Aalto University
School of Engineering

Aalto University

School of Engineering

Finite Element Method Basics

Created for

MEC-E1060 - Machine Design

Autumn Term - Period I

2020

Author:
Christoph Spurk

Contents

1 Introduction

The analytical calculation of the stress-deformation and the dynamic or thermal behaviour of machine tool frame components or also entire machines is often impossible due to complex geometry or heterogeneous material. The basic equations for the description of structural-mechanical problems such as deformations, stresses, velocities, pressure, temperatures, etc., are ordinary or partial differential equations or differential equation systems. The solutions of these differential equations have to meet certain boundary conditions. Although for many problems in engineering mechanics the boundary value problem can be formulated in the form of differential equations or differential equation systems, an analytical solution cannot always be found. In these cases, approximate solutions, including FEM, must be used.

The aim of the approximation methods is the transformation of the basic equations which cannot be solved directly into structures which are mathematically easier to handle. In the procedure, according to the FEM, differential equations are transformed into algebraic systems of equations. With the help of FEM calculations e.g. in the development phase, knowledge about the behavior of the components to be examined can be gathered early on. Ideally, the construction of a prototype and the associated metrological investigations can be dispensed with.

According to the current state of the art, the finite element method offers an important tool for the numerical solution of a multitude of technical problems concerning the design of machines. There are various calculation methods for the mathematical treatment of mechanical and thermal problems in mechanical engineering. The basis for all methods is the mathematical formulation of the physical laws.

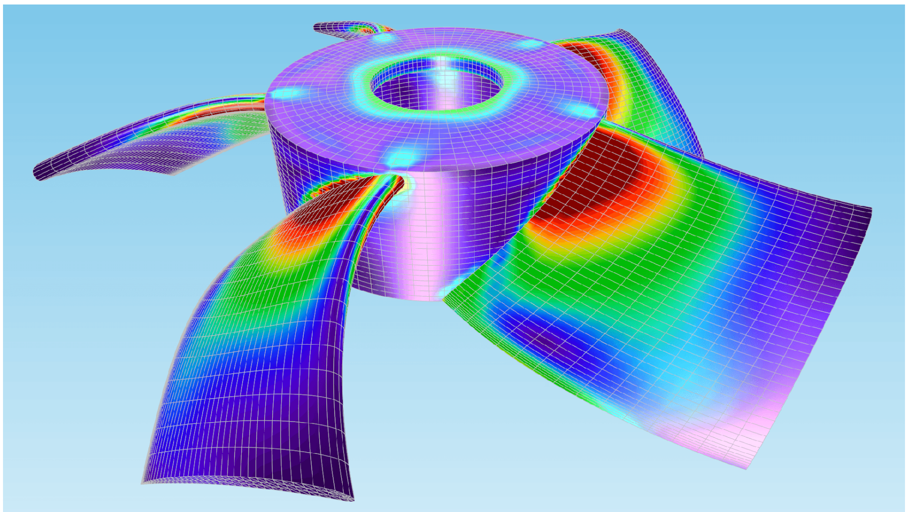


Figure 1: Cooling fan blade vibration [3]

The first step in this document is to explain the basic procedures and the different design levels. Then the material is discussed in order to be able to take its heterogeneity into account. Using simple examples, the calculation steps are explained, which can also be transferred to other element types. In the subsequent section, the relevance of a software-technical solution is clarified. Furthermore, typical sources of error in the analysis are shown (e.g. meshing) and the method is extended by a dynamic approach. Finally a calculation example is given which shows the difference in analytical and numerical approaches for calculating a maximum displacement of a beam.

2 Finite Element Method Basics

Sources: ([4], [5], [8], [9])

2.1 Theoretical basics and general knowledge

For complex problems in the field of continuum mechanics, exact analytical solutions, such as those shown on the right side, using the example of a bending beam or single-mass oscillator, cannot be found, so that approximate solutions must be used to solve them. In the case of approximate solutions, a distinction is made between closed solutions such as Ritz's method, in which a solution is sought over the entire area, and discrete or discretized approximate solutions, in which the problem-describing differential equations are approximated on small (discrete) subareas in order to obtain a linear and numerically solvable system of equations. The Finite Element Method (FEM) is one of these discrete solution methods, and has, in addition to the Boundary Element Method (BEM), established itself as the state of the art in many areas of engineering for structural analysis.

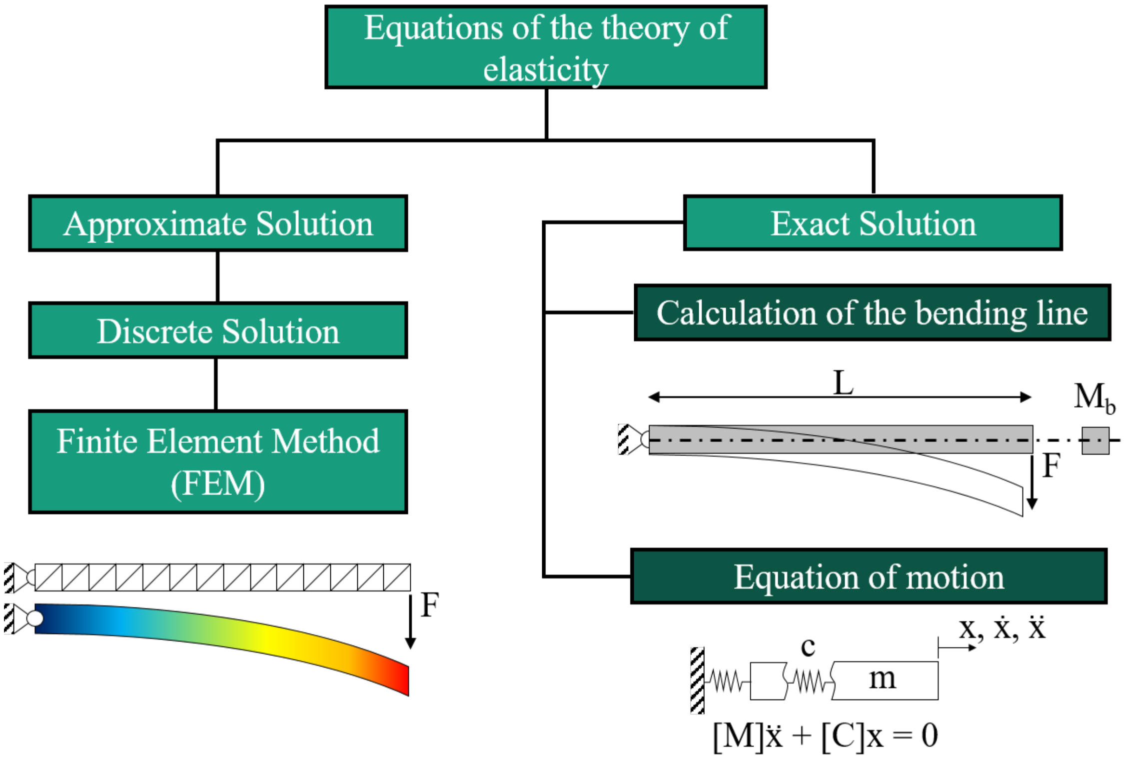


Figure 2: Classification of the Finite Element Method

FEM is a numerical method for solving problems in continuum mechanics. The development of this method is based on findings of Gauss (1795) and Bienzo-Koch (1923), who developed the weighted residuals method. Together with the variation principle of Raleigh (1870) and Ritz (1909), Courant (1943) and Prager-Synge (1947) formulated the first approaches to today's FEM. With the advent of powerful electronic computing systems in the 1970s, FEM finally became an effective computing method for engineering problems.

The approach to FEM can be divided into 6 steps:

1. **Discretization** of the basic geometry in finite elements
2. **Displacement approach** for each finite element
3. **Element stiffness matrix** for each finite element
4. **Global stiffness matrix** of all finite elements
5. **Inserting the boundary conditions** into the global stiffness matrix
6. **Solving** the reduced system of equations

At the beginning, the basic area (i.e. the component, the machine or even the flow area) is divided into finitely large subareas. This is the most important task of the user, as it determines the accuracy of the analysis and the required computing time. The further steps are performed by the system in the backend, i.e. automatically without user intervention. A displacement approach is selected for each individual finite element. Then a stiffness matrix is created for each individual element. The stiffness matrices are merged to form the global stiffness matrix. The boundary conditions (e.g. restraints and forced displacements of degrees of freedom) are inserted into the matrix. The resulting system of equations is solved. These individual steps are explained in more detail below. [9]

2.2 FE analysis specifications

Before starting the model creation, the goal of the calculation should be defined concretely. This will determine whether the calculation of the mechanical behavior should be performed for a component only, an assembly or an entire machine. Frequently, stress states are of particular interest only in highly loaded parts of the geometry, such as force application areas. When calculating a partial structure, a partial mesh is cut out of the geometry. As boundary conditions, nodal restraints in the cut surfaces must then be applied. If individual components are calculated from a complete machine, the properties of the joint must be taken into account. For example, the properties of a guide through a translational degree of freedom of the adjacent mesh nodes can be described (e.g. the mechanical behaviour of a fixed bearing is simulated by a rotational degree of freedom). In order to take the specifications into account ahead of time, the following points should be considered:

- Calculation target
  - Static or dynamic behaviour
  - Deformation behaviour
  - Stress curve
  - Temperature profile
- Boundary Conditions
  - Machine setup
  - Joints
  - Maximum deformations
  - Temperatures
- Component properties
  - Component geometry
  - Inner coupling and joining points
  - Mechanical and thermal material properties

- Loads
  - Forces
  - Moments/Torque
  - Pressures
  - Tensions
  - Heat sources

2.3 Different levels of design

For today's applications in aerospace or automotive engineering, lightweight applications are becoming increasingly important, as they reduce weight and thus, for example, bring with them further cost advantages such as the reduction of fuel consumption. It is therefore important to know exactly about the different materials and their mostly non-homogeneous properties. FEM analysis is used to simulate and analyze homogeneous as well as heterogeneous structures since they work with very clear definitions of the material.
Depending on the simulation case, different philosophies of machine design can make sense. If extremly low life risk is accepted, a **safe life** philosophy is used. This may result in a large structural weight, but the structure will never experience a detectable crack. **Fail safe** means that the structure will not endanger lives or properties when it fails. If the structure may sustain defects until repair can be done, engineers are setting just a **damage tolerance**.

A classical simplification in order to simplify the structural analysis is to divide the response assessment to three levels. The primary, secondary and tertiary levels are visualized in figure 3 using the example of a hull girder.

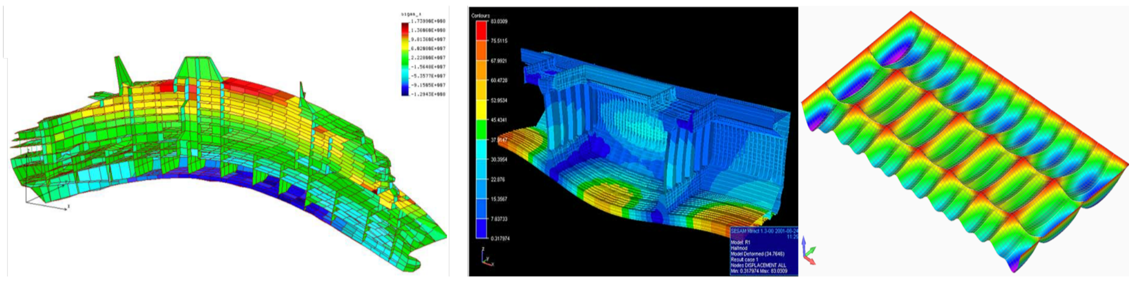


Figure 3: Different levels of design [8]

Level 1 shows only the main elements of the structure which form closed compartments. In the second level, the simulation consists of parts than can deform in larger scale. This level transfers the loads between level 1 and 3. The tertiary level whereas consists only of local parts, which can be assessed using basic beam and plate theory. As the production technology goes on, investigations at "level n" can be performed that analyse the structures also at a material scale.

The results of the analyses can provide information about the limit states which define the load-carrying capability of the structure. These states should be compared to the global deformation of the structure and the operational conditions. The goal is to know the ultimate strength of the structure avoiding e.g. fatigue over time.

2.4 Material properties

In general the selection of materials is a function of many different aspects, which are connected and interdependently. The main aspects are shown in 4.

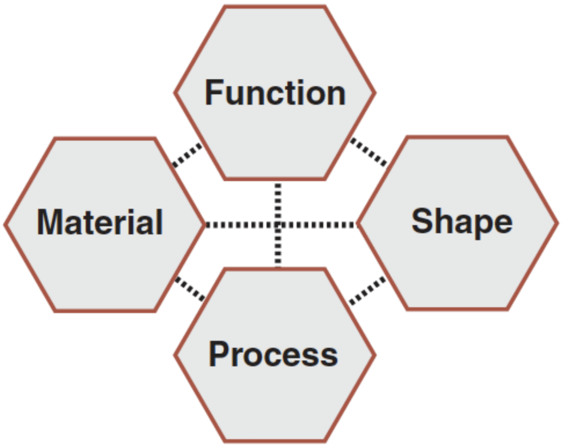


Figure 4: The interaction between function, material, shape and process as the central problem of material selection [1]

The energy absorbed by the specific material volume is is equal to the product of stress and strain. As long as there is plastic capacity in the material that makes the structure safe, engineers often allow material to yield locally. Accordingly, it's important to know the strain-stress behavior of the used material.

Metallic materials, like steel, aluminium, magnesium or titanium are the most common materials for e.g. thin-walled structures due to their availability, processability and price. By composing 2 or more individual materials, which are together stronger than alone, composite materials are createder. The main is to position the material in direction of principle stress. Considering the generalised Hooke's Law for elastic range, there are different ways to model a material in FEM.

- **Orthotropic materials:** three mutually orthogonal material planes
- **Isotropic materials:** no preferred direction of the material
- **Anisotropic materials:** preferred direction of the material

Furthermore, most of the materials have a significant plastic range that can be used in the design for example as safety margin. The plastic behavior is complex and depends on aspects like damage initiation, propagation or thermodynamic effects. These processes have to be modelled through continuum (average stress and strain) in an Representative Volume Element (RVE) of the material. To achieve the highest accuracy, the FEM simulations should start at a molecular (Ab initio) level. However, this kind of analysis brings the problem of high numerically intensive computations and reliable knowledge in the fields of physics and chemistry,

2.5 Derivation of the element stiffness matrix of one tension-compression bar

The elements of an FEM mesh are connected in the nodes, the element corner or end points, to form a complete system. For a more precise approximation curved geometries can also have intermediate nodes. The mechanical behaviour of the entire structure is determined by the discrete nodes. For a mathematical formulation of the stiffness properties, a relationship between the element loads acting in the nodes and the element deformations caused must be established. When using the displacement approach, the relationships between the acting forces and the element deformations are formulated using the node displacements.

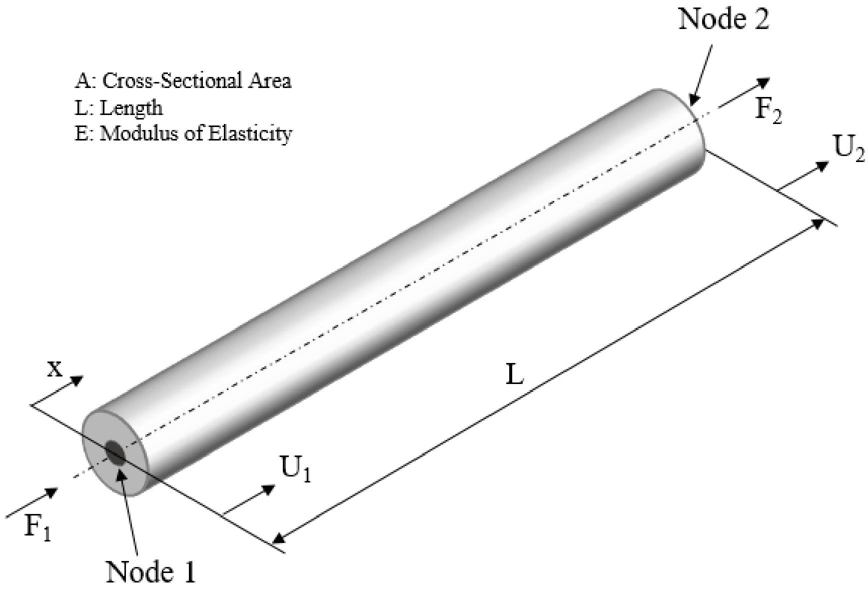


Figure 5: Sketch of the tension-compression beam

In theory, the tension-compression beam can only absorb forces in beam direction and therefore can only deform axially. The deformation of the beam is therefore the result of the displacement of the nodes in beam direction. The introduction of a local coordinate system with one coordinate in member direction simplifies the description of the node displacements. For example the displacement state $u(x)$ between the two nodes i and j is described by a linear polynomial approach with regard to the boundary conditions:

$$u_x(x) = a_0 + a_1 x \qquad u_x(0) = U_1 \qquad u_x(L) = U_2 \tag{1}$$

Combining these equations, it's possible to derive the displacement approach in x-direction for the two nodes 1 and 2 in matrix notation.

$$u_x = \begin{bmatrix} \frac{L-x}{L} & \frac{x}{L} \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} \Rightarrow [a] = \begin{bmatrix} \frac{L-x}{L} & \frac{x}{L} \end{bmatrix} \tag{2}$$

In most cases, however, the stress is also an important target parameter for analysis using FEM. According to Hooke's law, the modulus of elasticity and the strain can be used to calculate the stress in the nodes. To calculate the strain, the derivation of the displacement of a node is used.

$$\sigma_x = [H]\{\epsilon\} = E\frac{\partial u_x}{\partial x} = E \begin{bmatrix} \frac{-1}{L} & \frac{1}{L} \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} \Rightarrow [b] = \begin{bmatrix} \frac{-1}{L} & \frac{1}{L} \end{bmatrix} \tag{3}$$

Furthermore it can be assumed that the tension rod shown is in static equilibrium under the acting forces F_i and F_j (here F_1 and F_2). The principle of virtual work states that the sum of the inner virtual work W_i is equal to the sum of the outer virtual work W_o. The external work results from the external forces and the displacements caused. The vectors of this equation include the displacements of the element nodes and the internal and external forces acting on the nodes. The internal work comprises the virtual energy stored in the material from the product of the stresses and strains. It is formulated as an integral over the entire volume. With the strain-displacement relationship, the strain $\{\epsilon\}$ can be replaced by the displacements U_1 and U_2 relative to the element length L. The displacement matrix $[b]$ (in this case only a vector) contains the geometric information of each element. The strain vector $\{e\}^T$ is eliminated from the integral by reference to the stress-strain relationship. The elasticity matrix $[H]$, which contains the direction-dependent material elasticities, is simplified for the tensile member to the Young's modulus. Finally, if the integral is resolved via the body into its two components (the end points of the rod), the form of the linear system of equations $[K]\{U\} = \{F\}$ is created, which has to be solved afterwards. In formula notation, the applied principle of work is as follows:

$$\delta W_i = \int \delta\{\epsilon\}^T\{\sigma\}dV = \int \delta\{U\}^T[b]^T[H][b]\{U\}dV \tag{4}$$

$$\Rightarrow \{F\} = \int [b]^T[H][b]dV\{U\} \Rightarrow \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} \tag{5}$$

2.6 Derivation of the stiffness matrix of a bar structure

When calculating a bar structure, which consists of 2 or more individual elements, the manual calculation becomes much more difficult. Accordingly, to get the displacements in the nodes of the structure, the element equation systems are superimposed for each member. This can be demonstrated using the simple structure of three bar elements shown in figure 6.

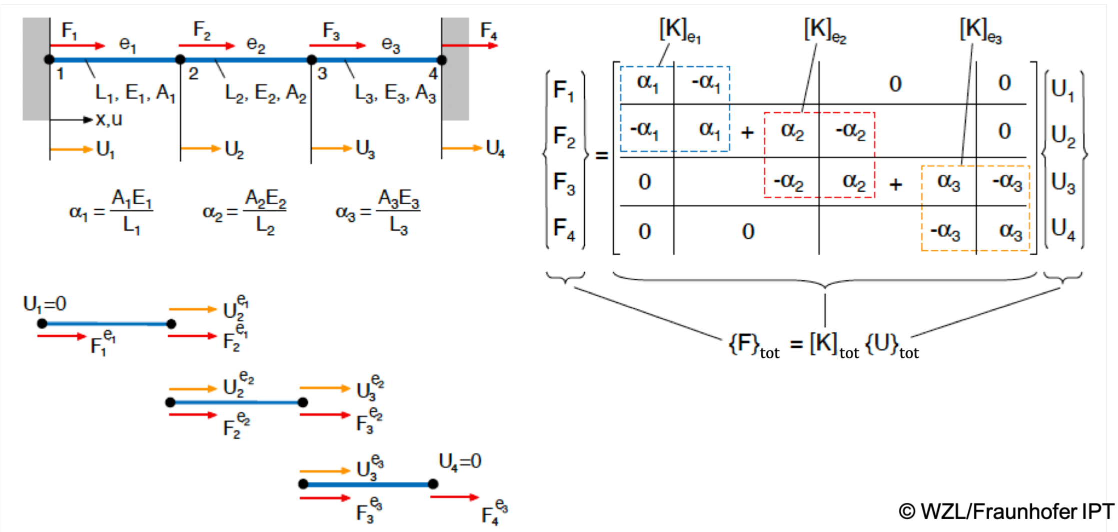


Figure 6: Combination of three tension-compression bars [9]

The three elements e_1, e_2 and e_3 are connected to each other in nodes 2 and 3 and to a solid foundation in nodes 1 and 4. The geometry of the structure, the material properties (Young's modulus) and the external forces F_2 and F_3 are known. The goal is to get the displacements in the nodes 2 and 3. The individual steps here are:

- Creation of the force-displacement relationships of the individual elements
- Superposition of the individual relationships to the force-displacement relationship of the bar system
- Determination of the displacement quantities by solving the equation system

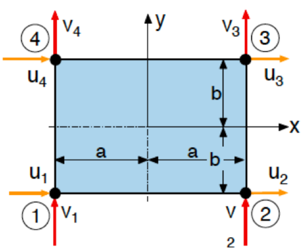
The compatibility condition states that the node displacements of neighboring elements in the nodes are equal. For example, element e_1 has the same displacement in node 2 as element e_2. The equilibrium conditions are established for the individual nodes. The sum of the external forces and the internal forces is set to zero for a static calculation in the nodes. This system of equations provides the starting point for the overall equation system of the structure.

The element forces are expressed by inserting the spring equations and the node displacements. For simplification, the spring constants of the individual elements are combined to α_i. After multiplication the system of equations is obtained in matrix form. The matrix shows the individual stiffness matrices of the elements, which are combined to form the overall stiffness matrix. This gives the matrix a typical band structure. The bandwidth of the total stiffness matrix directly influences the number of computational operations to be performed and therefore the time required for the calculation. The greater the bandwidth of the matrix, the more complex the calculation. Today's commercial FEM programs therefore usually have algorithms that automatically minimize the bandwidth of the total stiffness matrix. For this purpose the nodes of the FEM model are renumbered, which is equivalent to exchanging rows or columns of the matrix. Taking the boundary conditions into account ($U_1 = 0$ and $U_4 = 0$), the system can be solved for U_2 and U_3. [9]

2.7 Stiffness matrix for a square plate element

The procedure for the establishment of the overall equation system and its solution is automatically processed in an FEM program during a calculation run. Therefore it is not necessarily noticeable which work the FEM software does for the analysis. In practical problems, these systems of equations can contain several hundred thousand equations. All data of the model must be fed into the program as inputs. This includes the element data consisting of the node coordinates, node-element assignments and the material properties as well as the loads and the boundary conditions.

The stiffness matrix of a four-node disk element can be used to illustrate the disproportionately increasing complexity of the calculation effort with increasing element size. In the figure shown below (figure 7), there are two degrees of freedom for the displacement of each node. These degrees of freedom are translatory degrees of freedom in the element plane. This means that the element can also be loaded and stretched only in the element plane and has no stiffness properties in normal direction. 8 parameters are combined in the force and deformation vector. The element stiffness matrix is an $8x8$ matrix. The elements of the stiffness matrix contain the geometrical quantities and the transverse contraction number ν.



$$\boxed{\{F\} = [K]\,\{U\}}$$

Force Vector $\{F\}^T = (F_{1x}, F_{1y}, \dots, F_{4x}, F_{4y})$

Displacement Vector $\{U\}^T = (u_1, v_1, \dots, u_4, v_4)$

$$[K] = \frac{Et}{12(1-\upsilon^2)}
\begin{array}{c}
\begin{array}{cccccccc} (u_1) & (v_1) & (u_2) & (v_2) & (u_3) & (v_3) & (u_4) & (v_4) \end{array} \\
\begin{bmatrix}
4\frac{b}{a}+2(1-\nu)\frac{a}{b} & \frac{3}{2}(1+\nu) & -4\frac{b}{a}+(1-\nu)\frac{a}{b} & -\frac{3}{2}(1-3\nu) & -2\frac{b}{a}-(1-\nu)\frac{a}{b} & -\frac{3}{2}(1+\nu) & 2\frac{b}{a}-2(1-\nu)\frac{a}{b} & \frac{3}{2}(1-3\nu) \\
\frac{3}{2}(1+\nu) & 4\frac{a}{b}+2(1-\nu)\frac{b}{a} & \frac{3}{2}(1-3\nu) & 2\frac{a}{b}-2(1-\nu)\frac{b}{a} & -\frac{3}{2}(1+\nu) & -2\frac{a}{b}-(1-\nu)\frac{b}{a} & -\frac{3}{2}(1-3\nu) & -4\frac{a}{b}+(1-\nu)\frac{b}{a} \\
-4\frac{b}{a}+(1-\nu)\frac{a}{b} & \frac{3}{2}(1-3\nu) & 4\frac{b}{a}+2(1-\nu)\frac{a}{b} & -\frac{3}{2}(1+\nu) & 2\frac{b}{a}-2(1-\nu)\frac{a}{b} & -\frac{3}{2}(1-3\nu) & -2\frac{b}{a}-(1-\nu)\frac{a}{b} & \frac{3}{2}(1+\nu) \\
-\frac{3}{2}(1-3\nu) & 2\frac{a}{b}-2(1-\nu)\frac{b}{a} & -\frac{3}{2}(1+\nu) & 4\frac{a}{b}+2(1-\nu)\frac{a}{b} & \frac{3}{2}(1-3\nu) & -4\frac{a}{b}+(1-\nu)\frac{b}{a} & \frac{3}{2}(1+\nu) & -2\frac{a}{b}-(1-\nu)\frac{b}{a} \\
-2\frac{b}{a}-(1-\nu)\frac{a}{b} & -\frac{3}{2}(1+\nu) & 2\frac{b}{a}-2(1-\nu)\frac{a}{b} & \frac{3}{2}(1-3\nu) & 4\frac{b}{a}+2(1-\nu)\frac{a}{b} & \frac{3}{2}(1+\nu) & -4\frac{b}{a}+(1-\nu)\frac{a}{b} & -\frac{3}{2}(1-3\nu) \\
-\frac{3}{2}(1+\nu) & -2\frac{a}{b}-(1-\nu)\frac{b}{a} & -\frac{3}{2}(1-3\nu) & -4\frac{a}{b}+(1-\nu)\frac{b}{a} & \frac{3}{2}(1+\nu) & 4\frac{a}{b}+2(1-\nu)\frac{b}{a} & \frac{3}{2}(1-3\nu) & 2\frac{a}{b}-2(1-\nu)\frac{b}{a} \\
2\frac{b}{a}-2(1-\nu)\frac{a}{b} & -\frac{3}{2}(1-3\nu) & -2\frac{b}{a}-(1-\nu)\frac{a}{b} & \frac{3}{2}(1+\nu) & -4\frac{b}{a}+(1-\nu)\frac{a}{b} & \frac{3}{2}(1-3\nu) & 4\frac{b}{a}+2(1-\nu)\frac{a}{b} & -\frac{3}{2}(1+\nu) \\
\frac{3}{2}(1-3\nu) & -4\frac{a}{b}+(1-\nu)\frac{b}{a} & \frac{3}{2}(1+\nu) & -2\frac{a}{b}-(1-\nu)\frac{b}{a} & -\frac{3}{2}(1-3\nu) & 2\frac{a}{b}-2(1-\nu)\frac{b}{a} & -\frac{3}{2}(1+\nu) & 4\frac{a}{b}+2(1-\nu)\frac{b}{a}
\end{bmatrix}
\end{array}$$

Figure 7: Stiffness matrix for a square plate element [9]

It can be seen that even an apparently simple plate element produces a relatively large matrix. In reality however, a machine is not only approximated by one element. The FEM software has to solve a large number of matrices, which depend on the desired accuracy of the analysis. Furthermore, one should keep in mind that elements normally have not only 2 degrees of freedom per node. In this case, the third dimension and the rotational degrees of freedom were neglected, which would have only unnecessarily complicated the calculation effort in this example. Depending on the type of elements used in the FEM analysis (bar, beam, shell, disk, plate, shell, ...) the analysis time can be reduced by using the appropriate approximations for the application.

2.8 Typical elements for mechanical FEM calculations

The simplest element types are the one-dimensional elements, the bar and the beam element. The **bar** element already considered can only support loads in the bar direction and therefore has only one degree of freedom per node. The *beam* element can support any loads. Each node has 6 degrees of freedom, the translatory (u, v, w) and the rotational (j_x, j_y, j_z).

Planar components, such as fuselage and wing parts of an airplane or frame components of machine tools can be described with plane 2-dimensional elements. With these elements, a distinction must be made between disc, plate and shell elements. The elements can have a triangular or square shape. Depending on the attachment function (linear, parabolic, cubic) the elements still have intermediate nodes on the edges, resulting in elements with 4, 8, 12 nodes or 3, 6 or 9 nodes. The **disc** element can only support loads in the element plane. It has three degrees of freedom per node (x, y, j_z). **Plate** elements, on the other hand, can only support loads perpendicular to the plane and moments about the x or y axis. These elements also have 3 degrees of freedom per node (z, jx, jy). **Shell** elements are obtained by the superposition of disk and plate elements. These elements can be loaded in all directions both translatory and rotational. This results in 6 degrees of freedom per node. Compact or thick-walled components must be described with solid elements that have three degrees of freedom (u, v, w) per node. Rotations can be represented here by translating several nodes.

2.9 Analysis of the static and dynamic behaviour using FEM

The finite element description of a structure includes the finite elements as well as given loads and restraints. The basis of a mathematical description of the problem are the kinetic and kinematic equations of mechanics. The static equilibrium of a structure is described by equating internal and external forces. After the discretization of the problem in finite elements, the equation system to be solved is

$$[K]\{U\} = \{F\} \tag{6}$$

In the case of dynamic equilibrium, the influence of inertial forces and damping is added. Thus the system of equations to be solved is:

$$[M]\{\ddot{U}(t)\} + [C]\{\dot{U}(t)\} + [K]\{U(t)\} = \{F(t)\} \tag{7}$$

The mass matrix, the damping matrix and the stiffness matrix are calculated by the computer on the basis of the finite element structure. The vector of the outer forces have to be specified by the user. By transforming this equation in the frequency domain, it's possible to perform e.g. an eigenfrequency or buckling analysis of a given structure. As an simulation example in the FEM software FEMAP, figure 8 shows the displacement caused by an eigenfrequency oscillation of an simple stiffened plate at $1341,5Hz$. This allows the engineer to design his component in such a way that it cannot reach a critical vibration frequency, which could, for example, lead to the destruction of the machine.

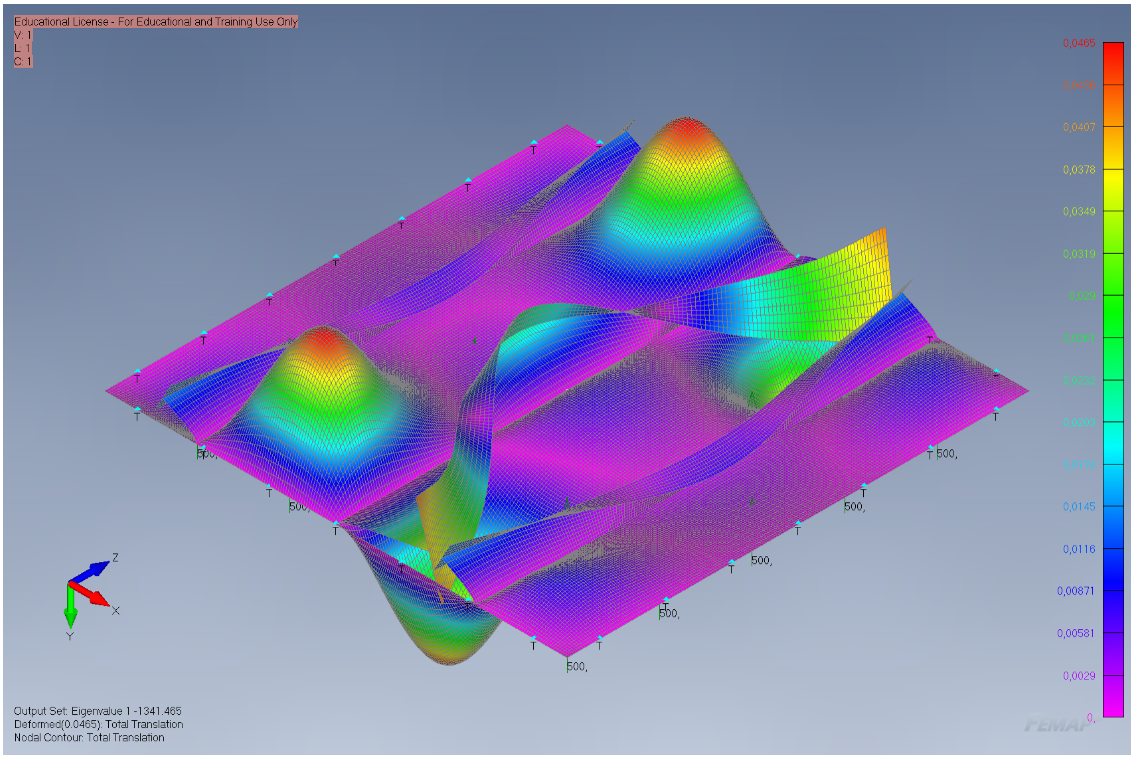


Figure 8: Eigenfrequency analysis of a stiffened plate at $1341,5Hz$ in FEMAP

2.10 Meshing and meshing failures

As we already learned are different geometries (e.g. triangular elements) used for meshing a structure. In chapter 2.13 is given an example where a beam is meshed very coarse with only 3 beam elements but we get the exact results like the analytical approach. However, the following example in FEMAP shows that the mesh has a significant influence on the final result of the simulation if the structure is rather complex.

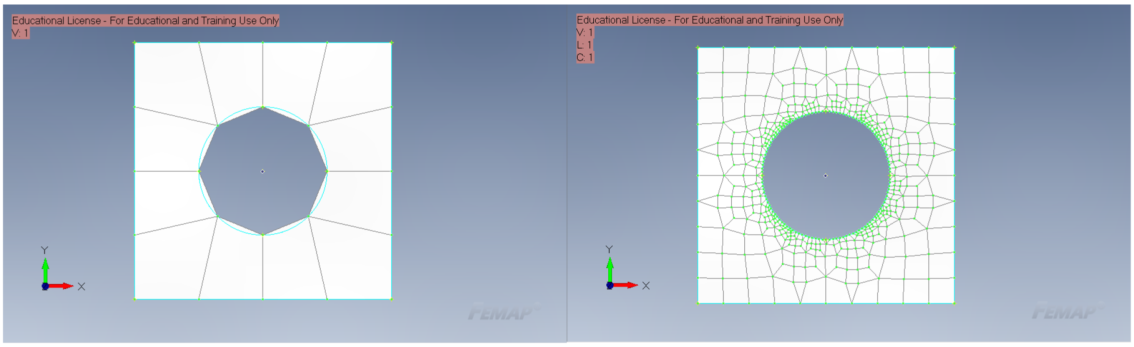


Figure 9: Plate geometry with coars (left) and fine mesh (right) around the hole

Figure 9 shows two different ways of meshing around a hole in a plate structure. Since we are using quadratic and triangular elements for meshing in this case, it's not possible to describe the circle curve of the hole exactly. To save computing time it is not useful to choose too small mesh sizes at a high distance from the hole. Near the hole, however, it makes sense to use an appropriately small element size. Otherwise the simulations near the hole will be too distorted and deviate too much from reality. Figures 10 and 11 show the influence on the FEM simulation when using different meshes.

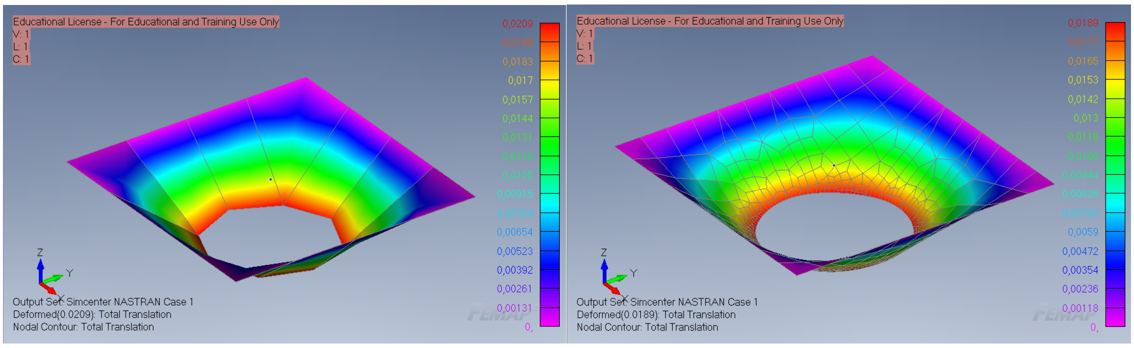


Figure 10: Total translation

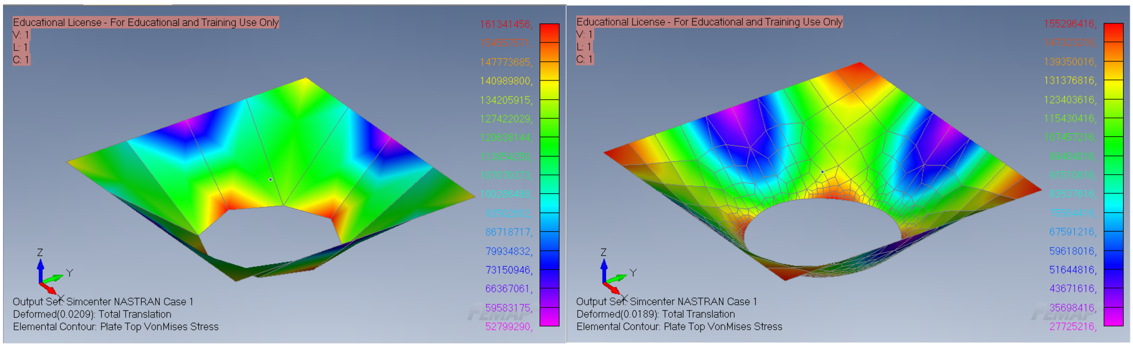


Figure 11: Von Mises stress

Regarding maximum total translation, the two simulations with the coarse and fine mesh around the hole differ by $10,58\%$. The Von Mises stress differs by $3,89\%$. This is because the circular structure of the coarse mesh cannot be reproduced correctly and must therefore be approximated too roughly.

2.11 Sources of error when using the finite element method

The correct application of the FEM requires a comprehensive knowledge of the associated avoidable and inherent sources of error in order to achieve meaningful and reliable results. These are:

- Discretization
  - Discretization and idealization of the structure
  - FE interpolation
- Numerical solution methods
  - Simplified load introduction, e.g. through concentrated point loads
  - Residual error of the numeric integration
- Network quality
  - Insufficient cross-linking density in areas with large gradients
  - Severe distortion of the elements
  - Use of incompatible elements of different integration orders
- Numerical inaccuracies
  - Rounding errors
  - Poor condition of the matrices due to very different element dimensions
  - Numerical singularities for boundary conditions

2.12 Structural optimization

The Finite Element Method (FEM) allows the machine designer to calculate the static and (with limitations) the dynamic and thermal behaviour of a machine design. This allows him to get an idea of the expected machine characteristics and to determine, for example, the static stiffness of the overall machine at the tool center point (TCP) or the values of the undamped natural frequencies before the first physical prototype of the machine is built. Finite Element Analysis (FEA) is usually used to identify weaknesses in the current design, which are then improved in a subsequent step of the design process will be.

However, the FEA does not make any statements on what these improvements should look like. If only individual machine elements are affected, such as guide shoes, these can be replaced by more powerful variants. If, however, structural designs of frame components are affected, the designer can only draw on his wealth of experience and assume which changes are to be made at which points. This often results in a time-consuming iterative process that drives up development costs. For this purpose numerical optimization methods have been developed, which are summarized under the collective term "structural optimization".The three most frequently used methods for structural optimization are topology optimization, dimensioning (sizing) and shape optimization. To be able to perform structural optimization at all, a suitable optimization algorithm can be used to create a new structure like in can be seen in figure 12.

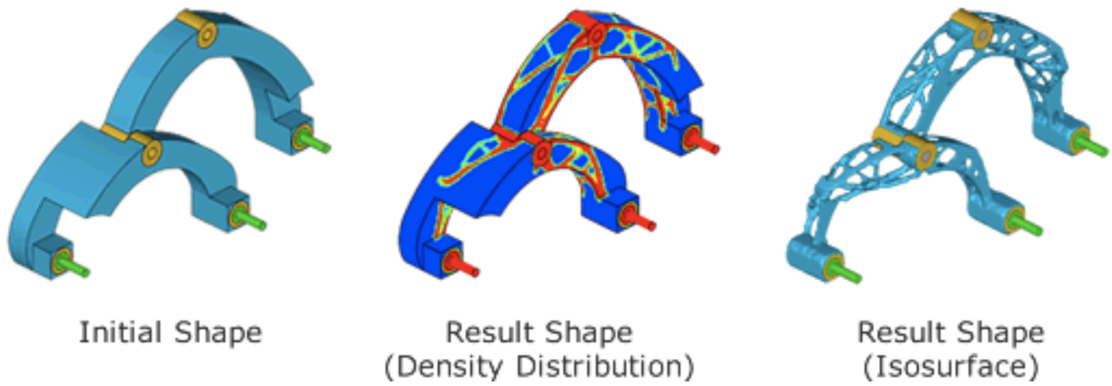


Figure 12: Structural optimization example [6]

All methods of structural optimization are based on a common mathematical model. The quantity to be optimized is called the objective function $f(x)$. x is the vector of the optimization parameters. The optimization aims to minimize the objective function $f(x)$ as a function of n optimization parameters x_1 to x_n. Each optimization parameter x_i is limited by an upper limit $x_{i,max}$ and a lower limit x_i, min. In addition, depending on the application, there are a number of inequality restrictions $g_i(x) \leq 0$, which must be observed. Equality restrictions $h_i(x) = 0$ hardly ever occur in structural optimization. For example, if the mass of a component is required to be reduced, $f(x)$ is the mass of the component. x_1 to x_n are the optimization parameters, for example the wall thickness of the ribbing. For example, if seven ribs are released for optimization, $n = 7$ and each of the seven optimization parameters x_i corresponds to a wall thickness. The limits usually depend on the possibilities of the foundry or, in the case of welded constructions, on the available sheet thicknesses. Since a mass minimization without restrictions is pointless (the optimization system would simply set all parameters to x_{min} in such a case), an additional requirement can be directed to the stiffness of the component, for example that a force of F in the tool center point causes a maximum permissible deformation. In this case:

$$u_{TCP}(x) \leq u_{max} \tag{8}$$

$$g(x) = u_{TCP}(x) - u_{max} \leq 0 \tag{9}$$

Conversely, the optimization could also be designed in such a way that the deformation on the tool center point is to be minimized and the mass of the component must not exceed a certain value m_max. In this case:

$$f(x) = u_{TCP}(x) \tag{10}$$

$$g(x) = m(x) - m_{max} \leq 0 \tag{11}$$

In the case that $f(x)$ is a known function, its extremes and saddle points can be determined by means of the curve discussion. The zeros of the first derivative indicate where extremes and saddle points are located, the behavior of the second derivative (Hesse Matrix) allows to draw conclusions about whether they are minima, maxima or saddle points. Unfortunately, $f(x)$ can only in very few cases be set up analytically closed. In industrially relevant cases this is no longer practicable with the means of elementary strength theory due to the complex geometries. For this reason, the objective function and restriction functions of structural optimization are done with FEM.

The actual optimization, i.e. the systematic change of the vector of the optimization parameters x to minimize the objective function $f(x)$ under consideration of restrictions $g(x)$ is performed by specially developed optimization algorithms. The most important optimization algorithms for structural optimization can be summarized in three classes:

- Optimality criteria (OC)
- Mathematical Programming (MP)
- Evolutionary Algorithms (EA) or Genetic Algorithms (GA)

These procedures work iteratively, i.e. a repeated finite element analysis is necessary. Due to their iterative nature, the optimization algorithms usually do not find the mathematically exact optimum, but an approximate solution. A convergence criterion indicates when a solution can be considered optimal; convergence criteria can be, for example, the maximum number of iterations or the percentage difference between the objective functions of two successive iterations.

In the following figure 13 are given some examples of these kinds of optimization.

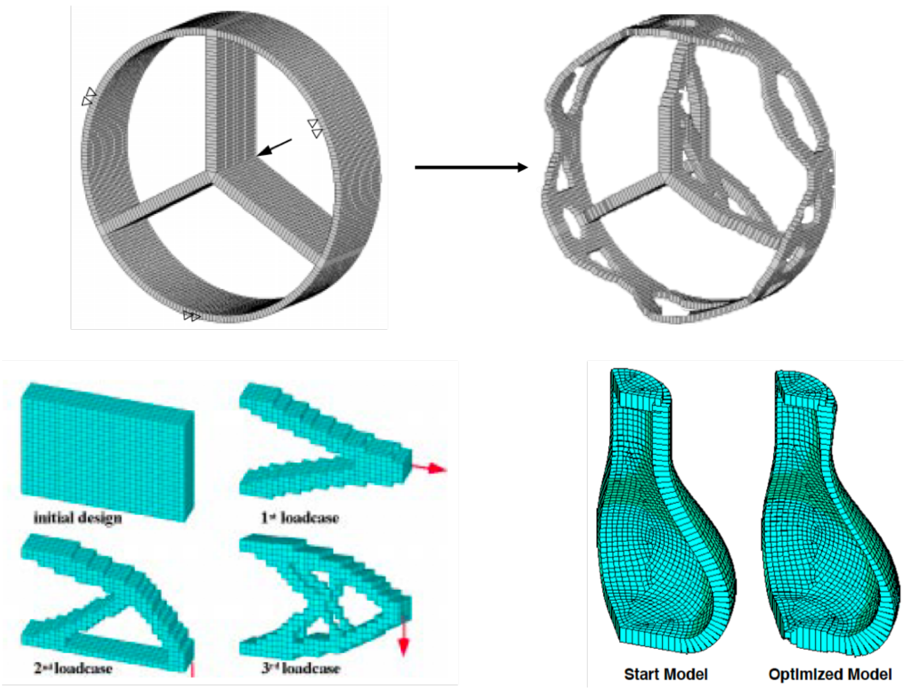


Figure 13: Examples for Structural Optimization [2]

In the following the individual structural optimization methods are explained in more detail.

- Topology optimization
  - Calculation of an optimal material distribution in a given installation space
- Dimensioning (Sizing)
  - Wall thickness optimization
  - Cross-section optimization
  - Layer thickness or fiber angle optimization of fiber composites
- Shape Optimization
  - Calculation of the optimum shape of a component
  - Notch stress optimization on roundings

2.13 Calculation Example: Beam under static point load

2.13.1 Analytical approach

The following example (see figure 14) illustrates the analytical approach to the calculation of the bending line. A 3m long beam (Stainless Steel Specialty 304, E=$1,9305 \cdot 10^{11} \frac{N}{m^2}$) with a square cross-section ($t = b = 0,05m$) is fixed to a wall and exposed to a point load of $10kN$.

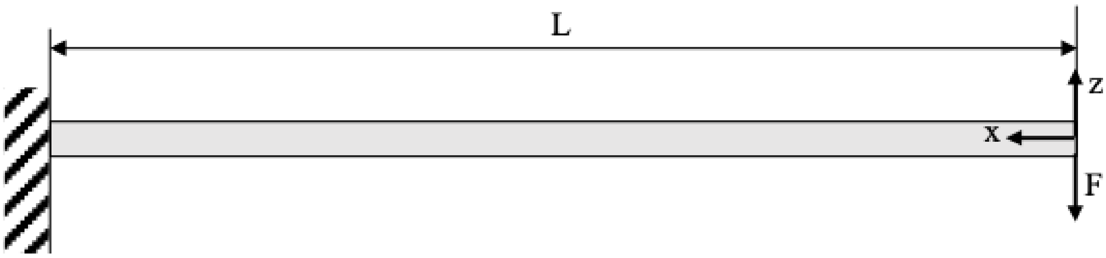


Figure 14: Calculation Example: Beam under static point load

Using the purely geometric definition of a curve curvature, the differential equation of beam bending follows, considering Hook's law of matter.

$$\frac{\frac{\partial^2 w}{\partial x^2}}{(1 + \frac{\partial w}{\partial x})^{\frac{3}{2}}} = \frac{-M_y(x)}{EI} \tag{12}$$

For small deflections ($\frac{\partial w}{\partial x} << 1$) the equation can be simplified. In addition, the definition of the moment as a function of x and the point load F is used, taking the positive cutting edge into account.

$$\frac{\partial^2 w}{\partial x^2} = \frac{Fx}{EI} \tag{13}$$

To solve the differential equation, the differential equation must be integrated twice.

$$\frac{\partial w}{\partial x} = \frac{Fx^2}{2EI} + C_1 \overset{!}{=} \Theta(x) \tag{14}$$

$$w(x) = \frac{Fx^3}{6EI} + C_1 x + C_2 \tag{15}$$

The integrated equation contains still unknown constants. To find them, we have to use our boundary conditions. Since our beam is fixed to the left wall, no displacement or rotation of the beam is possible at this location. Considering that the rotation of the beam at a location x is the first derivative of the displacement, the boundary conditions are in the given coordinate system:

$$w(x = L) = 0 \tag{16}$$

$$\theta(x = L) = \frac{\partial w}{\partial x}(x = L) = 0 \tag{17}$$

This can be used to calculate our constants:

$$C_1 = \frac{FL^2}{2} \wedge C_2 = \frac{-FL^3}{3} \tag{18}$$

Finally, the following equation follows for the displacement in z-direction and the angle Θ of the beam at any point x:

$$w(x) = \frac{F}{EI}(-\frac{x^3}{6} + \frac{L^2 x}{2} - \frac{L^3}{3}) \tag{19}$$

$$\Theta(x) = \frac{F}{2EI}(L^2 - x^2) \tag{20}$$

To be able to compare the analytical method with FEM, we now finally calculate the maximum displacement of the beam at the point $x = 0$.

$$w(x=0) = \frac{FL^3}{3EI} = \frac{10000N \cdot (3m)^3}{3 \cdot 1,9305 \cdot 10^{11} \frac{N}{m^2} \cdot \frac{(0,05m)(0,05m)^3}{12}} = 0,895105m \tag{21}$$

2.13.2 4-Node-Beam model calculated with FEM

In the last chapter we calculated the bending line analytically and determined that our maximum displacement is $0,895105m$. At first we compare the result with a simulation in FEMAP, in which the system was re-modelled (see figure 15).

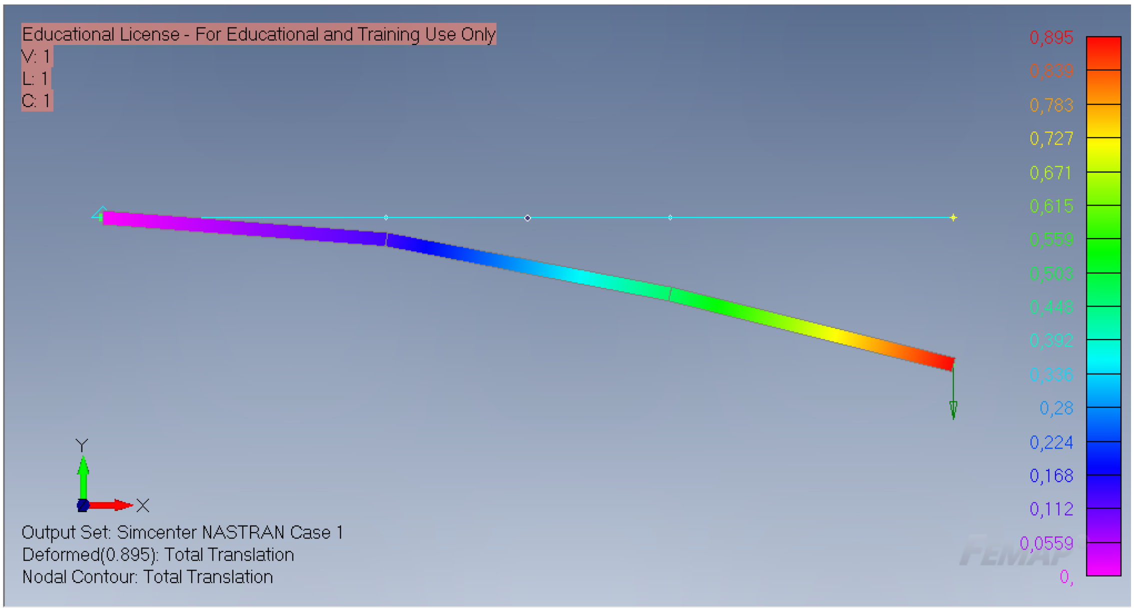


Figure 15: Beam model in FEMAP

The model was divided in a structure with 3 beams and 4 nodes. It can be seen that the maximum displacement is at least the same in the first decimal places of the analytical method and the FEM simulation. In the next steps, a method is used to show which steps the software can go through to calculate the shown results. For this purpose we first split our system into three beam elements with the corresponding nodes, cut the system free and add the forces and moments in the sketch in the xz-plane. The sketch is given in figure 16.

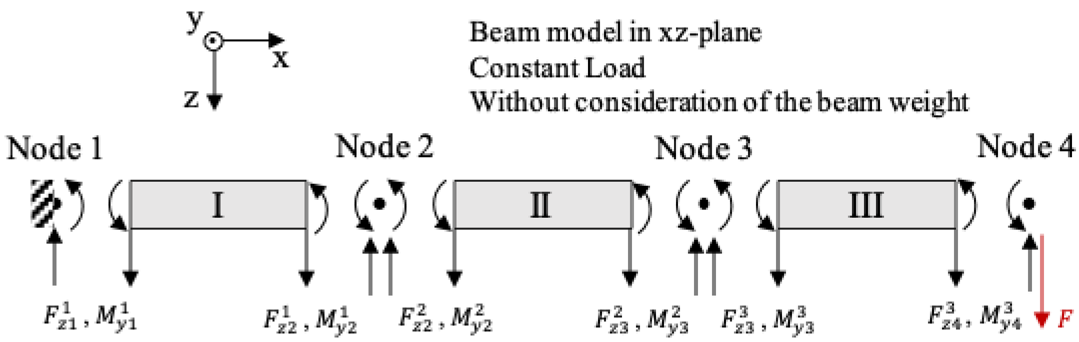


Figure 16: Sketch of the system

Only the displacement in the z- direction and rotation around the y-axis matter in the planar beam bending problem. In the first step, the element contributions of the beams have to be written down. In this case, the variable h is $1m$ since the beams have equal length. For the beam bending behaviour in the xz-plane [4], the following applies:

$$Beam\ I: \begin{Bmatrix} F_{Z1}^1 \\ M_{Y1}^1 \\ F_{Z2}^1 \\ M_{Y2}^1 \end{Bmatrix} = \frac{EI}{h^3} \begin{bmatrix} 12 & -6h & -12 & -6h \\ -6h & 4h^2 & 6h & 2h^2 \\ -12 & 6h & 12 & 6h \\ -6h & 2h^2 & 6h & 4h^2 \end{bmatrix} \begin{Bmatrix} u_{Z1} = 0 \\ \Theta_{Y1} = 0 \\ u_{Z2} \\ \Theta_{Y2} \end{Bmatrix} \tag{22}$$

$$Beam\ II: \begin{Bmatrix} F_{Z2}^2 \\ M_{Y2}^2 \\ F_{Z3}^2 \\ M_{Y3}^2 \end{Bmatrix} = \frac{EI}{h^3} \begin{bmatrix} 12 & -6h & -12 & -6h \\ -6h & 4h^2 & 6h & 2h^2 \\ -12 & 6h & 12 & 6h \\ -6h & 2h^2 & 6h & 4h^2 \end{bmatrix} \begin{Bmatrix} u_{Z2} \\ \Theta_{Y2} \\ u_{Z3} \\ \Theta_{Y3} \end{Bmatrix} \tag{23}$$

$$Beam\ III: \begin{Bmatrix} F_{Z3}^3 \\ M_{Y3}^3 \\ F_{Z4}^3 \\ M_{Y4}^3 \end{Bmatrix} = \frac{EI}{h^3} \begin{bmatrix} 12 & -6h & -12 & -6h \\ -6h & 4h^2 & 6h & 2h^2 \\ -12 & 6h & 12 & 6h \\ -6h & 2h^2 & 6h & 4h^2 \end{bmatrix} \begin{Bmatrix} u_{Z3} \\ \Theta_{Y3} \\ u_{Z4} \\ \Theta_{Y4} \end{Bmatrix} \tag{24}$$

E: Modulus of elasticity $[\frac{N}{m^2}]$; I: Second moment of inertia $[m^4]$; h: Length of the beam element $[m]$

Since we already know the displacements and rotations at node 1 ($u_{Z1} = \Theta_{Y1} = 0$), we only have to determine the equations for the other nodes:

$$-F_{Z2}^1 + -F_{Z2}^2 = 0 \tag{25}$$
$$M_{Y2}^1 + M_{Y2}^2 = 0 \tag{26}$$
$$-F_{Z3}^2 + -F_{Z3}^3 = 0 \tag{27}$$
$$M_{Y3}^2 + M_{Y3}^3 = 0 \tag{28}$$
$$-F_{Z4}^3 + F = 0 \tag{29}$$
$$M_{Y4}^3 = 0 \tag{30}$$

After the determination of the equations, we use the defined forces and moments from equations 22, 23 and 24 and insert them in the equations 25 to 30. After simplifying the equations, the following relationships result:

$$\frac{EI}{h^3}[24u_{Z2} - 12u_{Z3} - 6h\Theta_{Y3}] = 0 \tag{31}$$

$$\frac{EI}{h^3}[8h^2\Theta_{Y2} + 6hu_{Z3} + 2h^2\Theta_{Y3}] = 0 \tag{32}$$

$$\frac{EI}{h^3}[-12u_{Z2} + 6h\Theta_{Y2} + 24u_{Z3} - 12u_{Z4} - 6h\Theta_{Y4}] = 0 \tag{33}$$

$$\frac{EI}{h^3}[-6hu_{Z2} + 2h^2\Theta_{Y2} + 8h^2\Theta_{Y3} + 6hu_{Z4} + 2h^2\Theta_{Y4}] = 0 \tag{34}$$

$$\frac{EI}{h^3}[-12u_{Z3} + 6h\Theta_{Y3} + 12u_{Z4} + 6h\Theta_{Y4}] - F = 0 \tag{35}$$

$$\frac{EI}{h^3}[-6hu_{Z3} + 2h^2\Theta_{Y3} + 6hu_{Z4} + 4h^2\Theta_{Y4}] = 0 \tag{36}$$

To simplify further calculations, the equations can be written down in matrix formulation using a $6x$6-matrix. (Reminder: Normally we would get a 8x8 matrix with 2 degrees of freedom for every node. Since we know the behaviour of node 1, we simplify it to a 6x6 matrix.)

$$\frac{EI}{h^3}\begin{bmatrix} 24 & 0 & -12 & -6h & 0 & 0 \\ 0 & 8h^2 & 6h & 2h^2 & 0 & 0 \\ -12 & 6h & 24 & 0 & -12 & -6h \\ -6h & 2h^2 & 0 & 8h^2 & 6h & 2h^2 \\ 0 & 0 & -12 & 6h & 12 & 6h \\ 0 & 0 & -6h & 2h^2 & 6h & 4h^2 \end{bmatrix} \begin{Bmatrix} u_{Z2} \\ \Theta_{Y2} \\ u_{Z3} \\ \Theta_{Y3} \\ u_{Z3} \\ \Theta_{Y3} \end{Bmatrix} - \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ F \\ 0 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \tag{37}$$

$$\Leftrightarrow \begin{bmatrix} 24 & 0 & -12 & -6h & 0 & 0 \\ 0 & 8h^2 & 6h & 2h^2 & 0 & 0 \\ -12 & 6h & 24 & 0 & -12 & -6h \\ -6h & 2h^2 & 0 & 8h^2 & 6h & 2h^2 \\ 0 & 0 & -12 & 6h & 12 & 6h \\ 0 & 0 & -6h & 2h^2 & 6h & 4h^2 \end{bmatrix} \begin{Bmatrix} u_{Z2} \\ \Theta_{Y2} \\ u_{Z3} \\ \Theta_{Y3} \\ u_{Z3} \\ \Theta_{Y3} \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ \frac{Fh^3}{EI} \\ 0 \end{Bmatrix} \tag{38}$$

By inverting the matrix and inserting the geometric quantities, the displacements and rotations in the nodes can be calculated.

$$\begin{Bmatrix} u_{Z2} \\ \Theta_{Y2} \\ u_{Z3} \\ \Theta_{Y3} \\ u_{Z3} \\ \Theta_{Y3} \end{Bmatrix} = \begin{bmatrix} 24 & 0 & -12 & -6h & 0 & 0 \\ 0 & 8h^2 & 6h & 2h^2 & 0 & 0 \\ -12 & 6h & 24 & 0 & -12 & -6h \\ -6h & 2h^2 & 0 & 8h^2 & 6h & 2h^2 \\ 0 & 0 & -12 & 6h & 12 & 6h \\ 0 & 0 & -6h & 2h^2 & 6h & 4h^2 \end{bmatrix}^{-1} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ \frac{Fh^3}{EI} \\ 0 \end{Bmatrix} \tag{39}$$

$$\Leftrightarrow \begin{Bmatrix} u_{Z2} \\ \Theta_{Y2} \\ u_{Z3} \\ \Theta_{Y3} \\ u_{Z3} \\ \Theta_{Y3} \end{Bmatrix} = \begin{bmatrix} \frac{1}{3} & -\frac{1}{3} & \frac{5}{6} & -\frac{1}{2} & \frac{4}{3} & -\frac{1}{2} \\ -\frac{1}{2} & 1 & -\frac{3}{2} & 1 & -\frac{5}{2} & 1 \\ \frac{5}{6} & -\frac{3}{2} & \frac{8}{3} & -2 & \frac{14}{3} & -2 \\ -\frac{1}{2} & 1 & -2 & 2 & -4 & 2 \\ \frac{4}{3} & -\frac{5}{2} & \frac{14}{3} & -4 & 9 & -\frac{9}{2} \\ -\frac{1}{2} & 1 & -2 & 2 & -\frac{9}{2} & 3 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ \frac{Fh^3}{EI} \\ 0 \end{Bmatrix} \tag{40}$$

$$\Leftrightarrow \begin{Bmatrix} u_{Z2} \\ \Theta_{Y2} \\ u_{Z3} \\ \Theta_{Y3} \\ u_{Z3} \\ \Theta_{Y3} \end{Bmatrix} = \begin{Bmatrix} \frac{4Fh^3}{3EI} \\ -\frac{5Fh^3}{2EI} \\ \frac{14Fh^3}{3EI} \\ -\frac{4Fh^3}{EI} \\ \frac{9Fh^3}{EI} \\ -\frac{9Fh^3}{2EI} \end{Bmatrix} = \begin{Bmatrix} 0,312608m \\ -0,24864rad \\ 0,464128m \\ -0,397824rad \\ 0,895105m \\ -0,447552rad \end{Bmatrix} \widehat{=} \begin{Bmatrix} 0,312608m \\ -14,246^\circ \\ 0,464128m \\ -22,794^\circ \\ \mathbf{0{,}895105m} \\ -25,643^\circ \end{Bmatrix} \tag{41}$$

As we can see, the analytical solution and the numerical solution using FEM give us the same exact results. The reason behind this is that the beam element approximation matches the beam behaviour exactly ath the point of maximum displacement. For other geometries, which have a much more complex structure, the numerical solution is slightly different from the analytical approach. Is the difference between the two values too high, the meshing size of the FEM net (the approximation of the structure via FE) could be a problem.

3 Conclusion

This document is intended to highlight the basics of the finite element method in order to deepen the FEM simulation procedures to be learnt in the Machine Design course and to extend them with more detailed information. It is intended to be an additional document to better understand the simulations carried out in Siemens NX in theory and to get an insight under the hood.

The first step was to provide basic information about the method and to explain the first steps before the simulation, which should be thought through and carried out as preparation. Then basic calculation methods for simple FEM elements were explained and the calculation procedure was described. In addition, it was shown why a software-based solution is irreplaceable for larger simulations, but also for only a few elements, since the complexity of the matrix calculations increases disproportionately with increasing accuracy. By replacing an exact analytical solution, the possible sources of error in an FEM analysis, which the engineer must keep in mind, were then characterized. Finally, it was briefly pointed out how the static FEM analysis can be extended by dynamic simulations, for example to analyze the vibration behavior. To give the students a better insight behind the calculations, the analytical approach was explained using an example and the processes in the background of an FEM calculation were derived manually.

In the context of this document, mainly the finite element method was described to analyze and solve mechanical problems. However, it should not be forgotten that thermal, fluidic or electromagnetic phenomena can also be analyzed with FEM or very closely related methods. For more information about the FE method, the courses MEC-E1050 Finite Elements in Solids and MEC-E8005 Thin-Walled Structures can be recommended. Further information can be found in the relevant literature on the finite element method (See also [10], [2]).

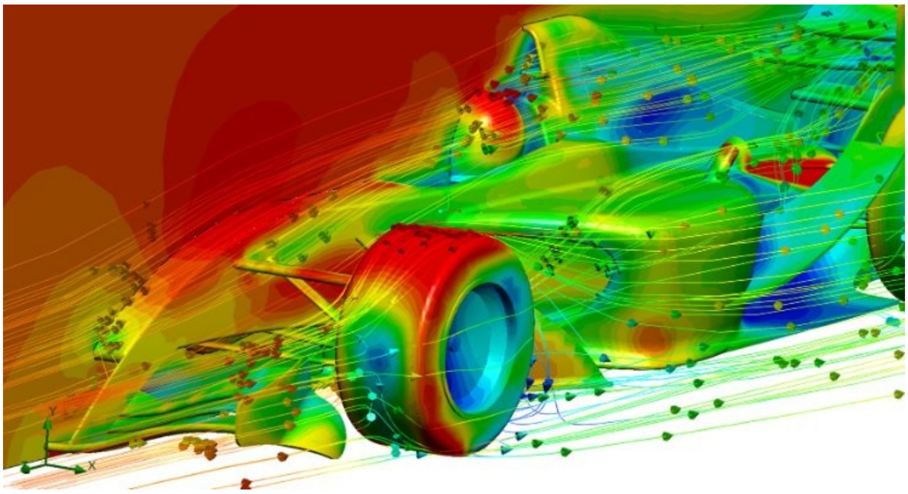


Figure 17: Computational Fluid Dynamics (CFD) and Finite Element Analysis (FEA) [7]

References

[1] M.F. Ashby. *Materials Selection in Mechanical Design*. Butterworth-Heinemann, 2016.

[2] Nima Bakhtiary. “A new approach for sizing, shape and Topology Optimization”. In: *1996 SAE International Congress and Exposition* (Feb. 1996).

[3] Comsol. *Finite Element Analysis (FEA) Software*. 2015. URL: `https://www.comsol.com/multiphysics/fea-software`.

[4] Jouni Freund. “Lecture: Finite Elements in Solids”. In: *Aalto University* (Oct. 2020).

[5] Kaur Jaakma. “Lecture: Machine Design”. In: *Aalto University* (Aug. 2020).

[6] QUINT. *Topology Optimization*. URL: `http://www.quint.co.jp/eng/pro/ots/ots_fnc-tpo.htm`.

[7] RahulKrishnan. *Computational Fluid Dynamics (CFD), Finite Element Analysis (FEA) - benefits, steps, scope, examples (Boeing, Saudi Aramco - ANSYS FLUENT)*. Jan. 2018. URL: `https://www.linkedin.com/pulse/computational-fluid-dynamics-cfd-finite-element-fea-rahul-krishnan/`.

[8] Jani Romanoff. “Lecture: Thin Walled Structures”. In: *Aalto University* (Aug. 2020).

[9] Dipl.-Ing. Simo Schmidt. “FEM I: Theoretische Grundlagen und Berechnungsablauf”. In: *Messtechnik und Strukturanalyse, RWTH Aachen* (2018).

[10] Zhu Zienkiewicz Taylor. *The Finite Element Method*. Butterworth-Heinemann, 2005.