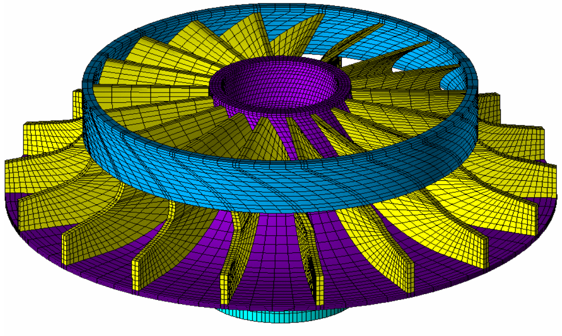
FINITE ELEMENT METHOD

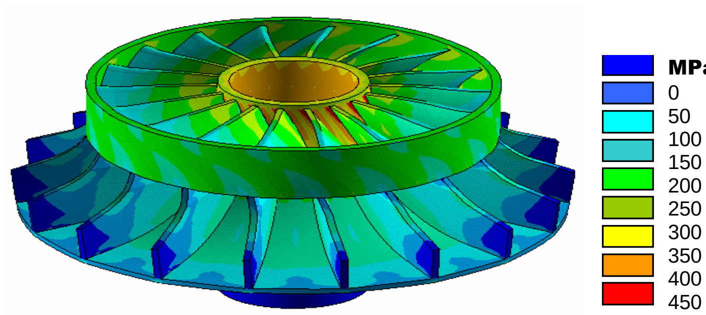
The FEM is a numerical procedure that can be used to solve a large class of engineering problems including mechanics of structures, heat transfer, electromagnetism, fluid flow and coupled fields problems (e.g. electro-thermal).

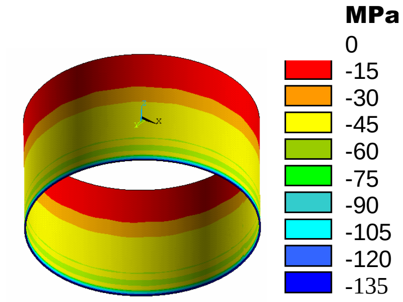
The simplest description.

The method involves dividing the geometrical model of the analysed structure into very small, simple pieces called **finite elements,** connected by nodes. The behaviour of of the element is described by adequate physical laws. An unknown quantity (e.g. temperature, displacement vector, electrical potential) is interpolated over an element from the nodal values using specially defined polynomials (called shape functions). The procedure leads to the set of simultaneus algebraic equations with the nodal values being unknown.

During the solution process the nodal values (DOF- degrees of freedom of the model) are found. Then all interesting quantities (strains, stresses) are calculated within the elements. Finally the results may be presented in the required graphical form (the typical form of presentation is a contour map)

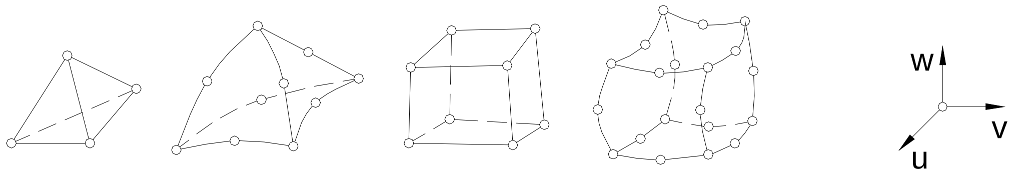






Finite Element Mesh *Von Mises stress distribution* *Contact pressure between the shaft and the rotor disk*

3D finite elements

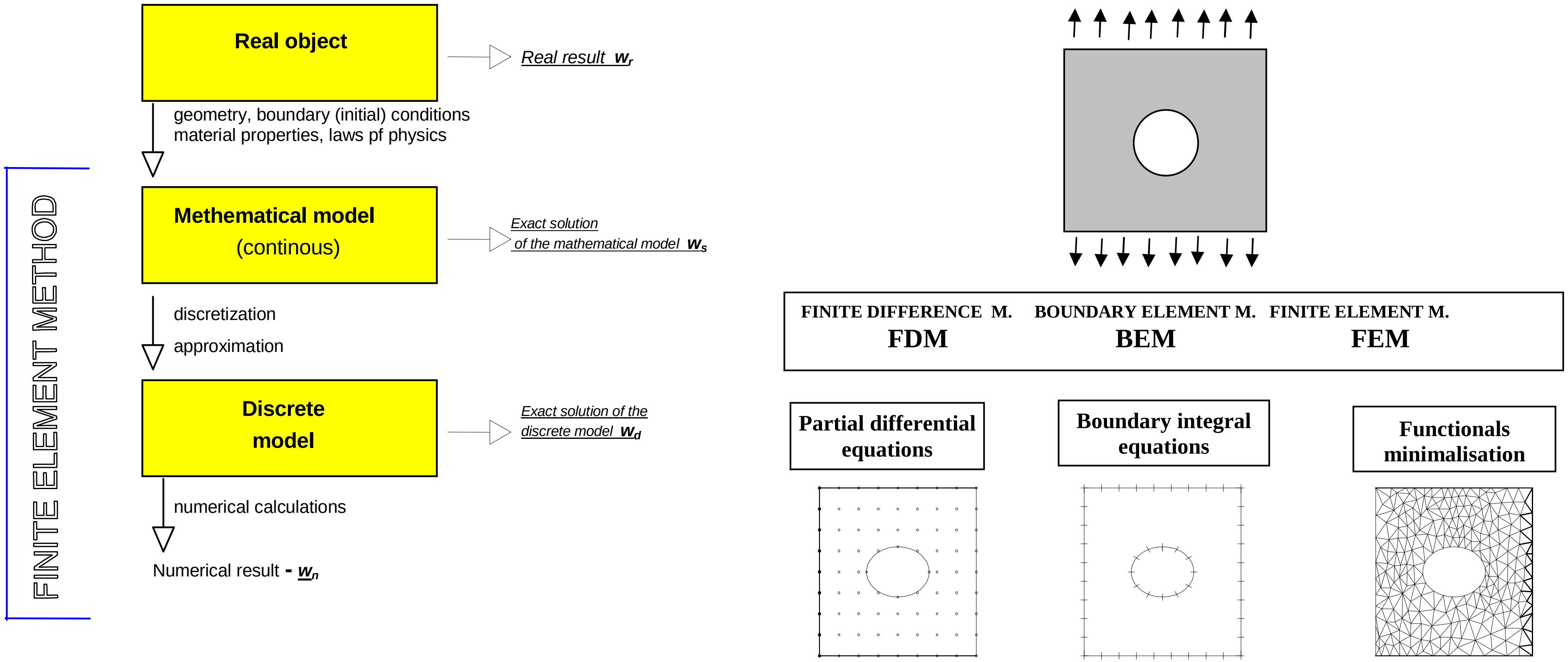


FEM was developed in 1950's for solving complex problems of stress analysis - mainly for aeronautical industry. The development of the method was connected with the progress in digital computers and numerical techniques.

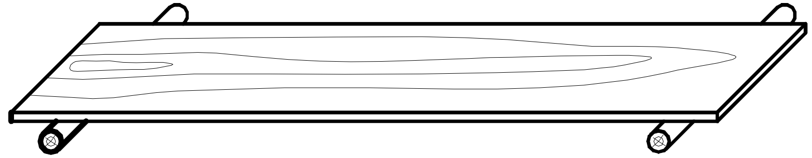
Today the method is considered as the most powerful analysis method for problems described by partial differential equations.

FEM is one of the approximate methods for solving continuous problems of mathematics and physics

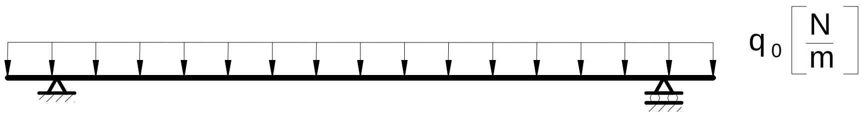
Approximate methods – flow chart



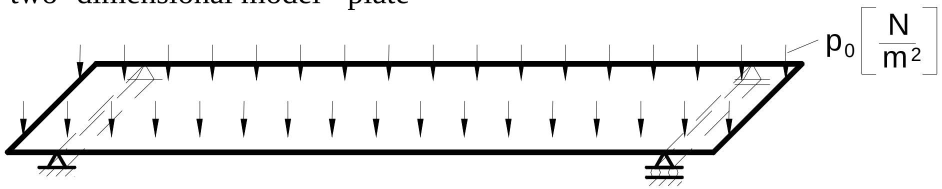
The example – wooden board
Different models for the problem



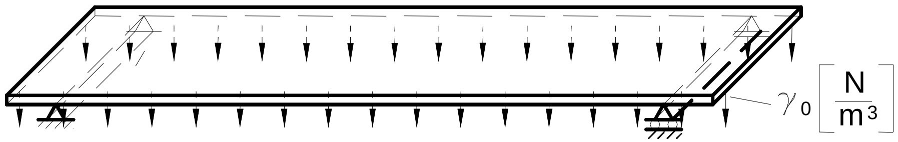
one dimensional model - beam



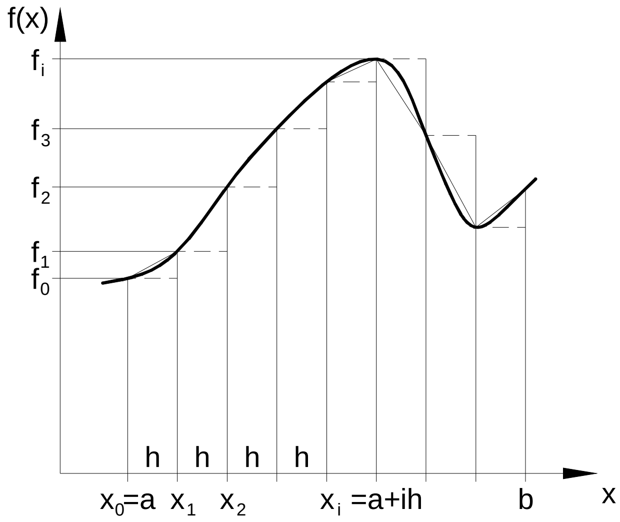
two- dimensional model - plate

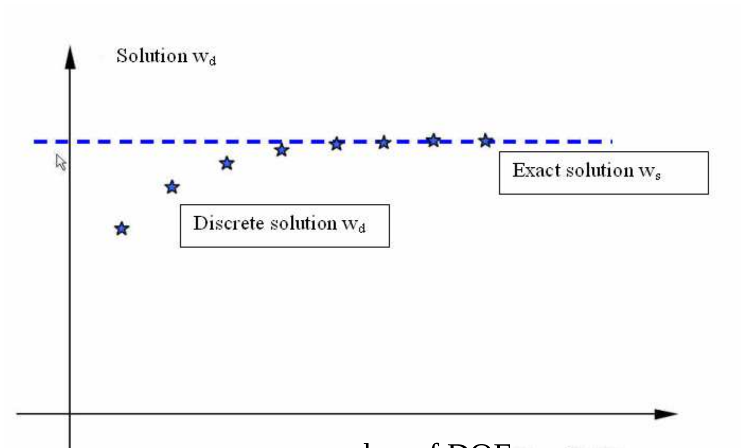


c) three dimensional model – solid volume



Discretization of the continuous problem – numerical estimation of the integral





number of DOF

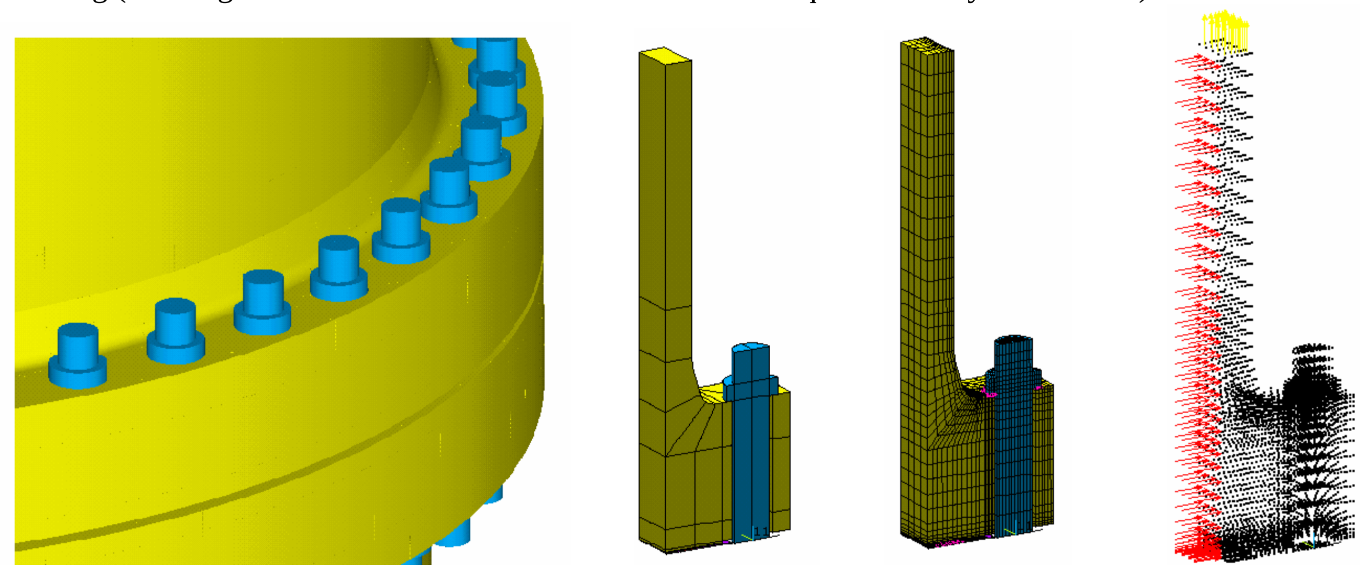
BASIC STEPS IN THE FINITE ELEMENT METHOD (FE modeling)

Preprocessor (preprocessing phase)

In the preprocessing phase the mathematical problem is described and presented in the numerical, discrete form:

Steps:
Description of :

- the analysed domain (geometry of the analysis object)
- the material properties
- the boundary conditions (loads and constraints)
- the meshing (dividing the domain into the finite elements of the required density distribution)



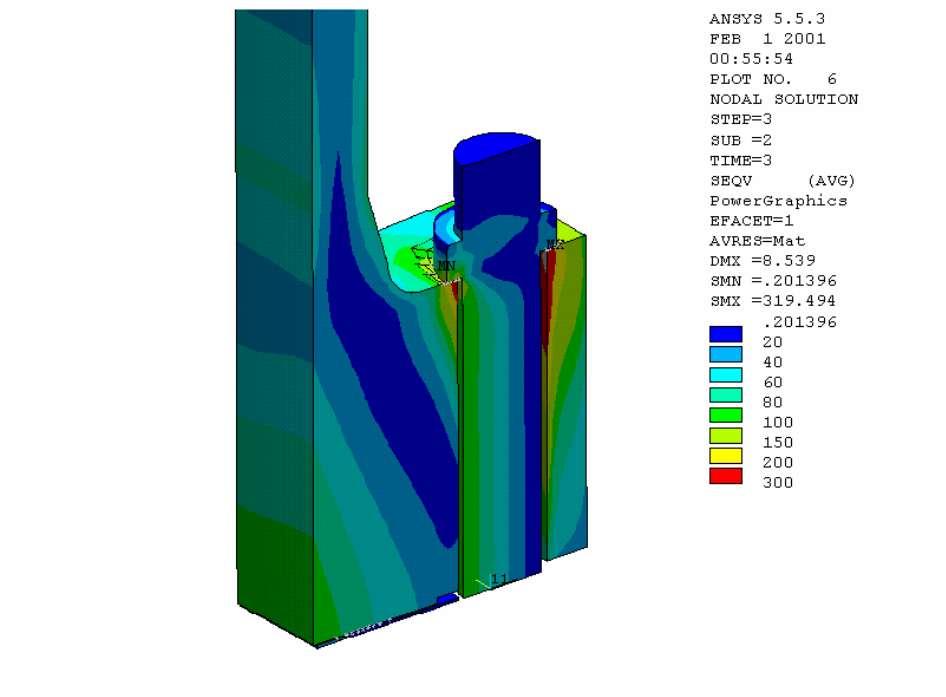
FE model of the bolted joint of the high pressure vessel
entire connection, representative part of the structure and its discretization, FE nodes with load symbols

Processor (solution phase)

In this phase the user of the FE program defines the type of analysis (static, linear or nonlinear, dynamic, buckling ......) and other details describing the method of calculations and solution process.
The FE program performs the calculations and writes the results in the adequate files.

Postprocesor

In this phase it is possible to present the interesting results in different forms: plots, graphs, animations, listings etc.
The user can create contour maps, tables, graphs and generate the reports.

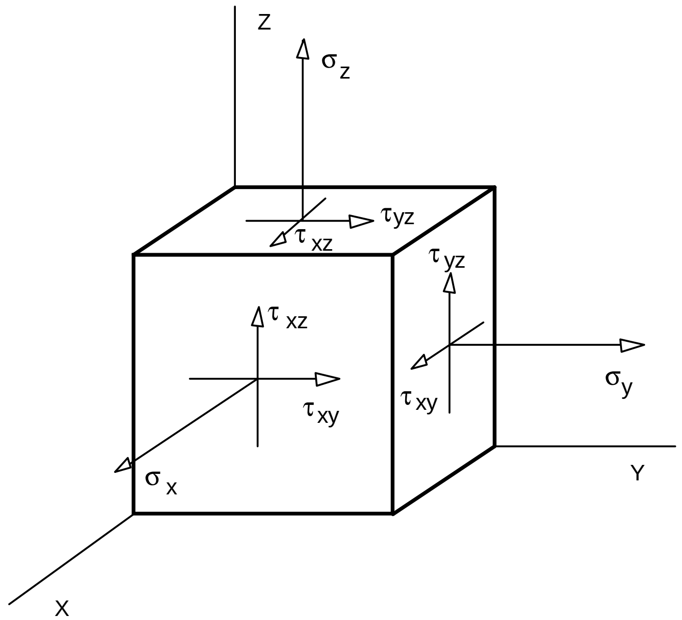


Von Mises equivalent stress distribution (MPa). Contour map

The results of FE analysis

Deformed model compared to undeformed structure

Displacement vector (u_x, u_y , u_z)
Stress state components within the model
$\{\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{xz}\}$



Strain state components
$\{\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{xz}\}$
Principal stresses .

Equivalent stress distribution according to an arbitrary criterion
Any other entity defined by the user (ANSYS – APDL commands)