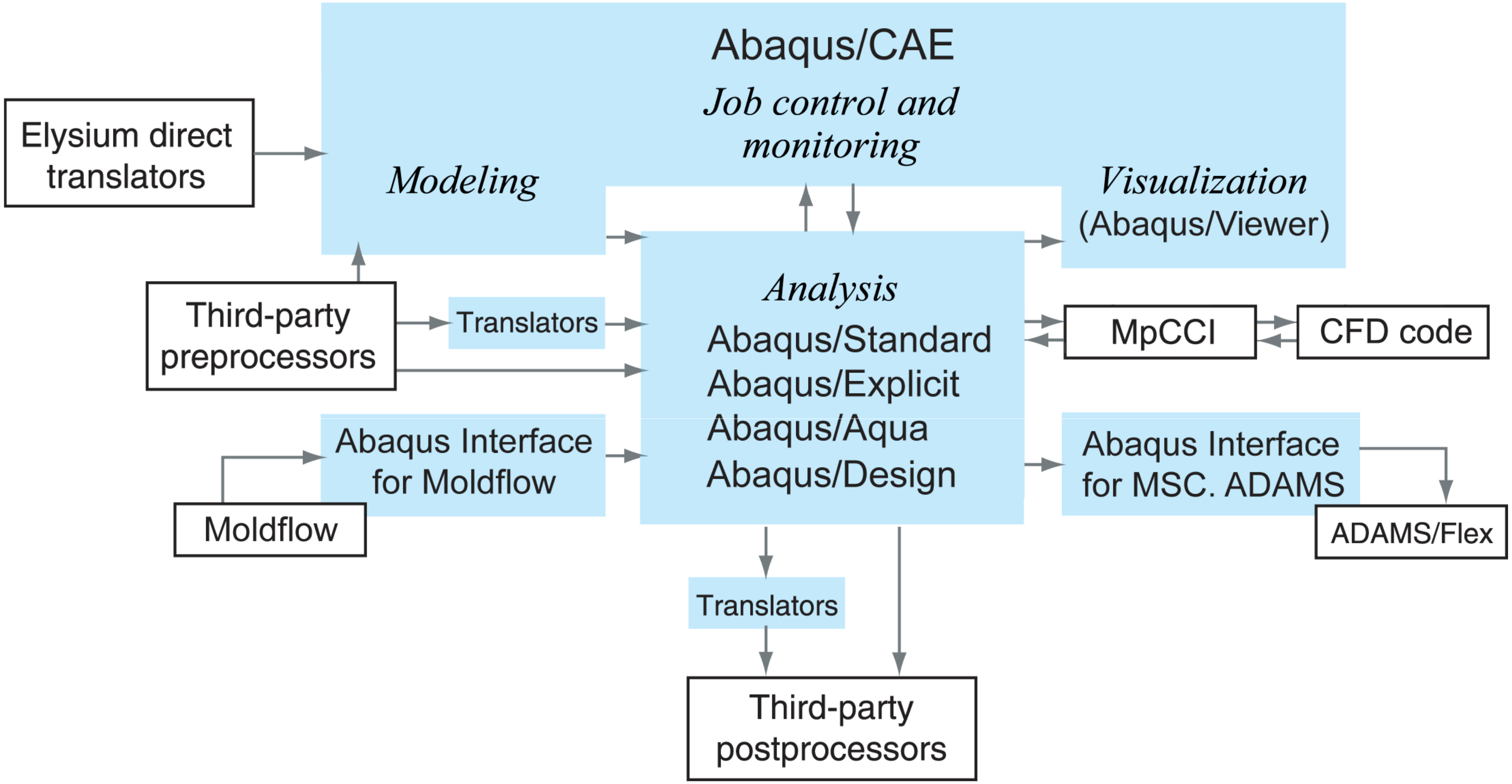
Nonlinearities in Structural Problems



. More than one nonlinearity can exist at the same time

Introduction

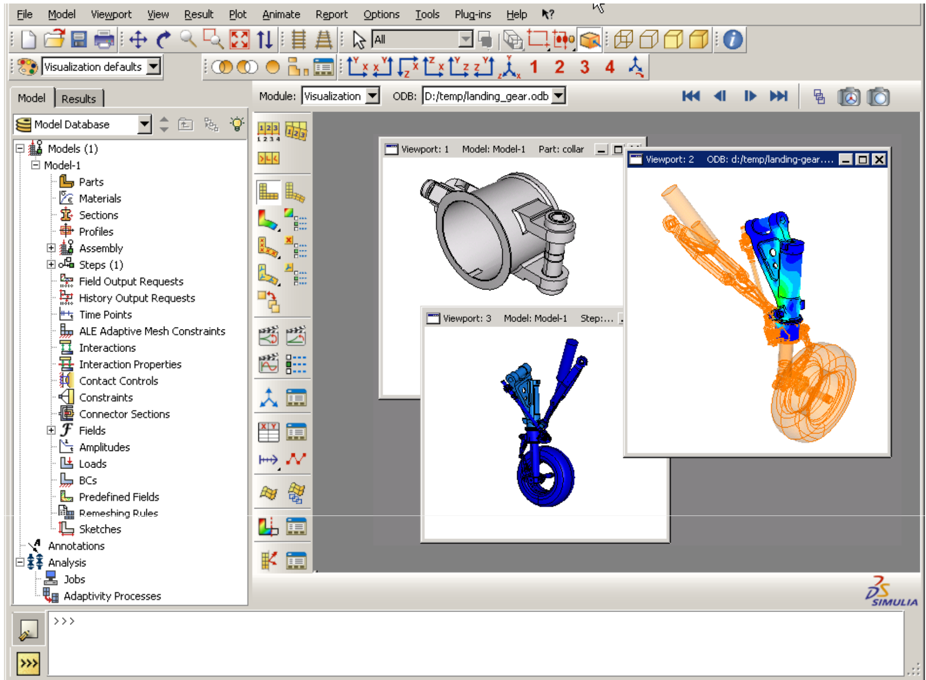
Abaqus FEA is a suite of finite element analysis modules



- SIMULIA Home Page: www.simulia.com
- Abaqus documentation—all usage details are covered in the user's manuals.

Abaqus/CAE

- Complete Abaqus Environment for modeling, managing, and monitoring Abaqus analyses, as well as visualizing results.
- Intuitive and consistent user interface throughout the system.
- Based on the concepts of parts and assemblies of part instances, which are common to many CAD systems.
- Parts can be created within Abaqus/CAE or imported from other systems as geometry (to be meshed in Abaqus/CAE) or as meshes.
- Built-in feature-based parametric modeling system for creating parts.



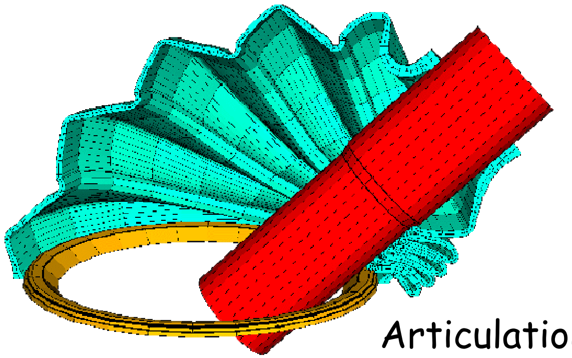
Abaqus/CAE main user interface

Introduction

- Analysis modules
- Abaqus/Standard and Abaqus/Explicit provide the user with two complementary analysis tools.

  Abaqus/Standard's capabilities:

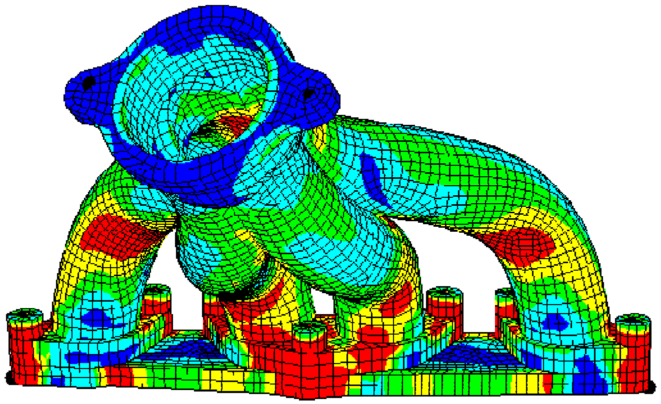
  - General analyses
    - Static stress/displacement analysis:
      - Rate-independent response
      - Rate-dependent (viscoelastic/creep/viscoplastic) response
    - Transient dynamic stress/displacement analysis
    - Transient or steady-state heat transfer analysis
    - Transient or steady-state mass diffusion analysis
    - Steady-state transport analysis



Articulation of an automotive boot seal

Introduction

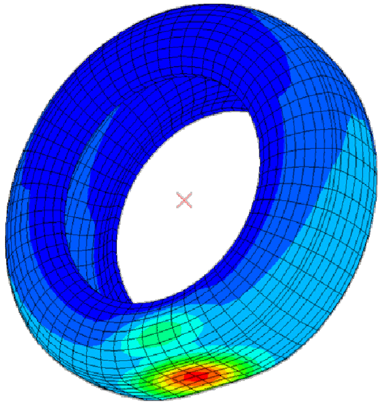
- **Multiphysics:**
  - Thermal-mechanical analysis
  - Structural-acoustic analysis
  - Thermal-electrical (Joule heating) analysis
  - Linear piezoelectric analysis
  - Fully or partially saturated pore fluid flow-deformation
  - Fluid-structure interaction



Thermal stresses in an exhaust manifold

Introduction

- Linear perturbation analyses
  - Static stress/displacement analysis:
    - Linear static stress/displacement analysis
    - Eigenvalue buckling load prediction
  - Dynamic stress/displacement analysis:
    - Determination of natural modes and frequencies
    - Transient response via modal superposition
    - Steady-state response resulting from harmonic loading
      - Includes alternative “subspace projection” method for efficient analysis of large models with frequency-dependent properties (like damping)
    - Response spectrum analysis
    - Dynamic response resulting from random loading

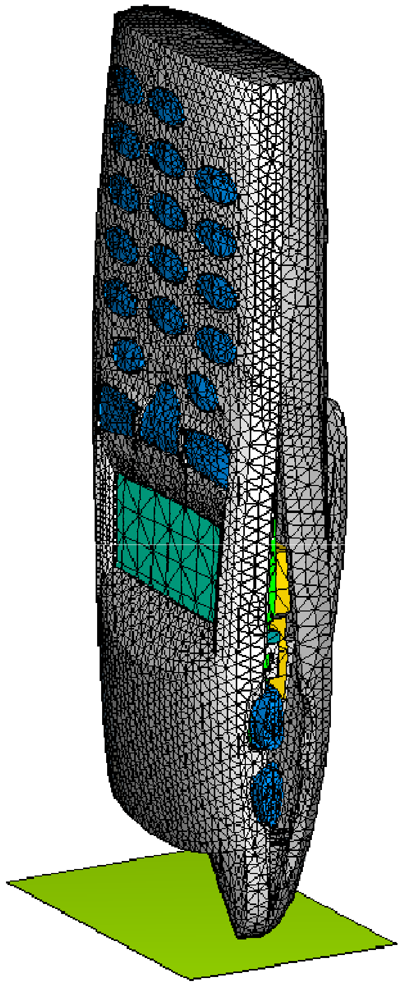


Harmonic excitation of a tire

Introduction

Abaqus/Explicit's capabilities:

- High-speed dynamics
- Quasi-static analysis
- Multiphysics
  - Thermal-mechanical analysis
    - Fully coupled: Explicit algorithms for both the mechanical and thermal responses
    - Can include adiabatic heating effects
  - Structural-acoustic analysis
  - Coupled Eulerian-Lagrangian (CEL)
  - Fluid-structure interaction



Drop test of a cell phone

Introduction

- **Comparing Abaqus/Standard and Abaqus/Explicit**

Abaqus/Standard

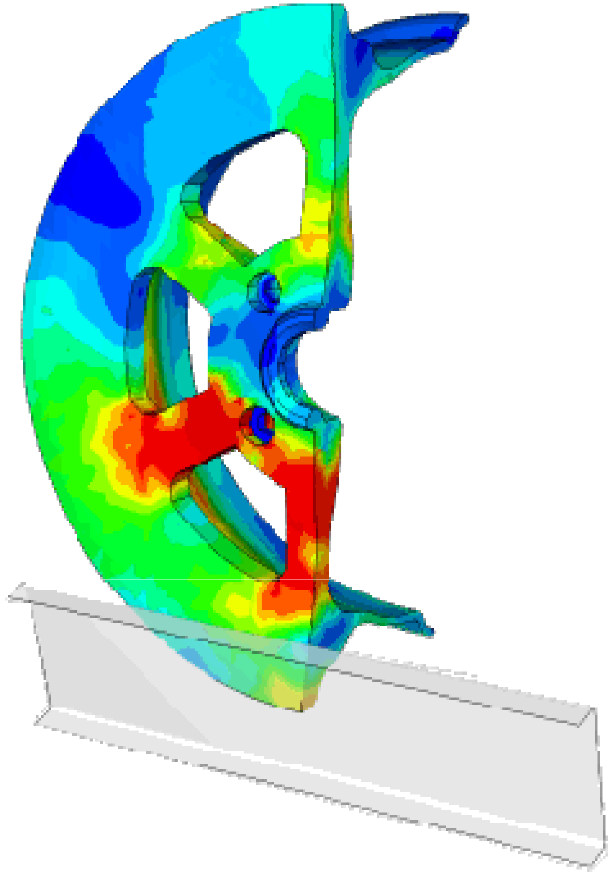
- A general-purpose finite element program.
  - Nonlinear problems require iterations.
- Can solve for true static equilibrium in structural simulations.
- Provides a large number of capabilities for analyzing many different types of problems.
  - Nonstructural applications.
  - Coupled or uncoupled response.

Abaqus/Explicit

- A general-purpose finite element program for explicit dynamics.
  - Solution procedure does not require iteration.
- Solves highly discontinuous high-speed dynamic problems efficiently.
- Coupled-field analyses include:
  - Thermal-mechanical
  - Structural-acoustic
  - CEL

Introduction

- **Interactive postprocessing**
  - Abaqus/Viewer is the postprocessing module of Abaqus/CAE.
    - Available with Abaqus/CAE or as a stand-alone product
    - Can be used to visualize Abaqus results whether or not the model was created in Abaqus/CAE
    - Provides efficient visualization of large models



Contour plot of an aluminum wheel hitting a curb in Abaqus/Viewer

Introduction