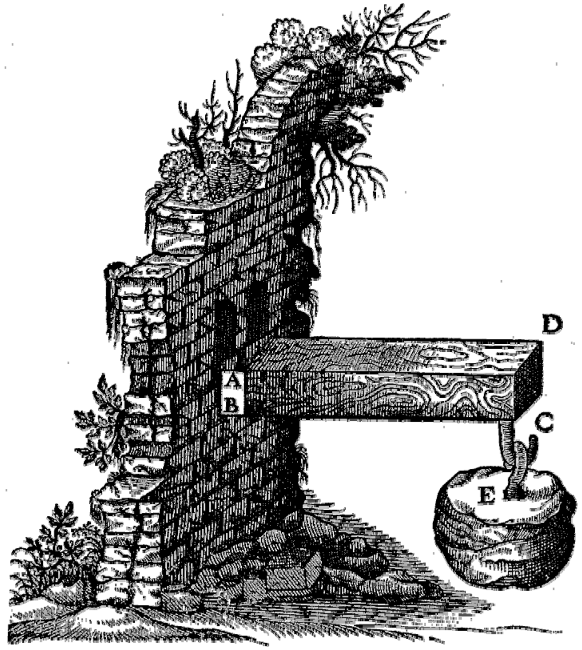
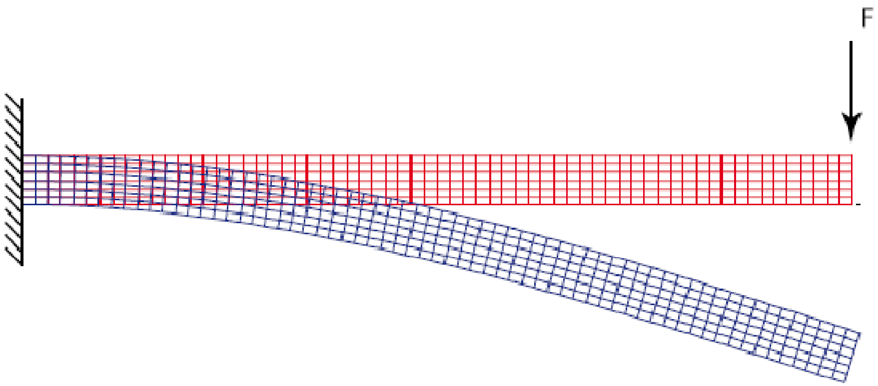
Introduction to FEM-Software





Contents

- FEM development
- Introduction to the use of Finite Elements
- Modelling the physical problem
- Finite elements as a tool for computer-aided design and assessment
- An overview of FEM-Software (Open source and commercial)
- Example: Abaqus

MFE development

- MFE is the confluence of three ingredients: matrix structural analysis, variational approach and computer
- 1950s, M.J. Turner at Boeing (aerospace industry in general): Direct Stiffness Method
- Academia: J.H. Argyris, R.W. Clough (name "finite element"), H.C. Martin and O.C. Zienkiewicz – popularisation
- 1960s, Melosh and De Veubeke: Variational Approach
- Commercial finite element computer codes

Introduction to the Use of Finite Elements

- Within the framework of continuum mechanics dependencies between geometrical and physical quantities are formulated on a differentially small element and than extended to the whole continuum
- As a result we obtain differential, partial differential or integral equations for which, generally, an analytical solution is not available – they have to be solved using some numerical procedure
- MFE is based on the physical discretization of the observed domain, thus reducing the number of the degrees of freedom; moreover the governing equations are, in general, algebraic

Steps in the MFE

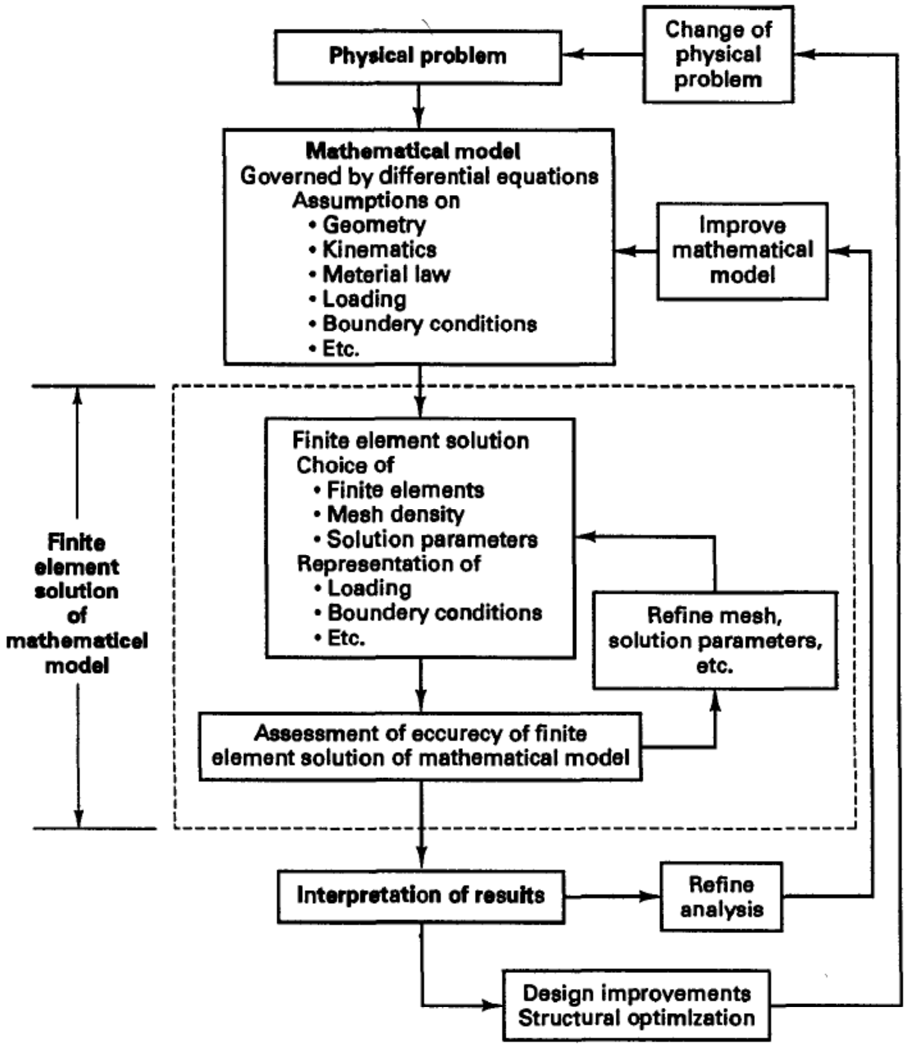
- Continuum is discretized in a mesh of finite elements
- Elements are connected at nodes located on element boundaries
- State of deformation, stresses, etc. in each element is described by interpolation (shape) functions and corresponding values in the nodes; these node values are basic unknowns of the method of finite elements

- The way in which these three steps are approached has a great influence on the results of the calculations

Basic Types of MFE

- Direct MEF: analogue to displacement method
- Variational MFE: based on the principle of stationarity of a functional, which is usually total potential energy or complementary potential energy
- Residual MFE: based on the differential equations that are used to describe the problem
- Energy Balance MFE: based on the balance of different energy types, mostly used to solve thermodynamic problems

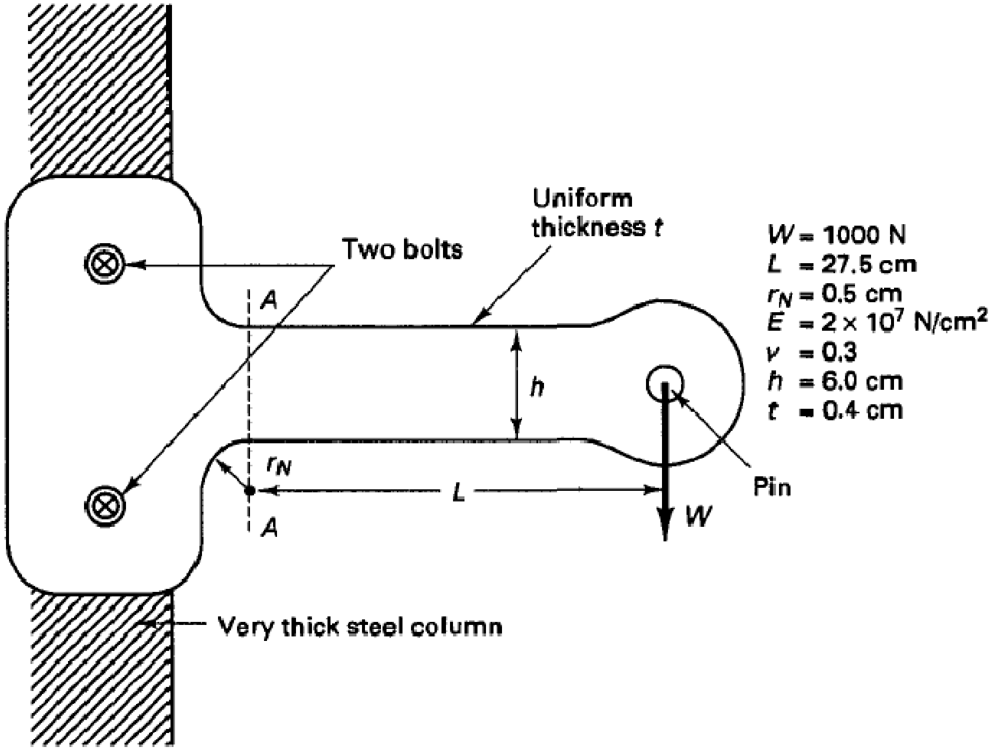
Modelling of the physical problem

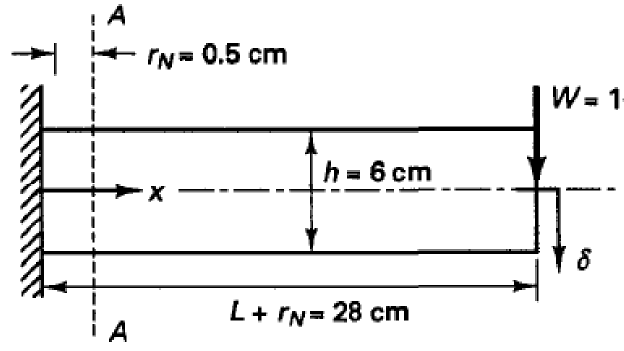


Modelling of the physical problem

- MFE is only a way of solving the mathematical model
- The solution of the physical problem depends on the quality of the mathematical model – the choice of the mathematical model is crucial
- Thus, mathematical model must be reliable and effective
- The chosen mathematical model is **reliable** if the required response can be predicted within a given level of accuracy measured on the response of a very comprehensive mathematical model
- The most **effective** mathematical model for the analysis is the one that gives the required response with sufficient accuracy and at least costs

Example

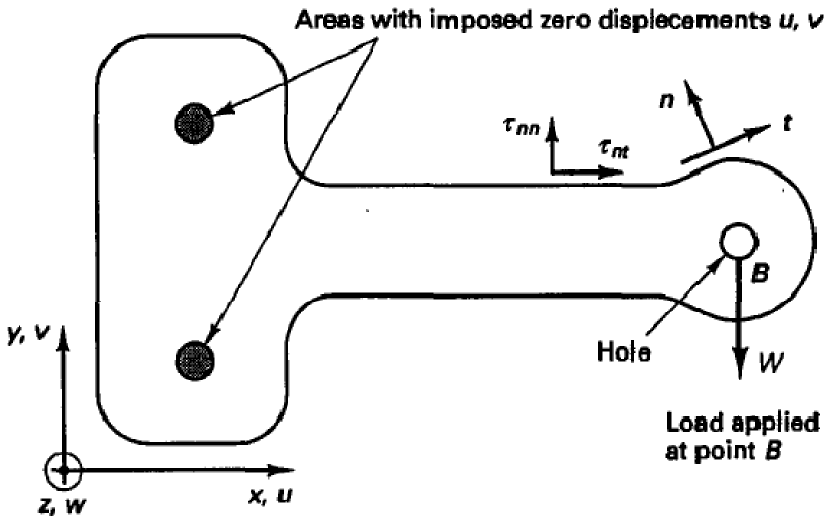




$$M = WL$$
$$= 27{,}500 \text{ N cm}$$
$$\delta\big|_{\text{at load } W} = \frac{1}{3}\frac{W(L + r_N)^3}{EI} + \frac{W(L + r_N)}{\frac{5}{6}AG}$$
$$= 0.053 \text{ cm}$$

Complex physical problem (steel bracket) modelled by a simple mathematical model

Example



$$\delta|_{\text{at load } W} = 0.064 \text{ cm}$$

$$M|_{x=0} = 27{,}500 \text{ N cm}$$

Equilibrium equations (see Example 4.2)

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

$\tau_{nn} = 0$, $\tau_{nt} = 0$ 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):

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

Detailed reference model – 2D plane stress model for MFE analysis

Example

- Choice of mathematical model must correspond to desired response
- The most effective mathematical model delivers reliable answers with the least amount of effort
- Any solution (including MFE) of a mathematical model is limited to information contained in or fed into the model: bad input – bad output (garbage in – garbage out)
- Assessment of accuracy is based on comparisons with the results from very comprehensive models – but in practice it has to be based on experience (experiments…)

MFE as a Tool for CAD/CAM

- Practical application requires that solutions obtained by MFE are reliable and efficient
- However, it is also necessary that the use of MFE is robust – this implies that minor changes in any input to an MFE analysis should not change the response quantity significantly
- Engineer (user) should be able to judge the quality of the obtained results (i.e. for plausiblity)

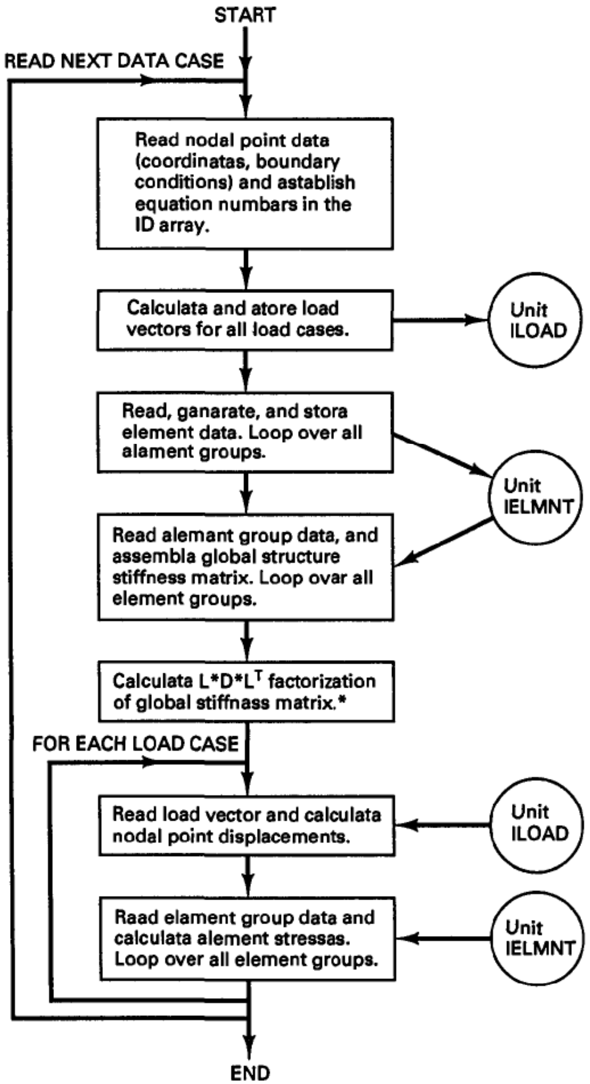
Implementation of MFE

- Calculation of system matrices **K**, **M**, **C** and **R** whichever applicable (nodal point and element information are read; element stiffness matrices, mass and damping matrices and equivalent loads are calculated; structure matrices are assembled)

- Solution of equilibrium equations

- Evaluation of element stresses

Implementation of MFE



- General flowchart of program STRAP (Bathe, page 989)
- Equation solver COLSOL (Bathe, Section 8.2.3)
- Goal: Using efficient finite elements programs
- Packages: Open source and commercial

Open source packages

- CalculiX is an Open Source FEA project. The solver uses a partially compatible ABAQUS file format.
- DUNE, Distributed and Unified Numerics Environment GPL Version 2 with Run-Time Exception
- FEniCS Project: a software package developed by American and European researchers
- ForcePAD: educational software forcepad.sourceforge.net
- freeFEM: a software from freefem.org
- Sundance: a software package developed at Sandia National Laboratories

Commercial packages

- Abaqus: Franco-American software from SIMULIA
- ADINA R&D, Inc. See http://www.adina.com/
- ANSYS: American software
- COMSOL Multiphysics
- COSMOSWorks: A SolidWorks module
- GTSTRUDL30
- LS-DYNA, LSTC - Livermore Software Technology Corporation
- Nastran: American software
- CUBUS

Example: Abaqus

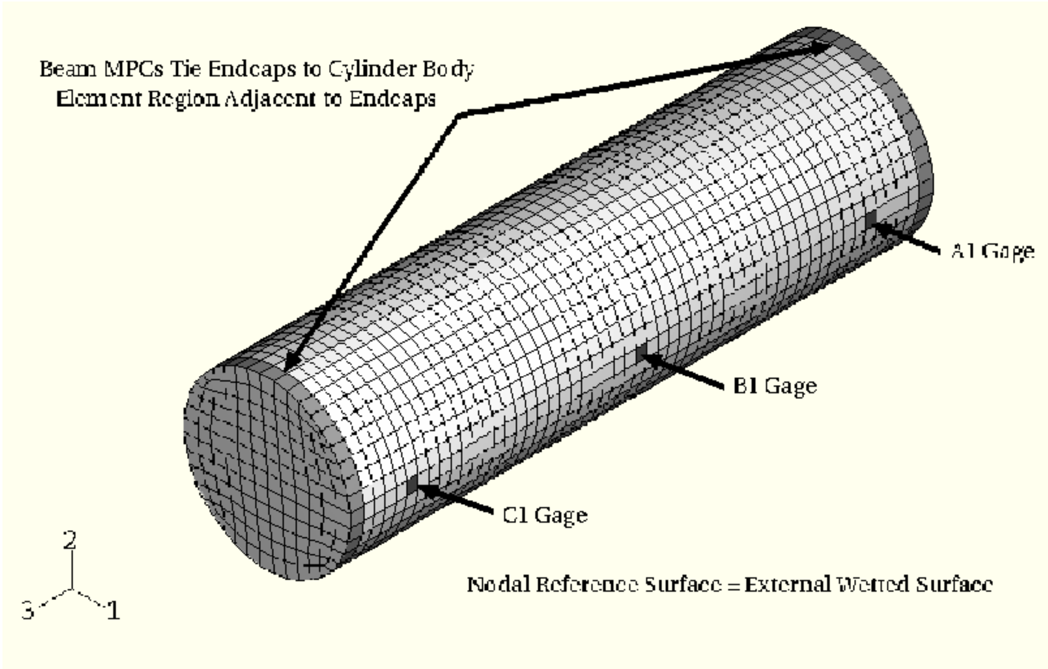
- Abaqus modules: Abaqus/Standard, Abaqus/Explicit and Abaqus/CAE
- Abaqus/Standard is a general-purpose solver using a traditional implicit integration scheme to solve finite element analyses
- Abaqus/Explicit uses an explicit integration scheme to solve highly nonlinear transient dynamic and quasi-static analyses
- Abaqus/CAE provides an integrated modeling (preprocessing) and visualization (postprocessing) environment for the analysis products

Abaqus features

- Wide material modeling capability
- Ability to be customized
- A good collection of multiphysics capabilities, such as coupled acoustic-structural, piezoelectric, and structural-pore capabilities
- Attractive for production-level simulations where multiple fields need to be coupled

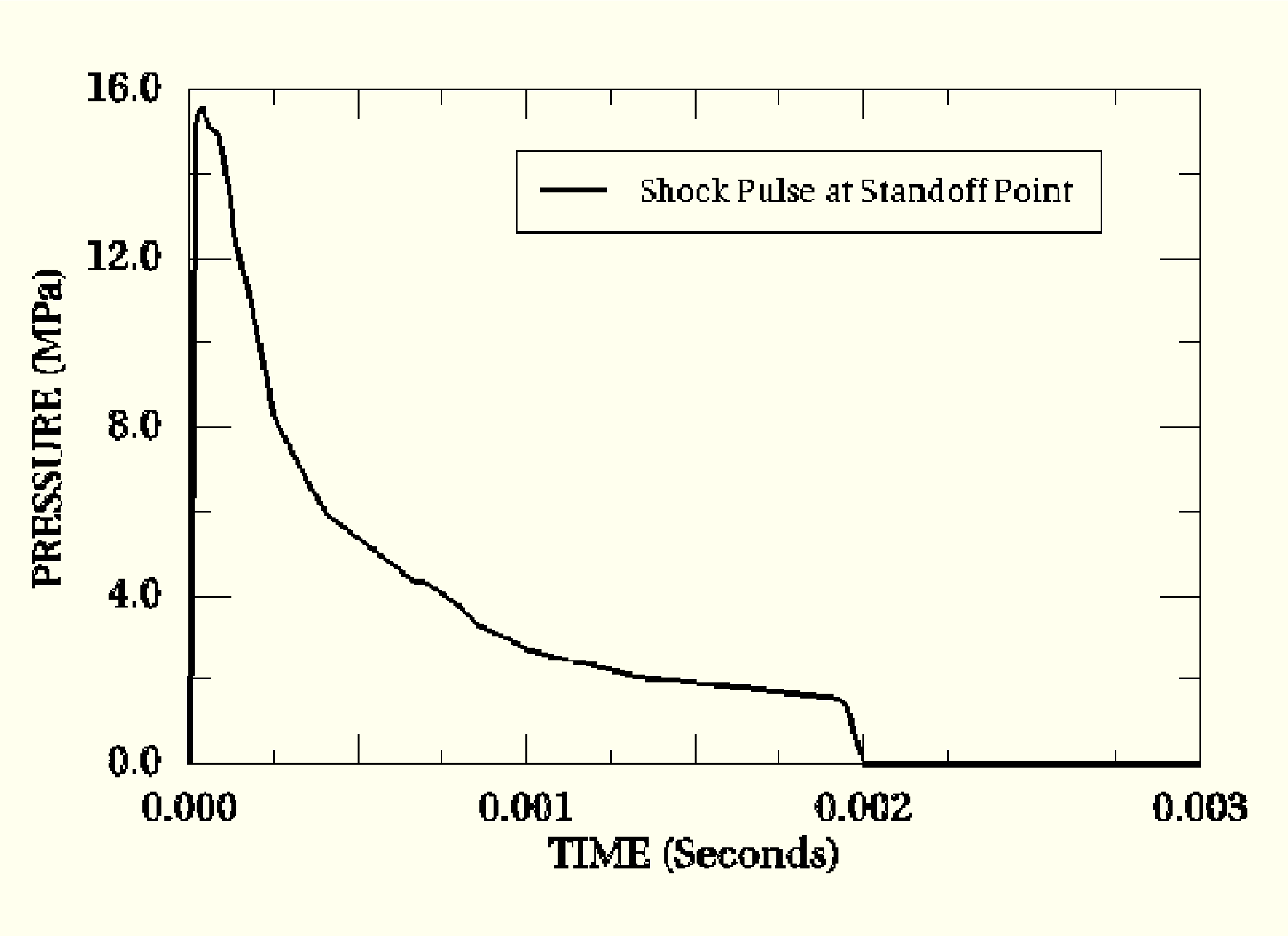
Abaqus: Multiphysics

- Coupled acoustic-structural analysis e.g. blast loading
- Response of a submerged cylinder to an underwater explosion shock wave (Abaqus/Explicit)
- Geometry



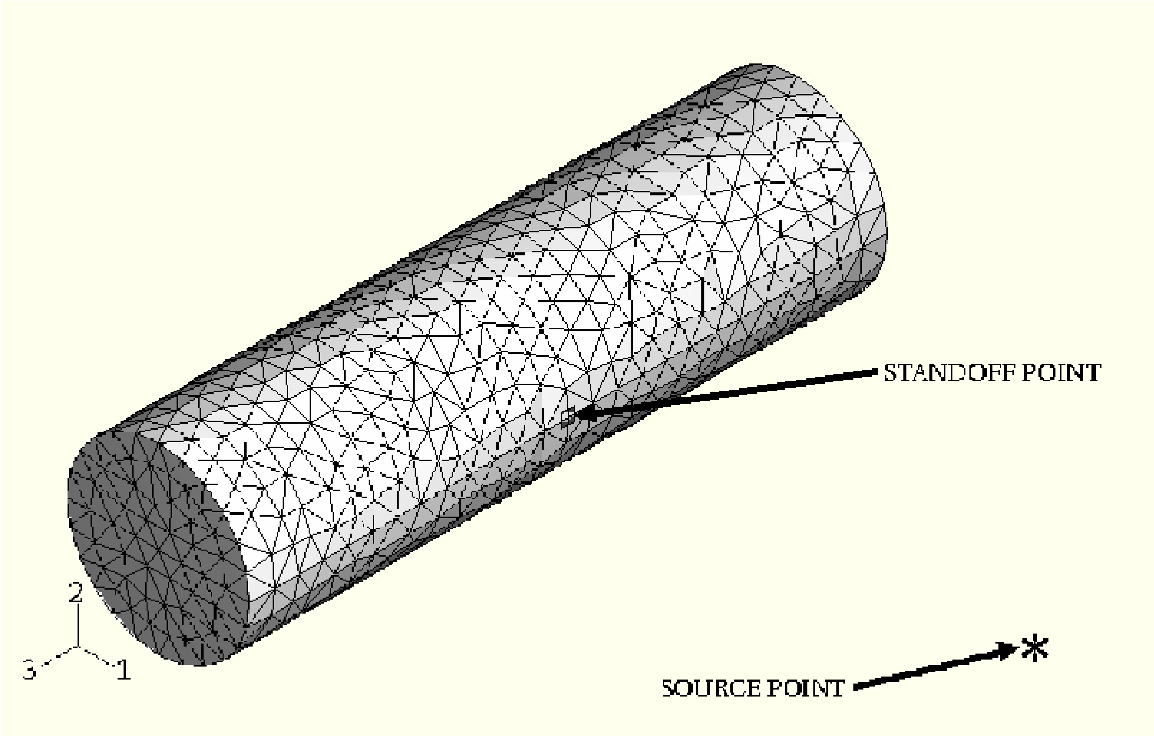
Abaqus: Multiphysics

- Loading



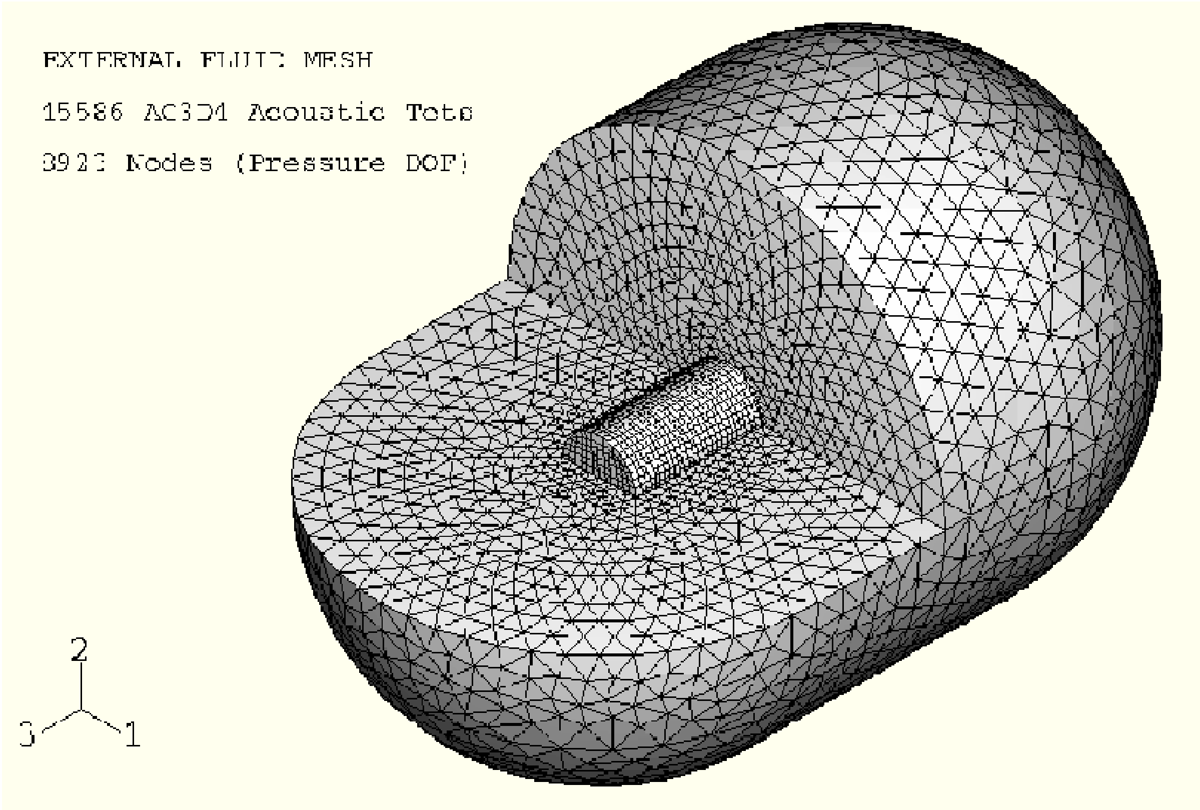
Abaqus: Multiphysics

- Stand-off point and source point



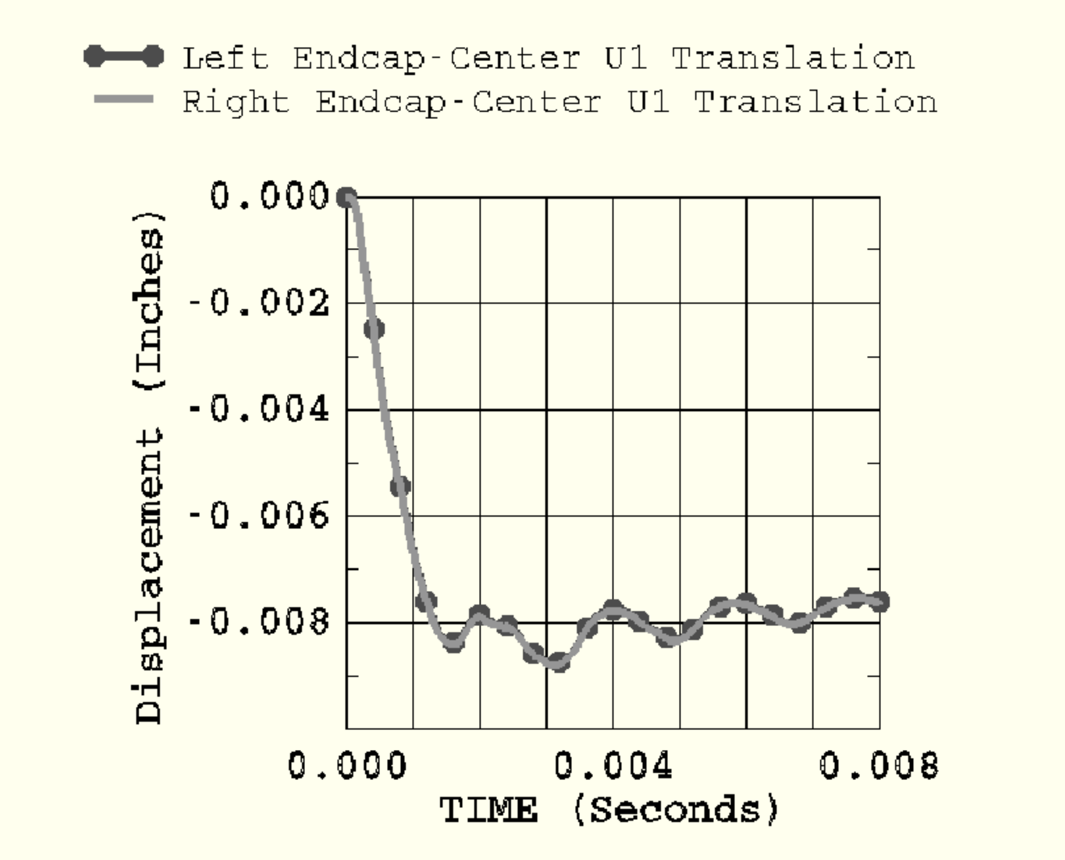
Abaqus: Multiphysics

- Structure and surroundings



Abaqus: Multiphysics

- Displacements at the center of the end caps



Abaqus: Multiphysics

- Accumulated equivalent plastic strains