NUMERICAL ANALYSIS OF ENGINEERING STRUCTURES

(LINEAR ELASTICITY AND THE FINITE ELEMENT METHOD)

SZÉCHENYI 2020

HUNGARIAN GOVERNMENT

European Union
European Social Fund

INVESTING IN YOUR FUTURE

NUMERICAL ANALYSIS OF ENGINEERING STRUCTURES

(LINEAR ELASTICITY AND THE FINITE ELEMENT METHOD)

Author:

Tamás Mankovits

Reviewed by: Zsolt Tiba

Manuscript closed: October 2014.

PUBLISHER: University of Debrecen
in the frames of the project ZENFE (Green Energy cooperation in Higher Education)
TÁMOP-4.1.1.C-12/1/KONV-2012-0012

Publisher: Edit Szűcs
Editor: Gábor Grasselli
Technical editor: Tibor Balla
Volume: 5 sheets

Debrecen, 2014

CONTENTS

LIST OF TABLES

LIST OF FIGURES

PREFACE

Finite element analysis is an important part of engineering design. Several commercial finite element programs are available. The proper usage of these programs highly depends on the user's ability and technical knowledge.

This note presents the theoretical background of the linear elasticity and the finite element method indispensable for the correct and reliable analysis. Numerical examples presented step by step make it easier for the user to elaborate an own numerical simulation.

This note is written to primarily students majoring in the mechanical engineering BSc in the framework of the Finite Element Method course. The understanding of the governing equations requires an extensive mathematical knowledge which explanation is not part of this note.

I would like to thank the contribution to this note of my colleague Dávid Huri and my student Tamás Antal Varga. They special effort in editing and developing new mechanical problems makes this note more colourful and this way it is easier to understand.

I thank Zsolt Tiba, who reviewed the manuscript and provided constructive suggestions and criticisms that have helped improve the note.

Tamás Mankovits

1. INTRODUCTION

It is a fundamental requirement that the important technical information in terms of the product lifecycle can be handled effective. The CAD (Computer Aided Design), the CAM (Computer Aided Manufacturing) and CAE (Computer Aided Engineering) systems are applied to fulfil these requirements. The above mentioned systems include conception modeling, geometric modeling, numerical analysis and simulation, drawing and illustration, documentation, manufacturing processing, database handling throughout standardized data communication.

The Computer Aided Design for creating design conceptions and drawings, defining geometry used in CAM and CAE systems is an appropriate application nowadays. The most common CAD softwares are AutoCAD, Solid Edge, SolidWorks, Catia, Creo, NX, etc. The Computer Aided Manufacturing technology are used for designing, organizing and control of the manufacturing process and also a good tool for simulating the manufacturing process necessary for producing parts or structures. Furthermore the CAM is suitable for operating the selection and positioning of the tools and workpieces of the manufacturing cells. Notable softwares are EdgeCAM and MasterCAM. The different finite element softwares belonging to the Computer Aided Engineering systems are useful for simulating the response of the products and structures. This feature is used in redesigning or optimalization in the industrial application. There are several finite element computer programs for common application like Femap, Ansys, Marc, Abaqus, Adina, etc. This book intends to give a brief overview about the finite element method and the application of Femap throughout numerous practical problems.

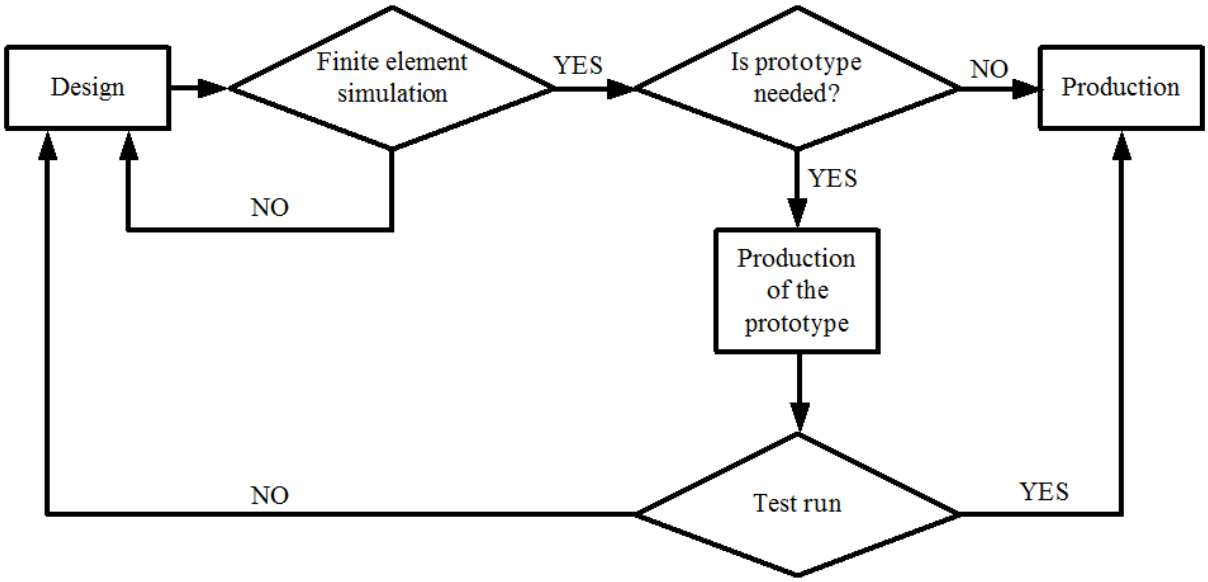


Figure 1.1. Simplified model of the production including the finite element simulation

The finite element method is a numerical tool for determining approximate solutions to several engineering problems. The background of the finite element method is the approximate solution of the partial different equation. This method has become an indispensible tool in mechanical analysis since almost every natural phenomena and behavior can be handled with its application thanks to the modern finite element softwares.

The need of the industry to reduce costs and improve quality based on the product lifecycle requires the usage of digital simulation. A significant part of the manufacturing costs are the producing prototypes and conducting experimental tests. These also require special conditions

to be performed and of course experts. Applying the finite element simulation these expenditures can be reduced. The product development in the everyday industrial application is inconceivable without finite element analysis. The simplified model of the production including the finite element simulation can be seen in Figure 1.1.
The basic idea is to find the solution of a complicated problem by replacing it by a simpler one. As a result of this idea the experts are able to find the approximate solution of a given problem. However, the existing mathematical and mechanical tools are not sufficient to find the exact solution. Since it is very difficult to find exact resolution to the problem investigated under any specified condition the approximate solution is good enough to obtain information.
In the finite element method the solution region is considered as built up of many small, interconnected sub-regions called finite elements, so the complex partial differential equations are reduced to linear or nonlinear simultaneous equations. This finite element discretization procedure gives finite number of unknowns at specified points called nodes.

1.1. Historical background of the finite element method

Analytical calculation of a beam structure divided into number of elements was executed by Cook around 1900. In 1943 an approach similar to the finite element method using continuous functions defined over triangular regions was suggested by Courant. Courant was introduced the finite element method solving a torsional problem. In the 50's an airplane wing was analyzed using triangular elements by Boeing. The stresses were evaluated. In 1956 plane strain problems were calculated by Turner et al. In the 60's the finite element method has become popular, Clough was played important role to make it widespread. Over this time heat transfer problems were also analyzed using the finite element method. The first softwares using displacement based finite element method were developed (NASTRAN, ASKA). The first book in this area was published in 1967 by Zienkiewicz and Cheung. The International Journal for Numerical Methods in Engineering was established in 1969 which is still one of the most important journal in the field of finite element method research. In the 70's nonlinear problems were analyzed by the method. We can declare almost every natural phenomena and behavior can be handled by finite element method. However, this development would not have been possible without the rapid development of the computers started in 1946. Today's finite element softwares are suitable for analyzing and computing several engineering problems in the field of structural mechanics (statics, strength and dynamical analyses, fracture mechanics, stability analysis, etc.), manufacturing process analysis (molding, etc.), fluid mechanics, heat transfer, electrostatics problem, etc.

1.2. Brief overview of the finite element method

The finite element method is used to solve physical problems in engineering analysis. The physical problem involves a structure or a structural component subjected to loads. The physical model than has to be idealized to a mathematical model which leads to differential equations. This mathematical model is solved by the finite element analysis. Since it is numerical procedure the expected accuracy of the solution has to be criteria. Better accuracy can be reached by finer mesh. The process of the finite element analysis can be seen in Figure 1.2.

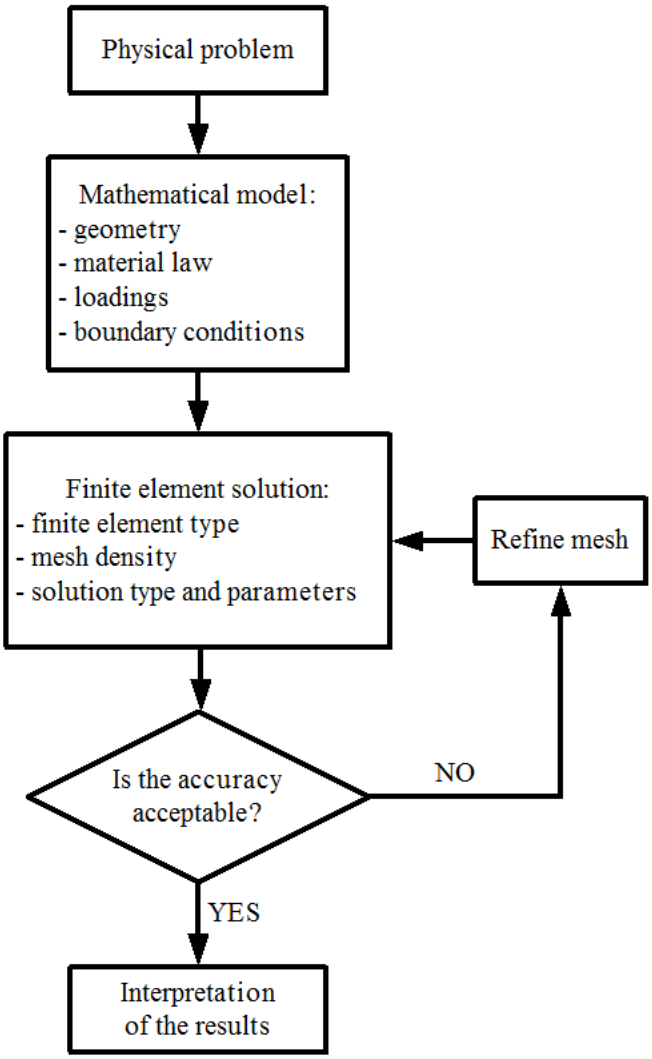


Figure 1.2. The process of the finite element analysis

In short, the finite element analysis starts with the finite element discretization. The examined domain has to be divided into small, interconnected finite elements. The elements are determined by the nodes. It includes numbering the elements and the nodes. This procedure is called meshing, see in Figure 1.3. The main properties of the nodes are the degree of freedom and its coordinates. The type of finite elements depending on the examined problem can be chosen. One-dimensional, two-dimensional and three-dimensional elements exist.

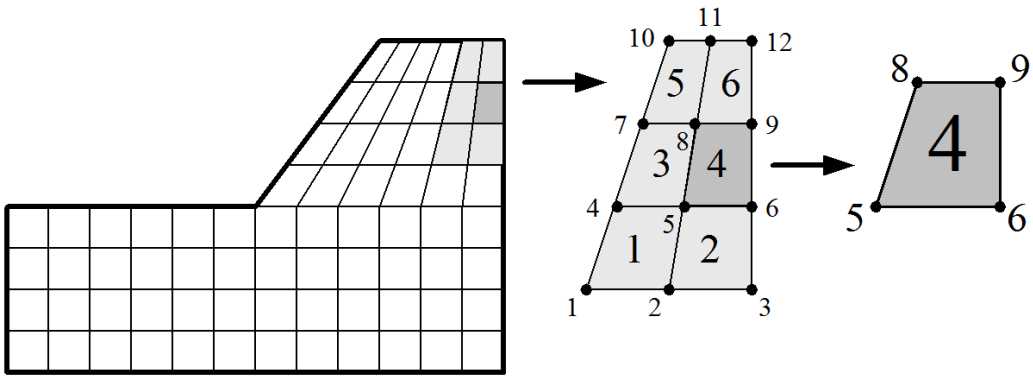


Figure 1.3. Finite element discretization

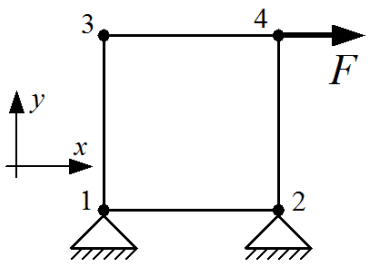
From these nodal values the values of the function sought (mainly the displacements) are calculated by interpolation method. The interpolation functions are called shape functions. The shape functions are mainly polynomials and linear or quadratic.
The next step is the transformation of the problem given in partial differential equations form. The aim is to produce an algebraic equation system which solution gives the functions sought in the nodes. The form of the algebraic equation system is

$$\mathbf{K}^e\mathbf{q}^e = \mathbf{f}^e, \tag{1.1}$$

where $\mathbf{K}^e$ is the element stiffness matrix, which refers to the property of the used elements, $\mathbf{q}^e$ is the nodal displacement vector of an element, $\mathbf{f}^e$ is the element load vector, ^{e} denotes the element.
Note, that some connecting elements are jointed at the same nodes, see in Figure 1.3. These topological features have to be taken into consideration when assembling the algebraic equation system for the global structure,

$$\mathbf{Kq} = \mathbf{f}, \tag{1.2}$$

where $\mathbf{K}$ is the global stiffness matrix, which refers to the property of the used elements, $\mathbf{q}$ is the global nodal displacement vector, $\mathbf{f}$ is the global load vector. It gives the degree of freedom of the structure.
We also have to ensure the boundary conditions. Supports and loads having a prescribed or zero values have to be eliminated. It reduces the size of the numerous algebraic equations to be solved, see in Figure 1.4. In Figure 1.4, the subscript denotes the node number, u and v are the nodal displacement coordinates, F_x and F_y are the components of the force vector in x and y directions, respectively.



$$\mathbf{q} = \begin{vmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{vmatrix} = \begin{vmatrix} 0 \\ 0 \\ 0 \\ 0 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{vmatrix} \qquad \mathbf{f} = \begin{vmatrix} F_{1x} \\ F_{1y} \\ F_{2x} \\ F_{2y} \\ F_{3x} \\ F_{3y} \\ F_{4x} \\ F_{4y} \end{vmatrix} = \begin{vmatrix} F_{1x} \\ F_{1y} \\ F_{2x} \\ F_{2y} \\ 0 \\ 0 \\ F \\ 0 \end{vmatrix}$$

Figure 1.4. Boundary conditions

The nodes labeled by 1 and 2 are constrained, while the node labeled by 3 is unloaded.
When solving the global algebraic equation system the aim is to determine the unknown parameters of $\mathbf{q}$.
There are number of methods to produce the solutions of the linear algebraic equation systems. Here, the multiplication with inverse of the global stiffness matrix is used,

$$\mathbf{q} = \mathbf{K}^{-1}\mathbf{f}. \tag{1.3}$$

After solving the global algebraic equation system the values of the displacement field in the nodes are known. Using the displacement field the quantities of interest can be calculated, like deformation and stress.
The quality of the finite element results depends on numerous factors including computational technology in the code, experience and level of understanding of the engineer analyst, and the

interpretation of the results. Deficiencies in any of the above mentioned parameters can lead to erroneous results or a poor design. However, an experienced analyst can use the analysis as a verification as well as a predictive tool for better product and process design. The finite element analysis does not replace component testing, but it can significantly reduce the testing of performance and structural integrity.

2. FUNDAMENTALS OF LINEAR ELASTICITY

The elasticity is the mechanics of elastic bodies. It is known that solid bodies change their shape under load. The elastic body is capable to deform elastically. The elastic deformation means that the body undergone deformation gets back its original shape once the load ends. The task of the elasticity is to determine the displacement, the strain and the stress states of the body points. Depending on the connection between the stress and strain this elastic deformation can be linear or nonlinear. If the function between the stress and strain is linear we say linear elastic deformation. Materials which behave as linear are the steel, cast iron, aluminum, etc. If the function between the stress and strain is nonlinear we say nonlinear elastic deformation. The most typical nonlinear material is the rubber. This book deals with only the theory of linear elasticity.
The theory of elasticity establishes a mathematical model of the problem which requires mathematical knowledge to be able to understand the formulations and the solution procedures. The governing partial differential equations are formulated in vector and tensor notation.

2.1. The unknown fields in the linear theory of elasticity

2.1.1. Displacement field

Considering the theory of linear elasticity the deformed body under loading gets back its original shape once the loading ends.
Now, let us consider a general elastic body undergone deformation as shown in Figure 2.1. As a result of the applied loadings, the elastic solids will change shape or deform, and these deformations can be quantified with the displacements of material points in the body. The continuum hypothesis establishes a displacement field at all points within the elastic solid.
We have selected an arbitrary point in the body called P. In the deformed state, point P moves to point P'.

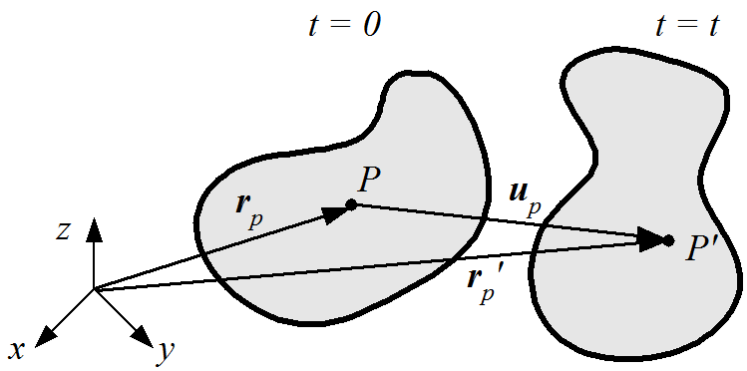


Figure 2.1. Derivation of the displacement vector

Using Cartesian coordinates the $\boldsymbol{r}_P$ and $\boldsymbol{r}'_P$ are the space vectors of P and P', respectively, where $\boldsymbol{r}'_P$ can be described using $\boldsymbol{r}_P$

$$\boldsymbol{r}_P = x_P\boldsymbol{i} + y_P\boldsymbol{j} + z_P\boldsymbol{k}, \qquad (2.1)$$
$$\boldsymbol{r}'_P = \boldsymbol{r}_P + \boldsymbol{u}_P,$$

where $\boldsymbol{u}_P$ is the displacement vector of point P,

$$\boldsymbol{u}_P = u_P\boldsymbol{i} + v_P\boldsymbol{j} + w_P\boldsymbol{k}. \tag{2.2}$$

Here u_P, v_P and w_P are the displacement coordinates in the x, y and z directions, respectively. It can be seen that the displacement vector $\boldsymbol{u}$ will vary continuously from point to point, so it forms a displacement field of the body,

$$\boldsymbol{u} = \boldsymbol{u}(\boldsymbol{r}) = u\boldsymbol{i} + v\boldsymbol{j} + w\boldsymbol{k}. \tag{2.3}$$

We usually express it as a function of the coordinates of the undeformed geometry,

$$u = u(\boldsymbol{r}) = u(x, y, z,); \quad v = v(\boldsymbol{r}) = v(x, y, z); \quad w = w(\boldsymbol{r}) = w(x, y, z). \tag{2.4}$$

The unit of the displacement is in *mm*.

2.1.2. Derivative tensor and its decomposition

Consider a Q point which is in the very small domain about the point P, and $P \neq Q$. We have formed the vector $\Delta\boldsymbol{r}$ connecting these points by a directed line segment shown in Figure 2.2. The difference of the $\boldsymbol{u}_P$ and $\boldsymbol{u}_Q$ is the relative displacement vector $\Delta\boldsymbol{u}$. An elastic solid is said to be deformed or strained when the relative displacements between points in the body are changed. This is in contrast to rigid body motion, where the distance between points remains the same.

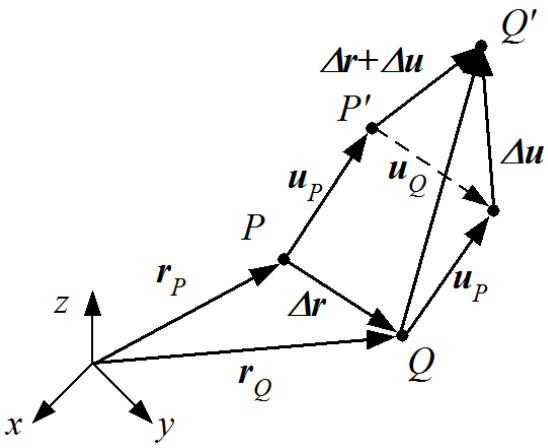


Figure 2.2. General deformation between two neighbouring points

It can be seen that $\Delta\boldsymbol{r}$ can be expressed

$$\begin{aligned}\Delta\boldsymbol{r} = \boldsymbol{r}_Q - \boldsymbol{r}_P = (x_Q - x_P)\boldsymbol{i} + (y_Q - y_P)\boldsymbol{j} + (z_Q - z_P)\boldsymbol{k} = \\ = \Delta x\boldsymbol{i} + \Delta y\boldsymbol{j} + \Delta z\boldsymbol{k}.\end{aligned} \tag{2.5}$$

Since P and Q are neighbouring points, we can use a Taylor series expansion around point Q to express the components. Note, that the higher-order terms of the expansion have been dropped since the components of $\boldsymbol{r}$ are small,

$$
\begin{aligned}
u_Q &\cong u_P + \frac{\partial u_P}{\partial x}\Delta x + \frac{\partial u_P}{\partial y}\Delta y + \frac{\partial u_P}{\partial z}\Delta z, \\
v_Q &\cong v_P + \frac{\partial v_P}{\partial x}\Delta x + \frac{\partial v_P}{\partial y}\Delta y + \frac{\partial v_P}{\partial z}\Delta z, \\
w_Q &\cong w_P + \frac{\partial w_P}{\partial x}\Delta x + \frac{\partial w_P}{\partial y}\Delta y + \frac{\partial w_P}{\partial z}\Delta z.
\end{aligned} \tag{2.6}
$$

It can be written in vector form

$$
\boldsymbol{u}_Q \cong \boldsymbol{u}_P + \begin{bmatrix} \frac{\partial \boldsymbol{u}_P}{\partial x} & \frac{\partial \boldsymbol{u}_P}{\partial y} & \frac{\partial \boldsymbol{u}_P}{\partial z} \end{bmatrix} \Delta \boldsymbol{r}, \tag{2.7}
$$

from where the approximation of the relative displacement vector is

$$
\Delta \boldsymbol{u} = \boldsymbol{u}_Q - \boldsymbol{u}_P \cong \begin{bmatrix} \frac{\partial \boldsymbol{u}_P}{\partial x} & \frac{\partial \boldsymbol{u}_P}{\partial y} & \frac{\partial \boldsymbol{u}_P}{\partial z} \end{bmatrix} \Delta \boldsymbol{r}. \tag{2.8}
$$

We can introduce now the derivative tensor $\boldsymbol{U}_P$ of the displacement field

$$
\Delta \boldsymbol{u} = \boldsymbol{U}_P \Delta \boldsymbol{r}, \tag{2.9}
$$

where

$$
\boldsymbol{U}_P = \begin{bmatrix} \frac{\partial \boldsymbol{u}_P}{\partial x} & \frac{\partial \boldsymbol{u}_P}{\partial y} & \frac{\partial \boldsymbol{u}_P}{\partial z} \end{bmatrix} = \begin{bmatrix} \frac{\partial u_P}{\partial x} & \frac{\partial u_P}{\partial y} & \frac{\partial u_P}{\partial z} \\ \frac{\partial v_P}{\partial x} & \frac{\partial v_P}{\partial y} & \frac{\partial v_P}{\partial z} \\ \frac{\partial w_P}{\partial x} & \frac{\partial w_P}{\partial y} & \frac{\partial w_P}{\partial z} \end{bmatrix}. \tag{2.10}
$$

In general

$$
\boldsymbol{U} = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} & \frac{\partial u}{\partial z} \\ \frac{\partial v}{\partial x} & \frac{\partial v}{\partial y} & \frac{\partial v}{\partial z} \\ \frac{\partial w}{\partial x} & \frac{\partial w}{\partial y} & \frac{\partial w}{\partial z} \end{bmatrix}. \tag{2.11}
$$

Considering the ∇ Hamilton differential operator

$$
\nabla = \frac{\partial}{\partial x}\boldsymbol{i} + \frac{\partial}{\partial y}\boldsymbol{j} + \frac{\partial}{\partial z}\boldsymbol{k}, \tag{2.12}
$$

the derivative tensor can be written in dyadic form

$$\boldsymbol{U} = \frac{\partial \boldsymbol{u}}{\partial x} \circ \boldsymbol{i} + \frac{\partial \boldsymbol{u}}{\partial y} \circ \boldsymbol{j} + \frac{\partial \boldsymbol{u}}{\partial z} \circ \boldsymbol{k} = \boldsymbol{u} \circ \nabla. \tag{2.13}$$

All tensors can be decomposed into the sum of a symmetric and an asymmetric tensor, so let us write the derivative tensor of the displacement field $\boldsymbol{U}$ into the following form,

$$\boldsymbol{U} = \frac{1}{2}(\boldsymbol{u} \circ \nabla + \nabla \circ \boldsymbol{u}) + \frac{1}{2}(\boldsymbol{u} \circ \nabla - \nabla \circ \boldsymbol{u}) = \frac{1}{2}(\boldsymbol{U} + \boldsymbol{U}^T) + \frac{1}{2}(\boldsymbol{U} - \boldsymbol{U}^T), \tag{2.14}$$

where is $\boldsymbol{U}^T$ the transpose of the derivative tensor,

$$\boldsymbol{U}^T = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{\partial v}{\partial x} & \frac{\partial w}{\partial x} \\ \frac{\partial u}{\partial y} & \frac{\partial v}{\partial y} & \frac{\partial w}{\partial y} \\ \frac{\partial u}{\partial z} & \frac{\partial v}{\partial z} & \frac{\partial w}{\partial z} \end{bmatrix}. \tag{2.15}$$

The first part is a symmetric tensor called strain tensor,

$$\boldsymbol{A} = \boldsymbol{U}_{symmetric} = \frac{1}{2}(\boldsymbol{U} + \boldsymbol{U}^T). \tag{2.16}$$

The second part is an asymmetric tensor called rotation tensor,

$$\boldsymbol{\Psi} = \boldsymbol{U}_{asymmetric} = \frac{1}{2}(\boldsymbol{U} - \boldsymbol{U}^T). \tag{2.17}$$

Using the Eq. 2.16 and Eq. 2.17 the derivative tensor can be written with the sum of the strain tensor and the rotation tensor,

$$\boldsymbol{U} = \boldsymbol{A} + \boldsymbol{\Psi}. \tag{2.18}$$

The strain tensor represents the pure deformation of the element, while the rotation tensor represents the rigid-body rotation of the element. The decomposition is illustrated for a two-dimensional case in Figure 2.3.

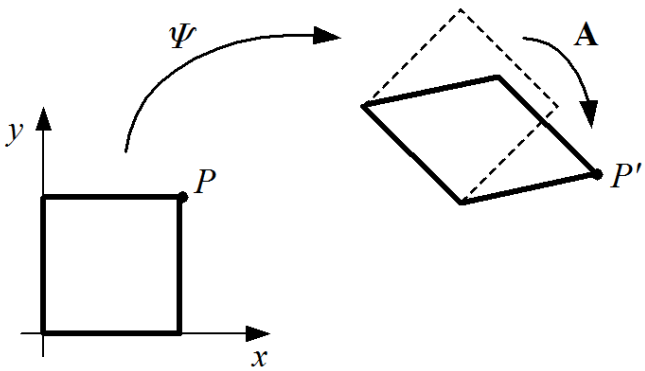


Figure 2.3. The physical interpretation of the strain tensor and the rotation tensor

2.1.3. Rotation tensor

The rotation tensor represents the rigid-body rotation of the element. Let's investigate the asymmetric part of the derivative tensor of the displacement.

$$\boldsymbol{\Psi} = \frac{1}{2}(\boldsymbol{U} - \boldsymbol{U}^T) = \begin{bmatrix} 0 & \frac{1}{2}\left(\frac{\partial u}{\partial y} - \frac{\partial v}{\partial x}\right) & \frac{1}{2}\left(\frac{\partial u}{\partial z} - \frac{\partial w}{\partial x}\right) \\ \frac{1}{2}\left(\frac{\partial v}{\partial x} - \frac{\partial u}{\partial y}\right) & 0 & \frac{1}{2}\left(\frac{\partial v}{\partial z} - \frac{\partial w}{\partial y}\right) \\ \frac{1}{2}\left(\frac{\partial w}{\partial x} - \frac{\partial u}{\partial z}\right) & \frac{1}{2}\left(\frac{\partial w}{\partial y} - \frac{\partial v}{\partial z}\right) & 0 \end{bmatrix} = \begin{bmatrix} 0 & -\varphi_z & \varphi_y \\ \varphi_z & 0 & -\varphi_x \\ -\varphi_y & \varphi_x & 0 \end{bmatrix}, \tag{2.19}$$

where φ is the angle displacement and $|\varphi| \ll 1$.
If there is no deformation, when $\boldsymbol{A} = \boldsymbol{0}$, then relative displacement can be expressed by

$$\Delta\boldsymbol{u} = \boldsymbol{\Psi}\Delta\boldsymbol{r}, \tag{2.20}$$

so the displacement vector of a body point Q can be written

$$\boldsymbol{u}_Q = \boldsymbol{u}_P + \boldsymbol{\Psi} \cdot \Delta\boldsymbol{r} = \boldsymbol{u}_P + \boldsymbol{\varphi} \times \Delta\boldsymbol{r}, \tag{2.21}$$

where $\boldsymbol{\varphi}$ is the angle displacement vector, $\boldsymbol{\varphi} = \varphi_x\boldsymbol{i} + \varphi_y\boldsymbol{j} + \varphi_z\boldsymbol{k}$ and $\boldsymbol{u}_P$ can be interpreted as a dislocation. The rigid-body rotation gives no strain energy which has an important role in the finite element calculation, so the rigid-body rotation must be constrained during the finite element model settings.

2.1.4. Strain tensor (state of strain)

The strain tensor represents the pure deformation of the element. Let's investigate the symmetric part of the derivative tensor of the displacement.

$$\boldsymbol{A} = \frac{1}{2}(\boldsymbol{U} + \boldsymbol{U}^T) = \begin{bmatrix} \frac{\partial u}{\partial x} & \frac{1}{2}\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right) & \frac{1}{2}\left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right) \\ \frac{1}{2}\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right) & \frac{\partial v}{\partial y} & \frac{1}{2}\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right) \\ \frac{1}{2}\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right) & \frac{1}{2}\left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z}\right) & \frac{\partial w}{\partial z} \end{bmatrix} = \begin{bmatrix} \varepsilon_x & \frac{1}{2}\gamma_{xy} & \frac{1}{2}\gamma_{xz} \\ \frac{1}{2}\gamma_{yx} & \varepsilon_y & \frac{1}{2}\gamma_{yz} \\ \frac{1}{2}\gamma_{zx} & \frac{1}{2}\gamma_{zy} & \varepsilon_z \end{bmatrix} = \boldsymbol{A}^T, \tag{2.22}$$

where ε_x, ε_y and ε_z are the normal strains in the x, y and z directions, respectively, and $\varepsilon \ll 1$, $\gamma_{xy} = \gamma_{yx}$, $\gamma_{yz} = \gamma_{zy}$ and $\gamma_{zx} = \gamma_{xz}$ are the so called shear angles and $\gamma \ll 1$ considering small deformations. The normal strain has no unit, while the shear angle's unit is radian. If the $\varepsilon > 0$ the unit length elongates, if the $\varepsilon < 0$ the unit length shortens. If the $\gamma > 0$ the original 90° decreases, if the $\gamma < 0$ the original 90° increases. The deformation of an elementary point is demonstrated in Figure 2.4. We can also introduce the strain vectors $\boldsymbol{\alpha}_x$, $\boldsymbol{\alpha}_y$ and $\boldsymbol{\alpha}_z$ for the strain description,

$$\boldsymbol{A} = \begin{bmatrix} \varepsilon_x & \frac{1}{2}\gamma_{xy} & \frac{1}{2}\gamma_{xz} \\ \frac{1}{2}\gamma_{yx} & \varepsilon_y & \frac{1}{2}\gamma_{yz} \\ \frac{1}{2}\gamma_{zx} & \frac{1}{2}\gamma_{zy} & \varepsilon_z \end{bmatrix} = [\boldsymbol{\alpha}_x \quad \boldsymbol{\alpha}_y \quad \boldsymbol{\alpha}_z], \tag{2.23}$$

so

$$\begin{aligned} \boldsymbol{\alpha}_x &= \varepsilon_x \boldsymbol{i} + \frac{1}{2}\gamma_{yx}\boldsymbol{j} + \frac{1}{2}\gamma_{zx}\boldsymbol{k}, \\ \boldsymbol{\alpha}_y &= \frac{1}{2}\gamma_{xy}\boldsymbol{i} + \varepsilon_y \boldsymbol{j} + \frac{1}{2}\gamma_{zy}\boldsymbol{k}, \\ \boldsymbol{\alpha}_z &= \frac{1}{2}\gamma_{xz}\boldsymbol{i} + \frac{1}{2}\gamma_{yz}\boldsymbol{j} + \varepsilon_z \boldsymbol{k}. \end{aligned} \tag{2.24}$$

The strain tensor can be written in dyadic form using strain vectors

$$\boldsymbol{A} = \boldsymbol{\alpha}_x \circ \boldsymbol{i} + \boldsymbol{\alpha}_y \circ \boldsymbol{j} + \boldsymbol{\alpha}_z \circ \boldsymbol{k}. \tag{2.25}$$

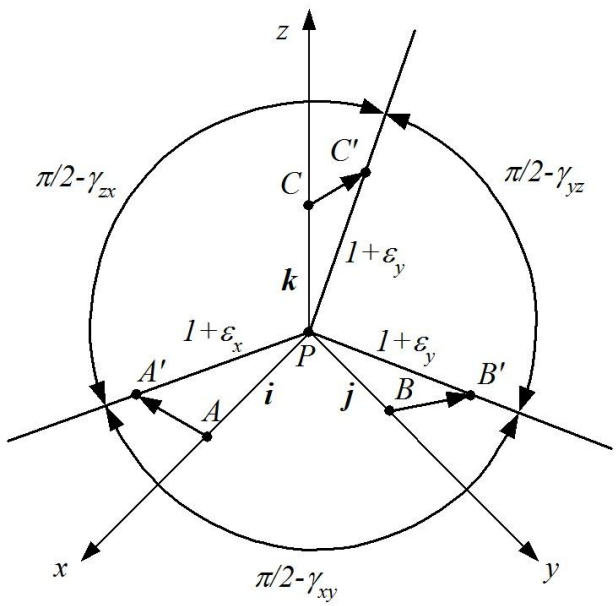


Figure 2.4. Deformation of an elementary point

In Figure 2.4 for the geometrical illustration of the strain state of a body point P we have to order a so called small cube denoted by the $\boldsymbol{i}$, $\boldsymbol{j}$ and $\boldsymbol{k}$ unit vectors in the very small domain of P with end points A, B and C.

2.1.5. Stress tensor (state of stress)

Consider a general deformable body loaded by an equilibrium force system, see in Figure 2.1. Let's pass through this body by a hypothetical plane and neglect one part of the body. Both the remaining part and the neglected part of the body have to be in equilibrium. There is acting force system distributed along the plane which effects this equilibrium. Furthermore, this distributed force system must be equivalent to the force system acting on the neglected part of the body. Note that the interface is common and equal where the so called split is applied, see in Figure 2.5. The intensity vector of this distributed force system is called stress vector $\boldsymbol{\varrho}$. The unit vector of the cross section is $\boldsymbol{n}$. We can state that we know the state of stresses with respect to the point of interest, if we know the stress vector at any cross section. Note, if the body is cut by another arbitrary hypothetical plane going through the body, it will result in another distributed force system having the same properties discussed above.
The Newton's third law of action and reaction is satisfied and can be expressed by

$$\boldsymbol{\varrho}(-\boldsymbol{n}) = -\boldsymbol{\varrho}(\boldsymbol{n}). \tag{2.26}$$

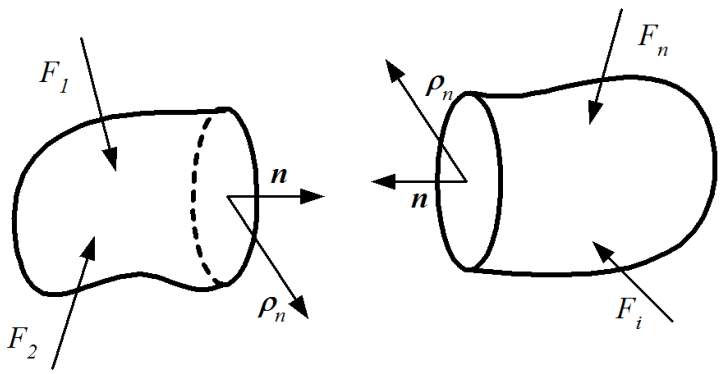


Figure 2.5. Introduction of the stress vector

The normal component is called normal stress σ_n, the component which is lying on the cross section is called shear stress τ. We can demonstrate the stress vectors belonging to the Cartesian coordinate system, thus we get the components of $\boldsymbol{\varrho}_x$, $\boldsymbol{\varrho}_y$ and $\boldsymbol{\varrho}_z$,

$$\begin{aligned} \boldsymbol{\varrho}_x &= \sigma_x \boldsymbol{i} + \tau_{yx} \boldsymbol{j} + \tau_{zx} \boldsymbol{k}, \\ \boldsymbol{\varrho}_y &= \tau_{xy} \boldsymbol{i} + \sigma_y \boldsymbol{j} + \tau_{zy} \boldsymbol{k}, \\ \boldsymbol{\varrho}_z &= \tau_{xz} \boldsymbol{i} + \tau_{yz} \boldsymbol{j} + \sigma_z \boldsymbol{k}. \end{aligned} \tag{2.27}$$

The first index in the case of shear stresses denotes the normal axis to the cross section, while the second one denotes the direction of the component. The normal components have only one subscript.
These components can be collected into a special tensor called stress tensor $\boldsymbol{T}$,

$$\boldsymbol{T} = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} = [\boldsymbol{\varrho}_x \quad \boldsymbol{\varrho}_y \quad \boldsymbol{\varrho}_z] = \boldsymbol{T}^T. \tag{2.28}$$

The stress tensor is symmetrical, that means the shear stresses at a point with reversed subscripts must be equal to each other, $\tau_{xy} = \tau_{yx}$, $\tau_{yz} = \tau_{zy}$ and $\tau_{zx} = \tau_{xz}$.

The stress tensor can be written in dyadic form using the stress vector

$$\boldsymbol{T} = \boldsymbol{\varrho}_x \circ \boldsymbol{i} + \boldsymbol{\varrho}_y \circ \boldsymbol{j} + \boldsymbol{\varrho}_z \circ \boldsymbol{k}. \tag{2.29}$$

Considering the Cauchy's theorem, the Cauchy stress vector can be determined

$$\boldsymbol{\varrho}_n = \boldsymbol{T}\boldsymbol{n}. \tag{2.30}$$

The generally used unit of the stress is $\frac{N}{mm^2} = MPa$.

2.1.6. Principal values of normal stresses, scalar invariants and equivalent stress

In finite element calculations we need to able to compare the allowable stress of the material used with the calculated stress. In general loading case the stress tensor is a full tensor, so an equivalent stress measure has to be determined for the comparison. There are different sizing theories for equivalent stress.
The question is how we can find those planes where the stress vector is parallel to the normal vector of the plane, thus

$$\boldsymbol{T}\boldsymbol{n} = \lambda \boldsymbol{n}, \tag{2.31}$$

or

$$(\boldsymbol{T} - \lambda \boldsymbol{I})\boldsymbol{n} = \boldsymbol{0}, \tag{2.32}$$

where $\boldsymbol{I}$ is the unit tensor. In mathematics this is called eigenvector, eigenvalue problem. The characteristic equation can be written

$$-\lambda^3 + T_I\lambda^2 - T_{II}\lambda + T_{III} = 0, \tag{2.33}$$

where T_I, T_{II} and T_{III} are the first, second and third scalar invariants of the stress tensor, respectively,

$$\begin{gathered} T_I = \sigma_x + \sigma_y + \sigma_z = \sigma_1 + \sigma_2 + \sigma_3, \\ T_{II} = \begin{vmatrix} \sigma_y & \tau_{yz} \\ \tau_{zy} & \sigma_z \end{vmatrix} + \begin{vmatrix} \sigma_x & \tau_{xz} \\ \tau_{zx} & \sigma_z \end{vmatrix} + \begin{vmatrix} \sigma_x & \tau_{xy} \\ \tau_{yx} & \sigma_y \end{vmatrix} = \sigma_2\sigma_3 + \sigma_1\sigma_3 + \sigma_1\sigma_2, \\ T_{III} = \det(\boldsymbol{T}) = \sigma_1\sigma_2\sigma_3. \end{gathered} \tag{2.34}$$

The solution results

$$\boldsymbol{T}(\boldsymbol{i},\boldsymbol{j},\boldsymbol{k}) = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \rightarrow \boldsymbol{T}(\boldsymbol{1},\boldsymbol{2},\boldsymbol{3}) = \begin{bmatrix} \sigma_1 & 0 & 0 \\ 0 & \sigma_2 & 0 \\ 0 & 0 & \sigma_3 \end{bmatrix}, \tag{2.35}$$

where $\sigma_1 \geq \sigma_2 \geq \sigma_3$ are the principal values of the normal stresses in $\boldsymbol{1}$, $\boldsymbol{2}$ and $\boldsymbol{3}$ principal axes, respectively.
In general case $\boldsymbol{T}$ and $\boldsymbol{A}$ are known for a combined load. The most common material test is the tensile one from which the allowable stress σ_{all} may be determined. The question is, how we can find connection with the tensile test? The connection is the equivalent stress σ_{red}. It is

important to mention that the equivalence does not mean the equivalence from statical point of view. It means only the material gets damaged at the same time for the two cases (combined case and simple tension). This is an equivalence of the simple tension and the combined stress state from the strength of materials point of view. Most finite element software uses the Huber-Mises-Hencky theory for equivalent stress from Mises σ_{red}(von Mises), which can be determined in the coordinate system of the principal axes

$$\sigma_{red}(\text{von Mises}) = \sqrt{\frac{1}{2}(\sigma_1 - \sigma_2)^2 + \frac{1}{2}(\sigma_2 - \sigma_3)^2 + \frac{1}{2}(\sigma_3 - \sigma_1)^2} \geq 0. \tag{2.36}$$

When checking the most dangerous point of the examined body the well known stressing assumption has to be satisfied,

$$\sigma_{red}^{max}(\text{von Mises}) \leq \sigma_{all}. \tag{2.37}$$

A general elasticity problem involves 15 unknowns including 3 displacements (u, v, w), 6 strains $(\varepsilon_x, \varepsilon_y, \varepsilon_z, \gamma_{xy}, \gamma_{yz}, \gamma_{zx})$ and 6 stresses $(\sigma_x, \sigma_y, \sigma_z, \tau_{xy}, \tau_{yz}, \tau_{zx})$ which have to be determined.

2.2. The basic equation system and the boundary conditions

2.2.1. Kinematic equations

The equation for the derivation of the strain tensor is the kinematic equation or the strain-displacement relation,

$$\boldsymbol{A} = \frac{1}{2}(\boldsymbol{U} + \boldsymbol{U}^T) = \frac{1}{2}(\boldsymbol{u} \circ \nabla + \nabla \circ \boldsymbol{u}). \tag{2.38}$$

This form of the kinematic equation is valid for only small deformation. It is a tensor equation, so written in scalar notation we get

$$\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x}, \\
\varepsilon_y &= \frac{\partial v}{\partial y}, \\
\varepsilon_z &= \frac{\partial w}{\partial z}, \\
\gamma_{xy} = \gamma_{yx} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}, \\
\gamma_{yz} = \gamma_{zy} &= \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}, \\
\gamma_{zx} = \gamma_{xz} &= \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}.
\end{aligned} \tag{2.39}$$

These 6 independent components completely describe the small displacement theory. The kinematic equations give the relation between the displacement field and the strain field.

2.2.2. Constitutive equations

Now let's investigate the material response. Relations characterizing the physical properties of materials are called constitutive equations. Wide variety of materials is used, so the development of constitutive equations is one of the most researched fields in mechanics. The constitutive laws are developed through empirical relations based on experiments. The mechanical behaviour of solid materials is defined by stress-strain relations. The relations express the stress as a function of the strain. In this case the elastic solid material model is chosen. This model describes a deformable solid continuum that recovers its original shape when the loadings causing the deformation are removed. Furthermore we investigate only the constitutive law which is linear leading to a linear elastic solid. Many structural materials including metals, plastics, wood, concrete exhibit linear elastic behavior under small deformations. The stress-strain relation of a linear elastic material can be seen in Figure 2.6.

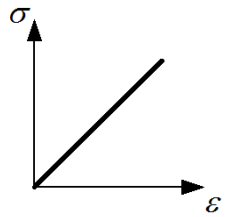


Figure 2.6. Linear elastic material model

The well known Hooke's law is valid for linear elastic material description. In the case of single axis stress state the simplified Hooke's law is

$$\sigma = E\varepsilon, \tag{2.40}$$

where E is the Young's modulus, which is a real material constants. In the case of torsion the simplified Hooke's law is

$$\tau = G\gamma, \tag{2.41}$$

where G is the shear modulus. Considering linear elastic material

$$E = 2G(1+\nu) \tag{2.42}$$

is valid, where ν is the Poisson ratio which describes the relationship between elongation and contraction.
In general case the Hooke's law can be expressed as tensor equations which is valid for arbitrary coordinate systems,

$$\boldsymbol{T} = 2G\left(\boldsymbol{A} + \frac{\nu}{1-2\nu}A_I\boldsymbol{I}\right), \tag{2.43}$$

or

$$\boldsymbol{A} = \frac{1}{2G}\left(\boldsymbol{T} - \frac{\nu}{1+\nu}T_I\boldsymbol{I}\right), \tag{2.44}$$

where A_I is the first scalar invariant of the strain tensor,

$$A_I = \varepsilon_x + \varepsilon_y + \varepsilon_z = \varepsilon_1 + \varepsilon_2 + \varepsilon_3. \tag{2.45}$$

The Eq. 2.43 can be written in scalar notation and we get

$$\begin{gathered}
\sigma_x = 2G\left[\varepsilon_x + \frac{\nu}{1-2\nu}\left(\varepsilon_x + \varepsilon_y + \varepsilon_z\right)\right], \\
\sigma_y = 2G\left[\varepsilon_y + \frac{\nu}{1-2\nu}\left(\varepsilon_x + \varepsilon_y + \varepsilon_z\right)\right], \\
\sigma_z = 2G\left[\varepsilon_z + \frac{\nu}{1-2\nu}\left(\varepsilon_x + \varepsilon_y + \varepsilon_z\right)\right], \\
\tau_{xy} = G\gamma_{xy}, \\
\tau_{yz} = G\gamma_{yz}, \\
\tau_{zx} = G\gamma_{zx}.
\end{gathered} \tag{2.46}$$

The Eq. 2.44 can be written in scalar notation and we get

$$\begin{gathered}
\varepsilon_x = \frac{1}{2G}\left[\sigma_x - \frac{\nu}{1+\nu}\left(\sigma_x + \sigma_y + \sigma_z\right)\right], \\
\varepsilon_y = \frac{1}{2G}\left[\sigma_y - \frac{\nu}{1+\nu}\left(\sigma_x + \sigma_y + \sigma_z\right)\right], \\
\varepsilon_z = \frac{1}{2G}\left[\sigma_z - \frac{\nu}{1+\nu}\left(\sigma_x + \sigma_y + \sigma_z\right)\right], \\
\gamma_{xy} = \frac{\tau_{xy}}{G}, \\
\gamma_{yz} = \frac{\tau_{yz}}{G}, \\
\gamma_{zx} = \frac{\tau_{zx}}{G}.
\end{gathered} \tag{2.47}$$

For isotropic linear elastic material these relations can be described in matrix form,

$$\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-2\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-2\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 1-2\nu \end{bmatrix} \begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix}, \tag{2.48}$$

or

$$\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix} = \begin{bmatrix} \frac{1}{E} & -\frac{\nu}{E} & -\frac{\nu}{E} & 0 & 0 & 0 \\ -\frac{\nu}{E} & \frac{1}{E} & -\frac{\nu}{E} & 0 & 0 & 0 \\ -\frac{\nu}{E} & -\frac{\nu}{E} & \frac{1}{E} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G} \end{bmatrix} \begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix}. \tag{2.49}$$

The 6 independent equations give the relation between the stress and strain.

2.2.3. Equilibrium equations

In order to derive the differential static equilibrium equations, consider a general deformable body defined by a volume V. The stress state in a deformable solid is continuously distributed within the body and determined from the applied loadings. The applied loadings satisfy the equations of static equilibrium, so the sum of the forces and moments is zero.
Consider a closed sub-domain with volume V and surface A within a body in equilibrium, see in Figure 2.7.

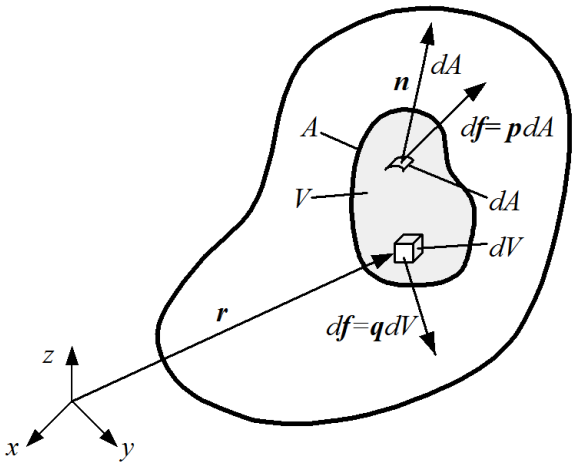


Figure 2.7. Body and surface forces of a closed sub-domain of a deformable body

The mechanical effects of the very small domain of V are considered with forces, so the force acting on the subdomain V can be expressed

$$d\boldsymbol{f} = \boldsymbol{q}dV, \tag{2.50}$$

where $\boldsymbol{q}$ is the body forces per unit volume, the force acting on the subsurface V can be expressed

$$d\boldsymbol{f} = \boldsymbol{p}dA = \boldsymbol{T}\boldsymbol{n}dA. \tag{2.51}$$

The closed sub-domain is in equilibrium, so

$$\int_{(V)} \boldsymbol{q}dV + \int_{(A)} \boldsymbol{T}\boldsymbol{n}dA = \boldsymbol{0}. \tag{2.52}$$

The second term of this equation can be transformed into a volume integral by using the Gauss-Osztrogradszkij theorem to give

$$\int_{(V)} \boldsymbol{q}dV + \int_{(V)} \boldsymbol{T}\nabla dV = \boldsymbol{0}, \tag{2.53}$$

so

$$\int_{(V)} (\boldsymbol{T}\nabla + \boldsymbol{q})dV = \boldsymbol{0}. \tag{2.54}$$

Because the region V is arbitrary and the integrand is continuous, the integrand must vanish

$$\boldsymbol{T}\nabla + \boldsymbol{q} = \boldsymbol{0}. \tag{2.55}$$

This equation is the equation of equilibrium, which can be written considering the Cartesian coordinates into the following form

$$\begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{bmatrix} \begin{bmatrix} \dfrac{\partial}{\partial x} \\ \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial z} \end{bmatrix} + \begin{bmatrix} q_x \\ q_y \\ q_z \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ 0 \end{bmatrix}. \tag{2.56}$$

Written in scalar notation we get

$$\begin{aligned} \frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} + q_x &= 0, \\ \frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} + q_y &= 0, \\ \frac{\partial \tau_{zx}}{\partial x} + \frac{\partial \tau_{zy}}{\partial y} + \frac{\partial \sigma_z}{\partial z} + q_x &= 0. \end{aligned} \tag{2.57}$$

The equations of the equilibrium give the relation between the body force and the stress field. Now we have 15 scalar equations (6 kinematic equations, 6 constitutive equations and 3 equilibrium equations) to determine the 15 unknowns.

2.2.4. Boundary conditions

It is very essential to discuss typical boundary conditions connected with the elastic model. The solution of the general system of equations requires appropriate boundary conditions on the body which is examined. The common types of boundary conditions in elastic problems are normally how the body is supported or loaded. This concept is formulated by specifying the displacements and stresses at boundary points, see in Figure 2.8. The boundary conditions play a very essential role in properly formulating and solving elasticity problems, so it is important to acquire a clear understanding of their specification. Improper specification results in a solution to a different problem or no solution.

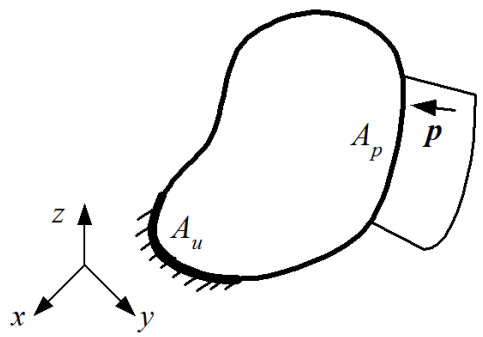


Figure 2.8. Typical boundary conditions

The surface of the body is A and it is valid that

$$A = A_u \cup A_p, \quad A \cap A_p = 0. \tag{2.58}$$

The displacement type boundary condition (Dirichlet type boundary condition) can be described by

$$\boldsymbol{u} = \widetilde{\boldsymbol{u}}, \quad \boldsymbol{r} \in A_u, \tag{2.59}$$

where $\widetilde{\boldsymbol{u}}$ is the prescribed displacement field (at fixed boundaries $\widetilde{\boldsymbol{u}} = \boldsymbol{0}$),

$$u = \tilde{u}, \quad v = \tilde{v}, \quad w = \tilde{w}. \tag{2.60}$$

The force type boundary condition (Neumann type boundary conditions) can be described by

$$\boldsymbol{T} \cdot \boldsymbol{n} = \widetilde{\boldsymbol{p}}, \quad \boldsymbol{r} \in A_p, \tag{2.61}$$

where $\widetilde{\boldsymbol{p}}$ is the prescribed force vector. The Eq. 2.61 in scalar notation is

$$\begin{aligned} \sigma_x n_x + \tau_{xy} n_y + \tau_{xz} n_z &= \tilde{p}_x, \\ \tau_{yx} n_x + \sigma_y n_y + \tau_{yz} n_z &= \tilde{p}_y, \\ \tau_{zx} n_x + \tau_{zy} n_y + \sigma_z n_z &= \tilde{p}_z. \end{aligned} \tag{2.62}$$

However, this general system of equations is too complex, and the solution by using analytical methods are impossible and further simplification is needed to solve these elastic boundary value problems.
The implementation of the solution can be done in two types of system, one is the primal system, when the displacement field is the basic variable (displacement based system - ~99.9% of the finite element softwares). In this case the kinematic boundary condition must be met the displacement field. The solution order is the next:

$\boldsymbol{u}$ → (kinematic equation) → $\boldsymbol{A}$ → (Hooke's law) → $\boldsymbol{T}$.

The other one is the so called dual system, when the stress tensor field is the basic variable (stress based system). In this case the dynamic boundary condition must be met the stress tensor field. The solution order is the next:

$\boldsymbol{T}$ → (Hooke's law) → $\boldsymbol{A}$ → (Cesaro-form) → $\boldsymbol{u}$.

These systems are equivalent with each other.

2.3. Analytical solution of a one-dimensional boundary value problem

There are some general solution strategies for solving elasticity problems. The so called analytical method seeks the way how to determine the solution of the boundary value problem by direct integration of the field equations. It is very essential that the boundary conditions have to be correctly satisfied. The complex problem solution encounters significant mathematical difficulties, thus only the analytical method is recommended for simple elasticity problems.

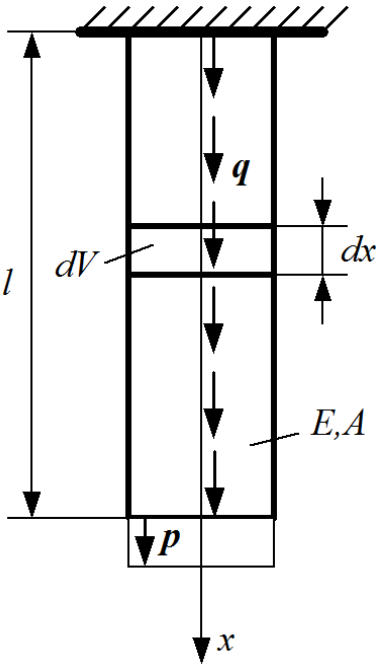


Figure 2.9. The elasticity problem

Consider the case of a uniform prismatic bar which is loaded by its own weight and distributed force system along the flange. The bar is fixed at the other end. The material of the bar is homogeneous isotropic and linear elastic. The problem is defined in Figure 2.9.
Assuming that in any arbitrary cross section of the bar uniformly distributed normal stress is arising. This assumption makes this problem one-dimensional. The mechanical model derived from the original problem is illustrated in Figure 2.10.

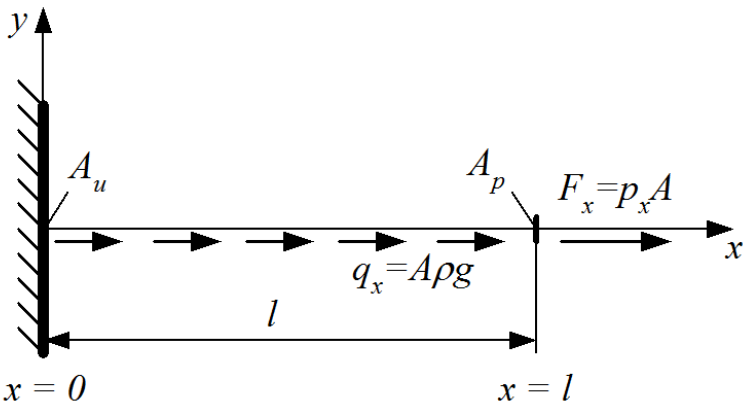


Figure 2.10. The one-dimensional mechanical model of the bar

In Figure 2.9 and in Figure 2.10 the applied notations are the following: A is the cross section of the bar, l is the length of the bar, E is the Young's modulus, ϱ is the material mass density, $\boldsymbol{g}$ is acceleration of gravity, $\boldsymbol{q}$ is the distributed force system intensity along the volume, $\boldsymbol{p}$ is the distributed force system intensity along the plane, dV and dx are the elementary bar volume and the elementary bar length, respectively, while A_u is the kinematic boundary (prescribed displacement) and A_p is the dynamic boundary (loaded cross section).
The task is to determine the displacement $u(x)$. The related kinematic equation, constitutive equation and equilibrium equation of the elastic boundary value problem with the correctly set boundary conditions are used.

From Figure 2.9 and from Figure 2.10

$$\boldsymbol{F} = \boldsymbol{p}A = p_x \boldsymbol{i} A \;\rightarrow\; F_x = p_x A,$$
$$\boldsymbol{q} = q_x \boldsymbol{i} = A\varrho g \boldsymbol{i}.$$

The related kinematic equation Eq. 2.39 is

$$\varepsilon_x = \frac{du(x)}{dx}, \quad 0 < x < l.$$

Using the Hooke's law Eq. 2.40 we get

$$\sigma_x = E\varepsilon_x \;\rightarrow N = \sigma_x A = AE\varepsilon_x = AE\frac{du(x)}{dx}, \quad 0 < x < l.$$

Now the equilibrium of the elementary bar length is investigated, see in Figure 2.11, to be able to derive the related equilibrium equation.

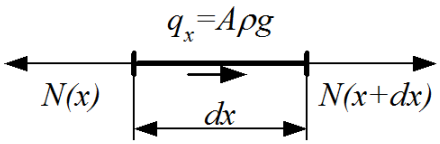


Figure 2.11. The equilibrium of the elementary bar length

Assuming the equilibrium we get

$$-N(x) + N(x + dx) + q_x dx = 0.$$

The normal force $N(x + dx)$ developed in a series until the linear term

$$N(x + dx) = N(x) + \frac{dN}{dx}dx + \cdots$$

can be substituted which gives the equilibrium equation

$$-N(x) + N(x) + \frac{dN}{dx}dx + q_x dx = 0$$
$$\frac{dN}{dx}dx + q_x dx = 0 \qquad /: dx$$
$$\frac{dN}{dx} + q_x = 0.$$

At A_u the displacement is known, so

$$u(0) = 0,$$

while at A_p the loading is known which means

$$N(l) = AE\left.\frac{du}{dx}\right|_{x=l} = F_x.$$

<u>The analytical solution:</u>

$$\frac{dN}{dx} + q_x = 0,$$
$$\frac{dAE\frac{du(x)}{dx}}{dx} + q_x = 0,$$

Since the bar is homogeneous and prismatic

$$AE\frac{d^2u}{dx^2} + q_x = 0,$$

After reordering the equation the equation has to be integrated according to x two times

$$\frac{d^2u}{dx^2} = -\frac{q_x}{AE} \qquad / \int dx$$
$$\frac{du}{dx} = -\frac{q_x}{AE}x + C_1 \qquad / \int dx$$
$$u(x) = -\frac{q_x}{2AE}x^2 + C_1x + C_2$$

The next step is to determine the integral constants C_1 and C_2 from the boundary conditions, according to the kinematic boundary condition,

$$u(0) = -\frac{q_x}{2AE}0^2 + C_1 0 + C_2 \;\rightarrow\; C_2 = 0.$$

From the dynamic boundary condition,

$$N(l) = AE\frac{du}{dx}(l) = F_x$$
$$AE(-\frac{q_x}{AE}l + C_1) = F_x$$
$$-q_x l + AEC_1 = F_x \;\rightarrow\; AEC_1 = q_x l + F_x \;\rightarrow\; C_1 = \frac{q_x l + F_x}{AE}$$

Substituting C_1 and C_2 into the $u(x)$ we get the sought displacement function

$$u(x) = -\frac{q_x}{2AE}x^2 + \frac{q_x l + F_x}{AE}x.$$

Further mechanical quantities can be determined from $u(x)$ like normal strain,

$$\varepsilon(x) = \frac{du}{dx} = -\frac{q_x}{AE}x + \frac{q_x l + F_x}{AE},$$

and the normal force,

$$N(x) = AE\frac{du(x)}{dx} = AE\varepsilon(x) = F_x + q_x(l - x).$$

Example 1.

Uniform prismatic bar having circular cross section is loaded by its own weight and by distributed force system along the flange. The bar is fixed at the other end. The material of the bar is aluminum. The problem is illustrated in Figure 2.9.

Data:

$d = 30mm$

$E = 6{,}9 \cdot 10^4 \frac{N}{mm^2}$

$l = 1000mm$

$\varrho = 2700 \frac{kg}{m^3}$

$F_x = 5N$

Results:

$$A = \frac{d^2\pi}{4} = 706{,}85mm^2$$

$$q_x = A\varrho g = 0{,}01872 \frac{N}{mm}$$

x [mm]	u [mm]	N [N]	σ [MPa]
0	0	23,72256	0,033560553
100	4,67191E-05	21,8503	0,030911853
200	8,95996E-05	19,97805	0,028263153
300	0,000128641	18,10579	0,025614453
400	0,000163844	16,23353	0,022965753
500	0,000195209	14,36128	0,020317053
600	0,000222734	12,48902	0,017668353
700	0,000246421	10,61677	0,015019653
800	0,00026627	8,744511	0,012370953
900	0,000282279	6,872256	0,009722253
1000	0,00029445	5	0,007073553

Table 2.1. The displacement field, the normal forces and the stress field

In Figure 2.12 the results can be seen.

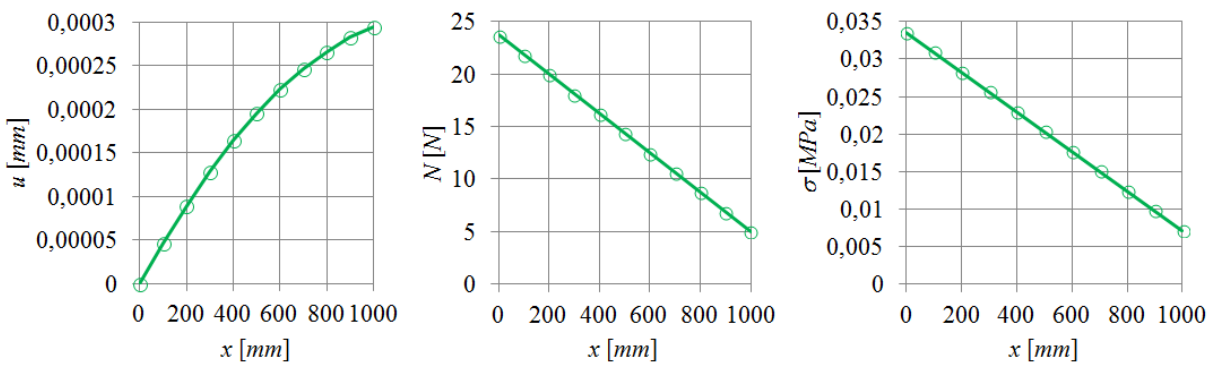


Figure 2.12. The displacements, normal forces and stresses

3. STRAIN ENERGY AND RELATED PRINCIPLES

3.1. Strain energy

The work W done by surface and body forces on an elastic solid are stored inside the body in the form of strain energy U. For an idealized elastic body, the stored energy is completely recoverable when the elastic solid is returned to its original unloaded configuration.
Let us consider a simple case when a cylinder is subjected to tension (for example the tensile test), see in Figure 3.1.

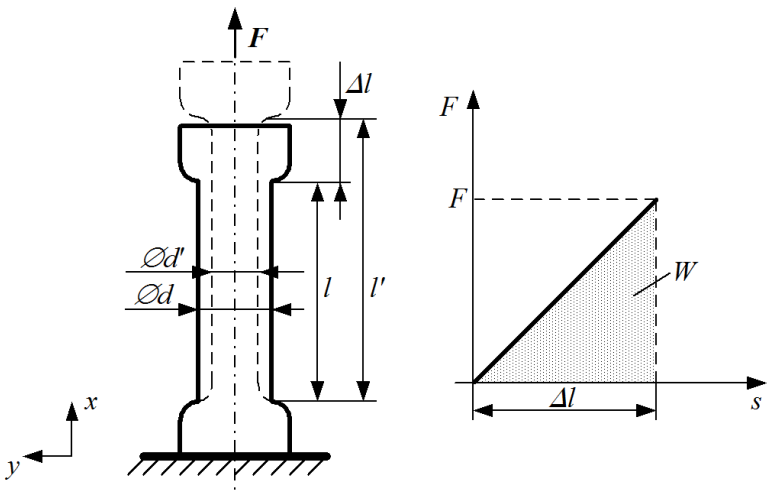


Figure 3.1. The tensile test and the characteristics

In this simple uniform uniaxial deformation case there is no body forces assumed. During the loading process we assume that the stress is slowly increasing from zero to a discrete σ_x value. The inertia effects are neglected. The strain energy stored is equal with the work done on the cylinder.

$$W = \int_0^s F(s)\,ds = \frac{1}{2}F\Delta l = \frac{1}{2}\frac{F}{A}\frac{\Delta l}{l}Al = \frac{1}{2}\sigma_x\varepsilon_x V = U. \tag{3.1}$$

Introducing the strain energy per unit volume (strain energy density) u,

$$U = \int_V u\,dV = \frac{1}{2}\int_V \sigma_x\varepsilon_x\,dV. \tag{3.2}$$

In general case the strain energy per unit volume can be described by

$$u = \frac{1}{2}\boldsymbol{T}\cdot\cdot\boldsymbol{A} = \frac{1}{2}\begin{bmatrix}\sigma_x & \tau_{xy} & \tau_{xz}\\ \tau_{yx} & \sigma_y & \tau_{yz}\\ \tau_{zx} & \tau_{zy} & \sigma_z\end{bmatrix}\begin{bmatrix}\varepsilon_x & \frac{1}{2}\gamma_{xy} & \frac{1}{2}\gamma_{xz}\\ \frac{1}{2}\gamma_{yx} & \varepsilon_y & \frac{1}{2}\gamma_{yz}\\ \frac{1}{2}\gamma_{zx} & \frac{1}{2}\gamma_{zy} & \varepsilon_z\end{bmatrix} = \tag{3.3}$$

$$= \frac{1}{2}\left[\sigma_x\varepsilon_x + \sigma_y\varepsilon_y + \sigma_z\varepsilon_z + \tau_{xy}\gamma_{xy} + \tau_{yz}\gamma_{yz} + \tau_{zx}\gamma_{zx}\right],$$

where the symbol "··" denotes the inner product of the tensors. In Eq. 3.3 it can be seen that the strain energy in general case is

$$U = \frac{1}{2}\int_V \boldsymbol{T} \cdot\cdot \boldsymbol{A}\, dV. \tag{3.4}$$

It can be observed that the strain energy is a positive definite quadratic form with the property

$$U \geq 0 \tag{3.5}$$

for all values of σ and ε. The equality is the case when $\sigma = 0$ and $\varepsilon = 0$.

3.2. Total potential energy

The total potential energy includes the stored elastic potential energy (strain energy) as well as the potential energy of applied loads. The general deformable elastic solid body can be seen in Figure 3.2.

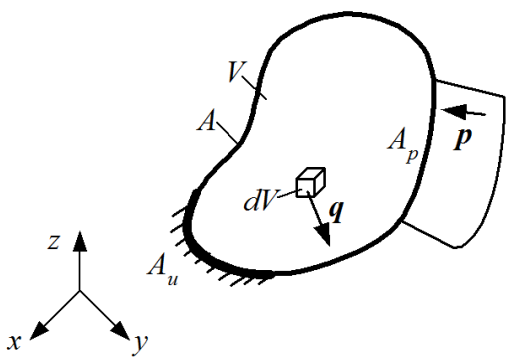


Figure 3.2. Deformable elastic solid body

The work of the external force system can be described by

$$W = \int_V \boldsymbol{uq}\, dV + \int_{A_p} \boldsymbol{up}\, dA, \tag{3.6}$$

where the first term is the work of the volumetric force system, the second term is the work of the surface force system. Here, we deal only with elastic systems subjected to conservative forces. A conservative force does mechanical work independent of the path of motion and the work is reversible or recoverable. The total potential energy Π is given by

$$\Pi = U - W. \tag{3.7}$$

Using Eq. 3.4 the total potential energy of linear elastic solid body is

$$\Pi(\boldsymbol{u}) = U - W = \frac{1}{2}\int_V \boldsymbol{T} \cdot\cdot \boldsymbol{A}\, dV - \int_V \boldsymbol{uq}\, dV - \int_{A_p} \boldsymbol{up}\, dA. \tag{3.8}$$

The total potential energy is a function which gives different scalar values in the case of different displacement fields.

3.3. Approximate solutions and calculus of variation

In the case of complex elastic boundary value problem only the approximate solution is available. However, against the approximate solution we can formulate expectations.
The displacement field $\boldsymbol{u}^*$ is kinematically admissible if $\boldsymbol{u}^*$ is continuous, differentiable and satisfies the kinematic boundary conditions. The stress field $\boldsymbol{T}^*$ is statically admissible if $\boldsymbol{T}^*$ satisfies the equilibrium equation and the dynamic boundary conditions.

3.3.1. Calculus of variation

The $\delta u(x)$ is the variation of the function $u(x)$. The $\delta u(x)$ means the deviation from the function $u(x)$. The variation of the displacement field is

$$u^*(x) = u(x) + \delta u(x). \tag{3.9}$$

From Figure 3.3 it can be seen that $u^*(x)$ satisfies the kinematic boundary condition, thus $u^*(0) = u(0)$ and $\delta u(0) = 0$.

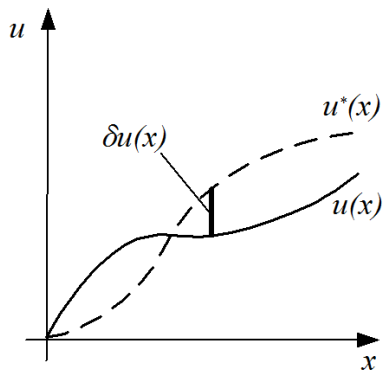


Figure 3.3. Interpretation of variation

A principle is needed to produce the approximate solution. Respect to the solution the chosen principle should serve the best approximation from the given function space.

3.3.2. Variation principles

There are numerous variational principles and they are used within the calculus of variation. If we want to compare the direct solution of the differential equation system to the investigation using variational principles of linear elastic solid problems, the application of the variational principles have the following advantages:

- the function connected to the variational principle mostly has physical meaning,
- the goodness of the approximation can be estimated,
- the existence of the solution can be demonstrated with the usage of variational principles,
- complex boundary conditions can be introduced with the usage of variational principles,
- based on the function having has physical meaning numerically stable procedures can be derived,
- etc.

3.4. Principle of minimum total potential energy

The definition of the principle of minimum total potential energy: in displacements satisfying the given boundary conditions of an elastic solid and the equilibrium equations the potential energy has a local minimum.
Kinematically admissible total potential energy Π^* can be calculated for the kinematically admissible displacement field $\boldsymbol{u}^*$, thus

$$\Pi^* = \Pi^*(\boldsymbol{u}^*) = \frac{1}{2}\int_V \boldsymbol{T}^* \cdot\cdot \boldsymbol{A}^*\, dV - \int_V \boldsymbol{u}^*\boldsymbol{q}\, dV - \int_{A_p} \boldsymbol{u}^*\boldsymbol{p}\, dA. \tag{3.10}$$

The principle of minimum total potential energy according to the definition,

$$\Pi^* - \Pi \geq 0. \tag{3.11}$$

$\Pi^* = \Pi$ is only when $\boldsymbol{A}^* = \boldsymbol{A}$ and $\boldsymbol{u}^* = \boldsymbol{u}$. We get the exact solution if the smallest kinematically admissible total potential energy is chosen. In this case $\Pi^*_{min} = \Pi$. We get only approximate solution if not the smallest kinematically admissible total potential energy is chosen. In this case $\Pi^*_{min} \neq \Pi$.

3.4.1. Langrange variational principle

The Lagrange variational principle is the variational formulation of the principle of minimum total potential energy. The total potential energy is a functional, thus

$$\Pi(\boldsymbol{u}) = U - W = \frac{1}{2}\int_V \boldsymbol{T} \cdot\cdot \boldsymbol{A}\, dV - \int_V \boldsymbol{u}\boldsymbol{q}\, dV - \int_{A_p} \boldsymbol{u}\boldsymbol{p}\, dA. \tag{3.12}$$

The $\boldsymbol{u}$ is known in the surface A_u, so the boundary condition is $\delta\boldsymbol{u}|_{A_u} = 0$
The necessary criterium of the extremum is $\delta\Pi = 0$,

$$\delta\Pi = \int_V \delta(\boldsymbol{T} \cdot\cdot \boldsymbol{A})\, dV - \int_V \delta\boldsymbol{u}\boldsymbol{q}\, dV - \int_{A_p} \delta\boldsymbol{u}\boldsymbol{p}\, dA = 0. \tag{3.13}$$

The sufficient condition of the minimum value is when $\delta^2\Pi \geq 0$. The principle of minimum total potential energy and the Lagrange variational principle have the same physical content.
It is important to prove that the exact solution calculated by the principle satisfies the equation system of the elasticity. The first term of Eq. 3.13 can be converted, thus

$$\int_V \delta(\boldsymbol{T} \cdot\cdot \boldsymbol{A})\, dV = \int_V \boldsymbol{T} \cdot\cdot \delta\boldsymbol{A} dV = \int_V \boldsymbol{T} \cdot\cdot \delta(\underbrace{\boldsymbol{u} \circ \nabla}_{\boldsymbol{U}}) dV = \int_V \boldsymbol{T} \cdot\cdot (\delta\boldsymbol{u} \circ \nabla) dV,$$

note, that

$$\int_V \boldsymbol{T} \cdot\cdot \delta(\underbrace{\boldsymbol{u} \circ \nabla}_{\boldsymbol{U}}) dV = \int_V \boldsymbol{T} \cdot\cdot \delta\boldsymbol{A} dV + \int_V \boldsymbol{T} \cdot\cdot \delta\boldsymbol{\Psi} dV = \int_V \boldsymbol{T} \cdot\cdot \delta\boldsymbol{A} dV\,.$$

The $\boldsymbol{T} \cdot\cdot \boldsymbol{\Psi}$=0 because of the definition, that the double scalar product of symmetric and asymmetric tensors are zero.

$$\int_V \boldsymbol{T} \cdot\cdot (\delta \boldsymbol{u} \circ \nabla) dV = \int_V [(\delta \boldsymbol{u} \cdot \boldsymbol{T} \cdot \nabla) - \delta \boldsymbol{u} \cdot (\boldsymbol{T} \cdot \nabla)] dV,$$

where

$$\int_V \delta \boldsymbol{u} \cdot \boldsymbol{T} \cdot \nabla \, dV = \int_A \delta \boldsymbol{u} \cdot \boldsymbol{T} \cdot \boldsymbol{n} \, dA = \int_{A_p} \delta \boldsymbol{u} \cdot \boldsymbol{T} \cdot \boldsymbol{n} \, dA + \underbrace{\int_{A_u} \underbrace{\delta \boldsymbol{u}}_{0} \cdot \boldsymbol{T} \cdot \boldsymbol{n} \, dA}_{0}.$$

Finally,

$$\int_V \delta (\boldsymbol{T} \cdot\cdot \boldsymbol{A}) \, dV = \int_{A_p} \delta \boldsymbol{u} \cdot \boldsymbol{T} \cdot \boldsymbol{n} \, dA - \int_V \delta \boldsymbol{u} \cdot (\boldsymbol{T} \cdot \nabla) \, dV.$$

After the conversion and substitution into Eq. 3.13 we get

$$\delta \Pi = -\int_V \delta \boldsymbol{u} \cdot (\boldsymbol{T} \cdot \nabla) \, dV - \int_V \delta \boldsymbol{u} \boldsymbol{q} \, dV + \int_{A_p} \delta \boldsymbol{u} \cdot \boldsymbol{T} \cdot \boldsymbol{n} \, dA - \int_{A_p} \delta \boldsymbol{u} \boldsymbol{p} \, dA = 0. \qquad (3.14)$$

The Eq. 3.14 is combinable, from which

$$\delta \Pi = -\int_V \delta \boldsymbol{u} \cdot (\boldsymbol{T} \cdot \nabla + \boldsymbol{q}) \, dV + \int_{A_p} \delta \boldsymbol{u} \cdot (\boldsymbol{T} \cdot \boldsymbol{n} - \boldsymbol{p}) \, dA = 0. \qquad (3.15)$$

While $\delta \boldsymbol{u}$ is arbitrary, the terms in the brackets have to be zero, thus the principle contains the equilibrium equations and the dynamic boundary conditions.

3.5. Linear spring

A linear elastic spring is a mechanical device which is capable of supporting only axial loading. The elongation or contraction of the spring is proportional to the applied axial load. The application of principle of minimum total potential energy can be introduced by linear spring problems.

Example 2.
Consider a simple linear spring of stiffness k and applied load F, see in Figure 3.4. Let u be the displacement of the spring under the load.

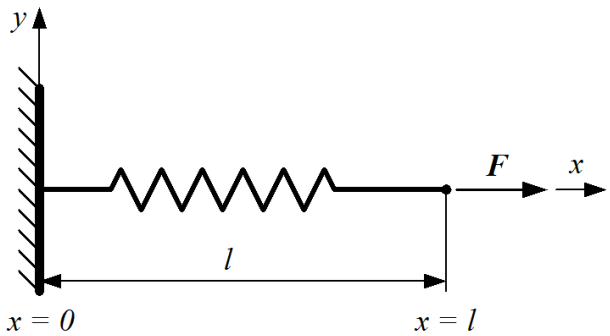


Figure 3.4. Linear spring

The total potential energy of the system is

$$\Pi = U - W = \frac{1}{2}ku^2 - Fu,$$

the minimum of the extremum problem is

$$\delta\Pi = \frac{d\Pi}{du} = 0 = ku - F \ \rightarrow u = \frac{F}{k}$$

and

$$\frac{d^2\Pi}{du^2} = k > 0.$$

For the graphical illustration (see Figure 3.5.) of the minimum potential energy,

$$\Pi = u\left(\frac{1}{2}ku - F\right) \rightarrow \ u_1 = 0, u_2 = \frac{2F}{k}$$

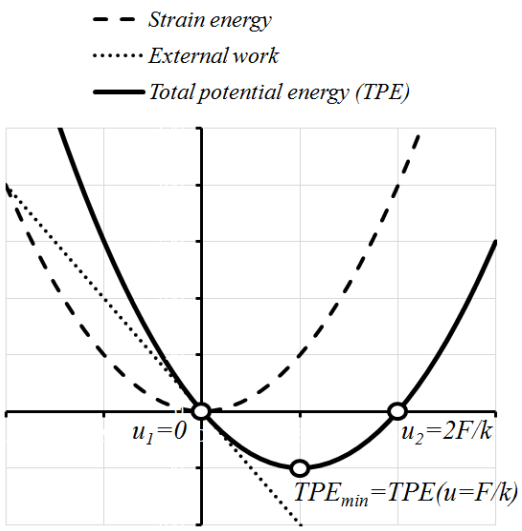


Figure 3.5. Interpretation of the total potential energy

The potential energy for one spring element in general (see Figure 3.6.) is

$$\Pi(u_1, u_2) = \frac{1}{2}k(u_2 - u_1)^2 - F_1 u_1 - F_2 u_2$$

The minimum value of the $\Pi(u_1, u_2)$ function

$$\frac{d\Pi}{du_1} = 0 = k(u_2 - u_1)(-1) - F_1 = ku_1 - ku_2 - F_1 = 0,$$
$$\frac{d\Pi}{du_2} = 0 = k(u_2 - u_1) - F_2 = -ku_1 + ku_2 - F_2 = 0.$$

In matrix form

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix},$$

$$\mathbf{Ku} = \mathbf{F},$$

where **K** is the stiffness matrix of the system, **u** is the unknown nodal displacements, while **F** is the load vector. It can be seen that the above equation is similar to the Eq. 1.2. This is the basic finite element equation system for one spring element.

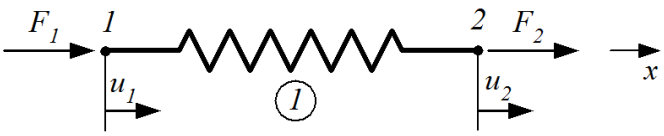


Figure 3.6. Spring element

Example 3.

Consider a spring structure shown in Figure 3.7. The applied load F and spring stiffness k_1 and k_2. The kinematic boundary condition is that $u_1 = 0$.

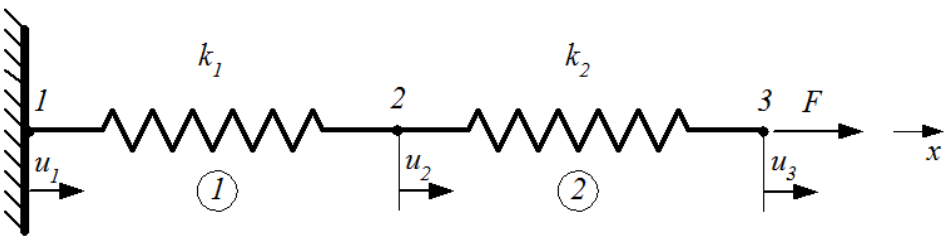


Figure 3.7. The spring structure

We are looking for the solution considering the minimum potential energy. The total potential energy of the spring structure can be expressed by

$$\Pi(u_2, u_3) = \underbrace{\Pi_1(u_2)}_{spring\ k_1} + \underbrace{\Pi_2(u_2, u_3)}_{spring\ k_2} = \frac{1}{2}k_1 u_2^2 + \frac{1}{2}k_2(u_3 - u_2)^2 - Fu_3.$$

The minimum of the $\Pi(u_2, u_3)$ function

$$\frac{d\Pi}{du_2} = 0 = k_1u_2 + k_2(u_3 - u_2)(-1) = (k_1 + k_2)u_2 - k_2u_3 = 0,$$
$$\frac{d\Pi}{du_3} = 0 = k_2(u_3 - u_2) - F = -k_2u_2 + k_2u_3 - F = 0.$$

Now we have two equations for u_2 and u_3, which can be written in matrix form

$$\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} - \begin{bmatrix} 0 \\ F \end{bmatrix} = 0,$$

$$\mathbf{Ku} = \mathbf{F}.$$

Example 4.
Consider a spring structure shown in Figure 3.8.

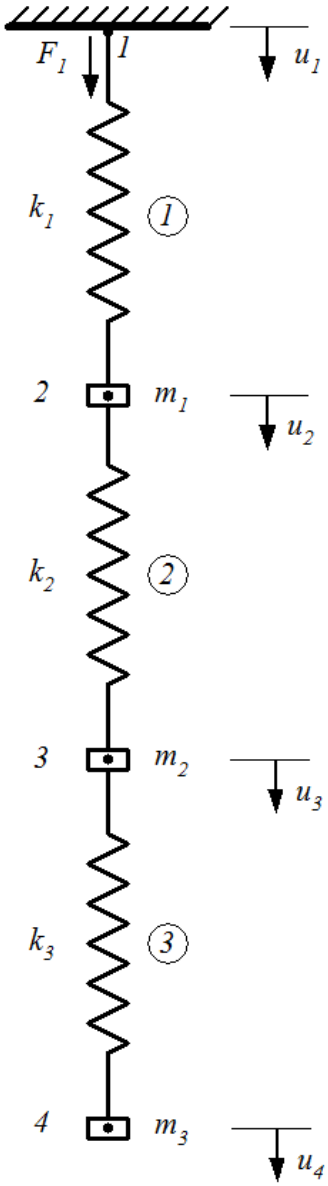


Figure 3.8. The spring structure

Data:

$k_1 = 2k, k_2 = 3k, k_3 = k = 50\frac{N}{m}$

$m = m_1 = m_2 = m_3 = 0{,}3kg$

$g \cong 10\frac{m}{s^2}$

$u_i = ?\ (i = 1, \dots, 4)$

$F_{1x} = ?$

The stiffness matrices of the springs (elements):

Element No.	Stiffness matrix $\mathbf{K}^i$	Nodal displacement vector $\mathbf{u}^i$	Load vector $\mathbf{F}^i$
1	$\begin{bmatrix} 2k & -2k \\ -2k & 2k \end{bmatrix}$	$\begin{bmatrix} u_1 \\ u_2 \end{bmatrix}$	$\begin{bmatrix} F_{1x} \\ mg \end{bmatrix}$
2	$\begin{bmatrix} 3k & -3k \\ -3k & 3k \end{bmatrix}$	$\begin{bmatrix} u_2 \\ u_3 \end{bmatrix}$	$\begin{bmatrix} 0 \\ mg \end{bmatrix}$
3	$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix}$	$\begin{bmatrix} u_3 \\ u_4 \end{bmatrix}$	$\begin{bmatrix} 0 \\ mg \end{bmatrix}$

The global equation system is

$$\mathbf{Ku} = \mathbf{F},$$

where

$$\mathbf{K} = \begin{bmatrix} 2k & -2k & 0 & 0 \\ -2k & 2k+3k & -3k & 0 \\ 0 & -3k & 3k+k & -k \\ 0 & 0 & -k & k \end{bmatrix} = \begin{bmatrix} 2k & -2k & 0 & 0 \\ -2k & 5k & -3k & 0 \\ 0 & -3k & 4k & -k \\ 0 & 0 & -k & k \end{bmatrix},$$

and

$$\mathbf{u} = \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix}, \qquad \mathbf{F} = \begin{bmatrix} F_{1x} \\ mg \\ mg \\ mg \end{bmatrix}.$$

$$\begin{bmatrix} 2k & -2k & 0 & 0 \\ -2k & 5k & -3k & 0 \\ 0 & -3k & 4k & -k \\ 0 & 0 & -k & k \end{bmatrix} \begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} F_{1x} \\ mg \\ mg \\ mg \end{bmatrix}$$

The kinematic boundary condition is that $u_1 = 0$.

$$\begin{bmatrix} 2k & -2k & 0 & 0 \\ -2k & 5k & -3k & 0 \\ 0 & -3k & 4k & -k \\ 0 & 0 & -k & k \end{bmatrix} \begin{bmatrix} 0 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} F_{1x} \\ mg \\ mg \\ mg \end{bmatrix}$$

We are looking for u_2, u_3, u_4 and F_{1x}.

$$\left.\begin{array}{c} -2ku_2 = F_{1x} \\ 5ku_2 - 3ku_3 = mg \\ -3ku_2 + 4ku_3 - ku_4 = mg \\ -ku_3 + ku_4 = mg \end{array}\right\} \begin{array}{c} F_{1x} = 9N\ (\uparrow) \\ u_2 = 0{,}09m \\ u_3 = 0{,}13m \\ u_4 = 0{,}19m \end{array}$$

3.6 Ritz method

In the case of relatively simple problems the equations of elasticity can be solved in closed form using integration, etc. For problems of complicated shape or loading, the solution to the governing differential equation cannot be found by analytical methods. For more complex problems, approximate procedures of solution must be used. The Ritz method is a classical technique where trial functions are used to obtain an approximate solution. This technique is very closely related to the finite element method and the finite element method can be regarded as an extension of this classical procedure.
Using the principle of minimum of potential energy approximate solution can be derived with the so called Ritz method. The Ritz method is based on the idea of constructing a series of trial approximating functions that satisfy the boundary conditions but not the differential equations. For the displacement based formulation this concept expresses the displacements in the form

$$\boldsymbol{u}^* = \boldsymbol{u}^*(C_0, C_1, \dots, C_n), \tag{3.16}$$

where $\boldsymbol{u}^*$ is the kinematically admissible displacement field, $C_0, C_1, \dots, C_n$ are unknown coefficients. Normally, these trial approximation functions are elementary functions such as polynomial, trigonometric, etc. The unknown coefficients have to be determined as the minimization of the potential energy. Using this approximation the total potential energy will be a function of these unknown coefficients

$$\Pi(\boldsymbol{u}^*) = \Pi^*(C_0, C_1, \dots, C_n), \tag{3.17}$$

and the minimizing condition can be expressed as a system of linear algebraic equation

$$\min \Pi^*(C_0, C_1, \dots, C_n) \ \rightarrow \ \frac{\partial \Pi^*}{\partial C_0} = 0, \frac{\partial \Pi^*}{\partial C_1} = 0, \dots \frac{\partial \Pi^*}{\partial C_n} = 0. \tag{3.18}$$

This system of algebraic equations have to be solved to obtain the coefficients $C_0, C_1, \dots, C_n$.

<u>Example 5.</u>
The problem introduced earlier (see Figure 2.9.) is analyzed by Ritz method.
The total potential energy in that case can be expressed with

$$\Pi^*(u^*) = \frac{1}{2}\int_0^l AE\left(\frac{du^*}{dx}\right)^2 dx - \int_0^l u^* q_x dx - F_x u(l)^*.$$

5a, Linear approximation
The trial function now is linear in form

$$u^*(x) = C_0 + C_1 x.$$

Since the kinematical boundary condition $u(0) = 0$,

$$u^*(0) = 0 = C_0 + C_1 0 \rightarrow C_0 = 0.$$

So

$$u^*(x) = C_1 x.$$

The normal strain

$$\varepsilon_x^* = \frac{du^*}{dx} = C_1.$$

Now we have the form for the total potential energy

$$\Pi^*(C_1) = \frac{1}{2}\int_0^l AEC_1^2 dx - \int_0^l C_1 x q_x dx - F_x C_1 l.$$

The minimum value of the $\Pi^*(C_1)$ function is

$$\frac{\partial \Pi^*}{\partial C_1} = 0 = \int_0^l AEC_1 dx - \int_0^l x q_x dx - F_x l = AEC_1[x]_0^l - q_x\left[\frac{x^2}{2}\right]_0^l - F_x l,$$

$$0 = AEC_1 l - q_x\frac{l^2}{2} - F_x l \quad \rightarrow \quad C_1 = \frac{F_x l + q_x\frac{l^2}{2}}{AEl} = \frac{F_x + q_x\frac{l}{2}}{AE}.$$

Substituting into the displacement field we get

$$u(x) = C_1 x = \frac{F_x + q_x\frac{l}{2}}{AE}x,$$
$$\varepsilon_x(x) = C_1 = \frac{F_x + q_x\frac{l}{2}}{AE},$$
$$N(x) = \sigma_x A = E\varepsilon_x A = F_x + q_x\frac{l}{2}.$$

Comparing the results came from the linear approximation and the analytical solution significant deviation can be observed.

5b, Quadratic approximation

The trial function now is quadratic in form

$$u^*(x) = C_0 + C_1 x + C_2 x^2.$$

Since the kinematical boundary condition $u(0) = 0$,

$$u^*(0) = 0 = C_0 + C_1 0 + C_2 0^2 \rightarrow C_0 = 0.$$

So

$$u^*(x) = C_1 x + C_2 x^2.$$

The normal strain

$$\varepsilon_x^* = \frac{du^*}{dx} = C_1 + 2C_2 x.$$

Now we have the form for the total potential energy

$$\Pi^*(C_1, C_2) = \frac{1}{2}\int_0^l AE(C_1 + 2C_2 x)^2 dx - \int_0^l (C_1 x + C_2 x^2) q_x dx - F_x(C_1 l + C_2 l^2).$$

The minimum of the $\Pi^*(C_1, C_2)$ function

$$\frac{\partial \Pi^*}{\partial C_1} = 0 = \int_0^l AE(C_1 + 2C_2 x) dx - \int_0^l x q_x dx - F_x l =$$
$$= \int_0^l AE C_1 dx + \int_0^l 2AE C_2 x\, dx - \int_0^l x q_x dx - F_x l,$$

$$0 = AE C_1 [x]_0^l + 2AE C_2 \left[\frac{x^2}{2}\right]_0^l - q_x \left[\frac{x^2}{2}\right]_0^l - F_x l,$$

$$0 = AE C_1 l + AE C_2 l^2 - q_x \frac{l^2}{2} - F_x l.$$

$$\frac{\partial \Pi^*}{\partial C_2} = 0 = \int_0^l 2AE(C_1 + 2C_2 x) x dx - \int_0^l x^2 q_x dx - F_x l^2 =$$
$$= \int_0^l 2AE C_1 x dx + \int_0^l 4AE C_2 x^2\, dx - \int_0^l x^2 q_x dx - F_x l^2,$$

$$0 = 2AE C_1 \left[\frac{x^2}{2}\right]_0^l + 4AE C_2 \left[\frac{x^3}{3}\right]_0^l - q_x \left[\frac{x^3}{3}\right]_0^l - F_x l^2,$$

$$0 = AE C_1 l^2 + 4AE C_2 \frac{l^3}{3} - q_x \frac{l^3}{3} - F_x l^2.$$

We are looking for C_1, C_2

$$\left.\begin{aligned} 0 &= AEC_1 l + AEC_2 l^2 - q_x \frac{l^2}{2} - F_x l \\ 0 &= AEC_1 l^2 + 4AEC_2 \frac{l^3}{3} - q_x \frac{l^3}{3} - F_x l^2 \end{aligned}\right\} \quad \begin{aligned} C_1 &= \frac{F_x + q_x l}{AE} \\ C_2 &= -\frac{q_x}{2AE} \end{aligned}$$

Substituting into the displacement field we get

$$\begin{aligned} u(x) &= C_1 x + C_2 x^2 = \frac{F_x + q_x l}{AE} x - \frac{q_x}{2AE} x^2, \\ \varepsilon_x(x) &= C_1 + 2C_2 x = \frac{F_x + q_x l}{AE} - \frac{q_x}{AE} x, \\ N(x) &= \sigma_x A = E\varepsilon_x A = F_x + q_x(l - x). \end{aligned}$$

The results of the usage of quadratic approximation are equal to the analytical solution.
It can be seen that two different approximation result in different solutions. Increase the degree of the approximation better solution to get. The result of the two different approximations is illustrated in Figure 3.9.

x [mm]	$u_{\text{analytical}}$ [mm]	$u_{\text{ritz\_lin}}$ [mm]	$u_{\text{ritz\_quad}}$ [mm]
0	0	0	0
100	4,67191E-05	2,9445E-05	4,67191E-05
200	8,95996E-05	5,889E-05	8,95996E-05
300	0,000128641	8,8335E-05	0,000128641
400	0,000163844	0,00011778	0,000163844
500	0,000195209	0,000147225	0,000195209
600	0,000222734	0,00017667	0,000222734
700	0,000246421	0,000206115	0,000246421
800	0,00026627	0,00023556	0,00026627
900	0,000282279	0,000265005	0,000282279
1000	0,00029445	0,00029445	0,00029445

Table 3.1. The displacement fields for the different methods

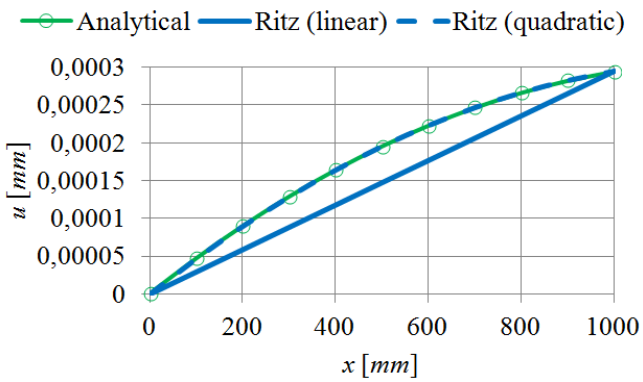


Figure 3.9. Solutions (Analytical, Linear approximation, Quadratic approximation)

4. FORMULATION OF THE FINITE ELEMENT METHOD

The finite element method has developed into a key and indispensable computational technology in modeling and simulating engineering problems. In order to analyze engineering problems mathematical models have to be developed to describe the problem which implies that some assumptions have to be made for simplification. The mathematical expression consists of differential equations and given conditions. The analytic procedure to get solution of these differential equations is very difficult in general. However, numerical solution techniques can find their approximate solutions. The finite element method is one of the most important numerical solution techniques. Nowadays the general purpose finite element softwares are able to analyze almost every kind of natural phenomenon.
The finite element procedure based on the following:

- The investigated body is divided into elements (procedure is called meshing). The unknown fields are approximated on these sub-regions. The valid fields for the body can be constructed by the connection of the applied fields of the sub-regions.
- The solution of the elasticity problem is produced with energy principles (Lagrange variational principle, Castigliano variational principle, etc.).

The usage of the Lagrange variational principle ($\delta\Pi = 0$) is widespread in finite element softwares, where the primer unknown is the displacement field.
The finite element formulation has to be based on a coordinate system. In formulating finite element equations for elements, it is often convenient to use a local coordinate system which is defined for an element in reference to the global coordinate system which is usually defined for the entire structure. Based on the local coordinate system defined on an element, the displacement within the element is assumed by polynomial interpolation using the nodal displacements. This approach of assuming the displacements is often called the displacement based method. In detail the displacement based method of analysis is introduced.
The displacement based finite element method relies on the following steps:

- The body is divided into arbitrary shaped, finite number of sub-regions, called finite elements.
- The displacement fields are approximated on elements separately. The approximation functions are polynomials in general. This is called local approximation.
- The elements are determined by nodes, the applied approximate functions on elements are joined with the help of the nodal displacement parameters.
- With the usage of the Lagrange variational principle a linear algebraic equation system for the nodal parameters can be obtained.
- The nodal displacement parameters are determined from the linear algebraic equation system.
- Using the displacement field the quantities of interest can be calculated, like deformation and stress.

4.1. General derivation of the displacement based finite element equilibrium equations

In order to construct the finite element equations matrix notation is applied. Consider the equilibrium of a general three-dimensional body. The shape, the geometry of the body and the material are given. Considering the body surface area, the body is supported on the area A_u with prescribed displacements (support conditions on A_u) and subjected to surface loads on the area A_p.

The aim is to determine the displacements of the body and the corresponding strains are stresses summarized as follows in matrix notation.
The matrix of the displacement vector

$$\mathbf{u} = \underbrace{\begin{bmatrix} u(x,y,z) \\ v(x,y,z) \\ w(x,y,z) \end{bmatrix}}_{(3x1)}. \tag{4.1}$$

The matrix of the strain vector

$$\boldsymbol{\varepsilon} = \underbrace{\begin{bmatrix} \varepsilon_x(x,y,z) \\ \varepsilon_y(x,y,z) \\ \varepsilon_z(x,y,z) \\ \gamma_{xy}(x,y,z) \\ \gamma_{yz}(x,y,z) \\ \gamma_{zx}(x,y,z) \end{bmatrix}}_{(6x1)}. \tag{4.2}$$

The matrix of the stress vector

$$\boldsymbol{\sigma} = \underbrace{\begin{bmatrix} \sigma_x(x,y,z) \\ \sigma_y(x,y,z) \\ \sigma_z(x,y,z) \\ \tau_{xy}(x,y,z) \\ \tau_{yz}(x,y,z) \\ \tau_{zx}(x,y,z) \end{bmatrix}}_{(6x1)}. \tag{4.3}$$

During the solution we assume linear analysis conditions, so the displacements are infinitesimally small.
The solid body is divided into finite number of arbitrary shaped and sized elements. The elements are determined by nodes. The e denotes the element (sub-region), i, j, k, l denote the nodes of the elements (see Figure 4.1.).

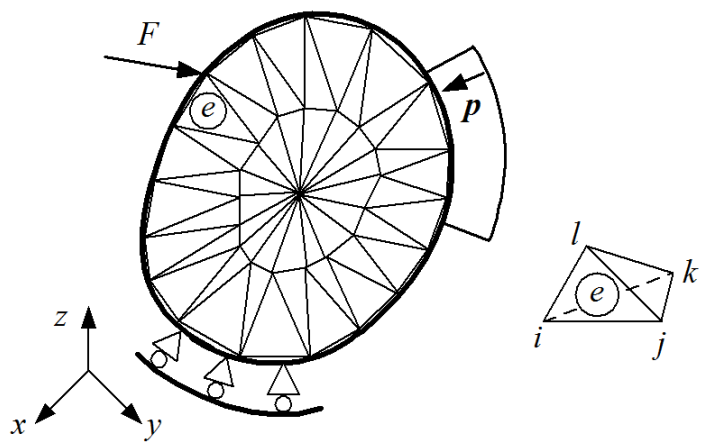


Figure 4.1. Finite element discretization and a finite element

A typical finite element is defined by nodes and straight line boundaries. The displacements are approximated element by element,

$$\underbrace{\mathbf{u}^e}_{(3x1)} = \underbrace{\mathbf{N}^e}_{(3xn)} \underbrace{\mathbf{q}^e}_{(nx1)} , \tag{4.4}$$

where $\mathbf{u}^e$ is the displacement field of the element, $\mathbf{N}^e$ is the approximation (shape function) matrix of the element, $\mathbf{q}^e$ is the nodal displacement vector of the element, n is the degrees of freedom of the element. The shape function matrix contains the approximation functions.
The nodal displacement vector of element e is

$$\mathbf{q}^e = \underbrace{\begin{bmatrix} \mathbf{q}_i \\ \mathbf{q}_j \\ \mathbf{q}_k \\ \vdots \\ \mathbf{q}_N \end{bmatrix}^e}_{(3Nx1)} , \tag{4.5}$$

where N is the number of nodes. The nodal displacement vector of element i is

$$\mathbf{q}_i^e = \underbrace{\begin{bmatrix} u_i \\ v_i \\ w_i \end{bmatrix}^e}_{(3x1)} . \tag{4.6}$$

Note that the displacement components can also consist of rotations for structures of beams and plates. The shape function matrix is predefined to assume shapes of the displacement variations with respect to the coordinates. The general form of the shape function matrix is

$$\mathbf{N}^e = \underbrace{\begin{bmatrix} \mathbf{N}_i \\ \mathbf{N}_j \\ \mathbf{N}_k \\ \vdots \\ \mathbf{N}_N \end{bmatrix}^e}_{(3Nx3)} . \tag{4.7}$$

The shape function block belonging to the ith nodes of the element e is

$$\mathbf{N}_i^e = \underbrace{\begin{bmatrix} N_{xxi} & N_{xyi} & N_{xzi} \\ N_{yxi} & N_{yyi} & N_{yzi} \\ N_{zxi} & N_{zyi} & N_{zzi} \end{bmatrix}^e}_{(3x3)} . \tag{4.8}$$

With the assumption on the displacements the strain state of an element can be approximated,

$$\boldsymbol{\varepsilon}^e = \underbrace{\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix}^e}_{(6x1)} = \underbrace{\begin{bmatrix} \frac{\partial u}{\partial x} \\ \frac{\partial v}{\partial y} \\ \frac{\partial w}{\partial z} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \\ \frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \end{bmatrix}^e}_{(6x1)} = \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix}^e}_{(6x3)} \underbrace{\begin{bmatrix} u \\ v \\ w \end{bmatrix}^e}_{(3x1)}. \tag{4.9}$$

Here

$$\mathbf{D}^e = \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix}^e}_{(6x3)} \tag{4.10}$$

is the differential matrix, thus the strains can be expressed by using Eq. 4.4

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e \mathbf{N}^e \mathbf{q}^e = \mathbf{B}^e \mathbf{q}^e, \tag{4.11}$$

where $\mathbf{B}^e$ is the strain displacement matrix

$$\mathbf{B}_i^e = \underbrace{\begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_i}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_i}{\partial z} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial z} & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial z} & 0 & \frac{\partial N_i}{\partial x} \end{bmatrix}^e}_{(6x3)}. \tag{4.12}$$

With the assumption on the strains the stress state of an element can be approximated using the Hooke's law Eq. 2.48 considering isotropic material

$$\boldsymbol{\sigma}^e = \underbrace{\begin{bmatrix} \sigma_x \\ \sigma_y \\ \sigma_z \\ \tau_{xy} \\ \tau_{yz} \\ \tau_{zx} \end{bmatrix}^e}_{(6x1)} = \underbrace{\begin{bmatrix} c_1 & c_2 & c_2 & 0 & 0 & 0 \\ c_2 & c_1 & c_2 & 0 & 0 & 0 \\ c_2 & c_2 & c_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & c_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & c_3 & 0 \\ 0 & 0 & 0 & 0 & 0 & c_3 \end{bmatrix}^e}_{(6x6)} \underbrace{\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \varepsilon_z \\ \gamma_{xy} \\ \gamma_{yz} \\ \gamma_{zx} \end{bmatrix}^e}_{(6x1)}, \tag{4.13}$$

where

$$\begin{aligned} c_1 &= \frac{E(1-\nu)}{(1-2\nu)(1+\nu)} \\ c_2 &= \frac{E\nu}{(1-2\nu)(1+\nu)} \\ c_3 &= \frac{E}{2(1+\nu)} = G \end{aligned} \tag{4.14}$$

The elasticity matrix is introduced

$$\mathbf{C}^e = \underbrace{\begin{bmatrix} c_1 & c_2 & c_2 & 0 & 0 & 0 \\ c_2 & c_1 & c_2 & 0 & 0 & 0 \\ c_2 & c_2 & c_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & c_3 & 0 & 0 \\ 0 & 0 & 0 & 0 & c_3 & 0 \\ 0 & 0 & 0 & 0 & 0 & c_3 \end{bmatrix}^e}_{(6x6)}, \tag{4.15}$$

from which the stress using Eq. 4.11

$$\boldsymbol{\sigma}^e = \mathbf{C}^e \boldsymbol{\varepsilon}^e = \mathbf{C}^e \mathbf{B}^e \mathbf{q}^e. \tag{4.16}$$

According to Hooke's law the stresses in a finite element are related to the element strains, where $\mathbf{C}^e$ is the elasticity matrix of element.
The total potential energy of the element according to Eq. 3.8 is

$$\Pi^e = U - W = \frac{1}{2}\int_V \boldsymbol{\varepsilon}^{eT} \boldsymbol{\sigma}^e \, dV - \int_V \mathbf{u}^{eT} \mathbf{q}^e \, dV - \int_{A_p} \mathbf{u}^{eT} \mathbf{p}^e \, dA. \tag{4.17}$$

Substituting equations Eq. 4.4, Eq. 4.12 and Eq. 4.16 into the Eq. 4.17 and taking the constants from the integrals the total potential energy of the element is

$$\Pi^e = \frac{1}{2}\mathbf{q}^{eT} \int_V \mathbf{B}^{eT} \mathbf{C}^e \mathbf{B}^e \, dV \mathbf{q}^e - \mathbf{q}^{eT} \int_V \mathbf{N}^{eT} \mathbf{q}^e \, dV - \mathbf{q}^{eT} \int_{A_p} \mathbf{N}^{eT} \mathbf{p}^e \, dA, \tag{4.18}$$

where

$$\mathbf{K}^e = \int_V \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, dV \tag{4.19}$$

is the stiffness matrix of the element. The stiffness matrix of the element is symmetric, thus $\mathbf{K}^e = \mathbf{K}^{eT}$. The stiffness matrix defines the geometric and material properties of the element.
The nodal load vector with respect to the volume forces is

$$\mathbf{f}_q^e = \int_V \mathbf{N}^{eT}\mathbf{q}^e \, dV. \tag{4.20}$$

The nodal load vector with respect to the surface forces is

$$\mathbf{f}_p^e = \int_{A_p} \mathbf{N}^{eT}\mathbf{p}^e \, dA. \tag{4.21}$$

The nodal load vector of the element

$$\mathbf{f}^e = \mathbf{f}_q^e + \mathbf{f}_p^e. \tag{4.22}$$

Using the notations the total potential energy of the element

$$\Pi^e = \frac{1}{2}\mathbf{q}^{eT}\mathbf{K}^e\mathbf{q}^e - \mathbf{q}^{eT}\mathbf{f}^e. \tag{4.23}$$

The principle of minimum total potential can be applied only on the solid body. It is not valid on individual elements. The total potential energy of the solid body is equal with sum of the potential energy of the elements,

$$\Pi = \sum_{e=1}^{Q} \Pi^e = \frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \mathbf{q}^T\mathbf{f}, \tag{4.24}$$

where Q is the number of finite elements applied for the discretization, $\mathbf{K}$ is the stiffness matrix of the body, $\mathbf{q}$ is the nodal displacement vector of the body and $\mathbf{f}$ is the nodal force vector of the body.
According to the Lagrange variational principle $\delta\Pi = 0$, so the Eq. 4.24

$$\delta\Pi = \delta\left(\frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \mathbf{q}^T\mathbf{f}\right) = \delta\mathbf{q}^T(\mathbf{K}\mathbf{q} - \mathbf{f}) = 0. \tag{4.25}$$

The Eq. 4.25 results

$$\mathbf{K}\mathbf{q} = \mathbf{f}. \tag{4.26}$$

In linear analysis the Eq. 4.26 finite element system equilibrium equations are to be solved.
Note that actual nodes belong to more elements which reduce the global equation system (see Figure 4.2.).

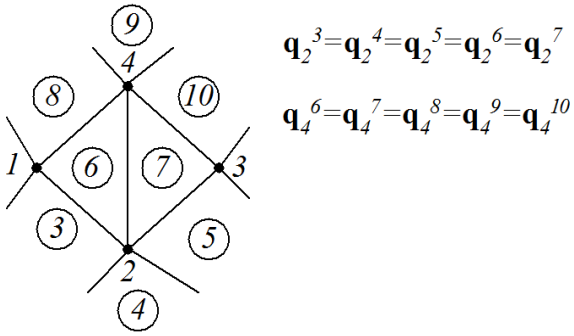


Figure 4.2. Nodes belong to neighbor elements

The kinematical boundary condition has to be taken into consideration, thus actual nodal displacement vector can have zero components. This fact reduces the global equation system either.

4.2. Truss element

The truss is one of the simplest structural elements. It is a straight bar which is designed to take only axial forces, therefore it deforms only in axial direction. The cross sections of the bar can be arbitrary, but the dimensions of the cross section have to be much smaller than the axial direction. This element called truss element or bar element.
The examined domain is divided into small, interconnected sub-region. The displacements are approximated separately on these sub-regions locally. These sub-regions are called finite elements. The sub-regions (elements) are connected in nodes.
Let us consider the problem (see Figure 2.9.) analyzed earlier. Here, the domain is divided into three equal part which means that $l = 3L$, see Figure 4.3. We also have to take into consideration that some elements are connected in the same nodes, for example $u_2^1 = u_2^2$, where the subscript denotes the number of the nodes.

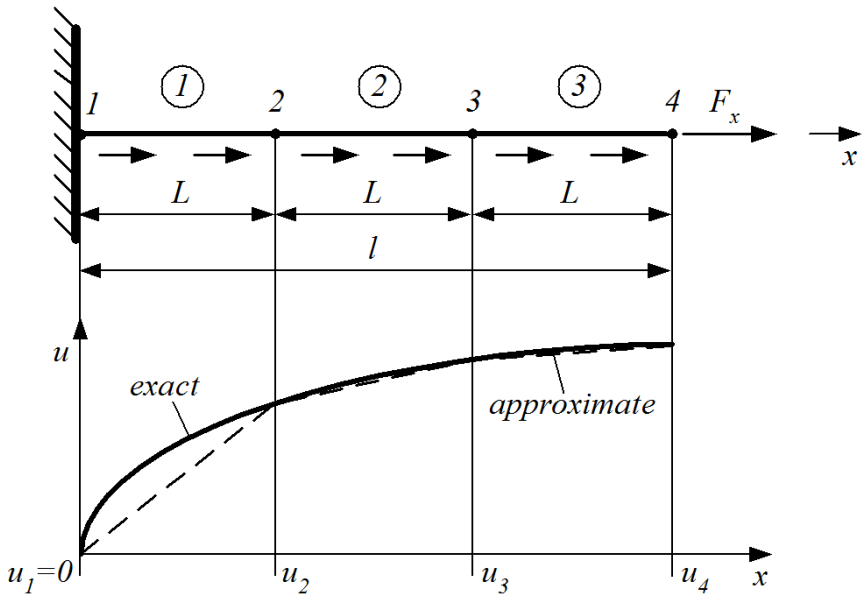


Figure 4.3. The finite element model and the local approximation

The trial approximation function can be constructed as the linear combination of shape functions N_i $(i = 1,2,3,4)$ belonging to the nodes, thus

$$u^*(x) = u(x) = \sum_{i=1}^{4} N_i(x)\, u_i. \tag{4.27}$$

The kinematic boundary condition is $u_1 = 0$.
The total potential energy of the structure is the sum of the potential energy calculated on each element and the work of the external load

$$\Pi = \sum_{e=1}^{3} \Pi^e - F_x u_4. \tag{4.28}$$

The approximation of the displacement of an element can be seen n Figure 4.4.

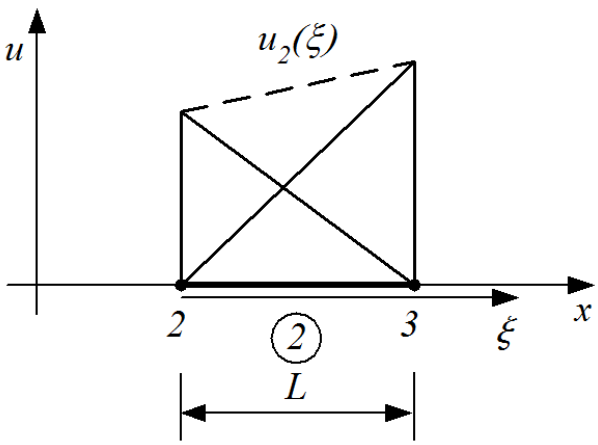


Figure 4.4. Approximation of the displacement on element No.2

The potential energy of one element using local coordinates can be expressed

$$\Pi^e(u_i, u_j) = \frac{1}{2}\int_0^L AE\left(\frac{du^e}{d\xi}\right)^2 d\xi - \int_0^L u^e q_x\, d\xi = \underbrace{\frac{1}{2}\int_0^L \left(\frac{du^e}{d\xi}\right) AE\left(\frac{du^e}{d\xi}\right) d\xi}_{U^e\ strain\ energy} - \underbrace{\int_0^L u^e q_x\, d\xi}_{\substack{W\ work\ of \\ distributed \\ force\ system}}. \tag{4.29}$$

Now investigate the approximation on the element No.2. The displacement field along the element can be expressed by the coordinate ξ and the u_i nodal displacement coordinates, thus

$$u^2(\xi) = u_2 + \frac{u_3 - u_2}{L}\xi. \tag{4.30}$$

It clearly states that the displacement within the element varies linearly. The element is therefore called a linear element. Reorder Eq. 4.30 according to the nodal displacement

$$u^2(\xi) = \left(1 - \frac{\xi}{L}\right) u_2 + \frac{\xi}{L} u_3 = \begin{bmatrix} \left(1 - \frac{\xi}{L}\right) & \frac{\xi}{L} \end{bmatrix} \begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} u_2 & u_3 \end{bmatrix} \begin{bmatrix} \left(1 - \frac{\xi}{L}\right) \\ \frac{\xi}{L} \end{bmatrix}. \tag{4.31}$$

The approximation of an element can be derived in general case with the same notation,

$$u^e(\xi) = \left(1 - \frac{\xi}{L}\right) u_i + \frac{\xi}{L} u_j = \begin{bmatrix} \left(1 - \frac{\xi}{L}\right) & \frac{\xi}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \begin{bmatrix} u_i & u_j \end{bmatrix} \begin{bmatrix} \left(1 - \frac{\xi}{L}\right) \\ \frac{\xi}{L} \end{bmatrix}. \tag{4.32}$$

If the displacement is known, strain can also be calculated by

$$\varepsilon^e(\xi) = \frac{du^e}{d\xi} = \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \begin{bmatrix} u_i & u_j \end{bmatrix} \begin{bmatrix} -\frac{1}{L} \\ \frac{1}{L} \end{bmatrix} \tag{4.33}$$

which is a direct result from differentiating Eq. 4.32 with respect to ξ. Note that the strain in Eq. 4.33 is a constant value in the element. Using the Hooke's law the normal force can be derived

$$N^e(\xi) = A\sigma_x^e = AE\varepsilon^e(\xi) = AE \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \begin{bmatrix} u_i & u_j \end{bmatrix} \begin{bmatrix} -\frac{1}{L} \\ \frac{1}{L} \end{bmatrix} AE. \tag{4.34}$$

The strain energy of the element

$$\begin{aligned} U^e &= \frac{1}{2} \int_0^L \left(\frac{du^e}{d\xi}\right) AE \left(\frac{du^e}{d\xi}\right) d\xi = \frac{1}{2} \int_0^L \begin{bmatrix} u_i & u_j \end{bmatrix} \begin{bmatrix} -\frac{1}{L} \\ \frac{1}{L} \end{bmatrix} AE \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix} d\xi = . \\ &= \frac{1}{2} \begin{bmatrix} u_i & u_j \end{bmatrix} \int_0^L \begin{bmatrix} -\frac{1}{L} \\ \frac{1}{L} \end{bmatrix} AE \begin{bmatrix} -\frac{1}{L} & \frac{1}{L} \end{bmatrix} d\xi \begin{bmatrix} u_i \\ u_j \end{bmatrix} = \frac{1}{2} \begin{bmatrix} u_i & u_j \end{bmatrix} \int_0^L \begin{bmatrix} \frac{AE}{L^2} & -\frac{AE}{L^2} \\ -\frac{AE}{L^2} & \frac{AE}{L^2} \end{bmatrix} d\xi \begin{bmatrix} u_i \\ u_j \end{bmatrix}. \end{aligned} \tag{4.35}$$

Note, that

$$\int_0^L \frac{AE}{L^2} d\xi = \left[\frac{AE}{L^2} \xi\right]_0^L = \frac{AE}{L}. \tag{4.36}$$

Using Eq. 4.36 and substituting into Eq. 4.35 we get

$$U^e = \frac{1}{2}[u_i \quad u_j]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_i \\ u_j \end{bmatrix} = \frac{1}{2}\mathbf{q}^{eT}\mathbf{K}^e\mathbf{q}^e, \tag{4.37}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element and $\mathbf{K}^e$ is the stiffness matrix of the element.
The work of the distributed force system along the element length (last term of Eq. 4.29)

$$\int_0^L u^e q_x \, d\xi = [u_i \quad u_j]\int_0^L \begin{bmatrix} 1 - \frac{\xi}{L} \\ \frac{\xi}{L} \end{bmatrix} q_x \, d\xi. \tag{4.38}$$

Note, that

$$\begin{gathered} \int_0^L \left(1 - \frac{\xi}{L}\right) q_x \, d\xi = \left[\left(\xi - \frac{\xi^2}{2L}\right) q_x\right]_0^L = \left(L - \frac{L^2}{2L}\right) q_x = \frac{q_x L}{2} \\ \int_0^L \frac{\xi}{L} q_x \, d\xi = \left[\frac{\xi^2}{2L} q_x\right]_0^L = \frac{q_x L}{2}. \end{gathered} \tag{4.39}$$

Using Eq. 4.39 and substituting into Eq. 4.38 we get

$$\int_0^L u^e q_x \, d\xi = [u_i \quad u_j]\begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_x L}{2} \end{bmatrix} = \mathbf{q}^{eT}\mathbf{f}_q^e, \tag{4.40}$$

where $\mathbf{f}_q^e$ is the load vector of the element. Using Eq. 4.37 and Eq. 4.40 the potential energy of the element is

$$\Pi^e = \frac{1}{2}\mathbf{q}^{eT}\mathbf{K}^e\mathbf{q}^e - \mathbf{q}^{eT}\mathbf{f}_q^e, \tag{4.41}$$

or

$$\Pi^e = \frac{1}{2}[u_i \quad u_j]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_i \\ u_j \end{bmatrix} - [u_i \quad u_j]\begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_x L}{2} \end{bmatrix}. \tag{4.42}$$

The total potential energy of the structure

$$\Pi = \sum_{e=1}^{3} \Pi^e - F_x u_4 = \Pi^1 + \Pi^2 + \Pi^3 - F_x u_4. \tag{4.43}$$

The potential energy of the elements:

Element No.	Total potential energy Π^e
1	$\frac{1}{2}[u_1 \quad u_2]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} - [u_1 \quad u_2]\begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_x L}{2} \end{bmatrix}$
2	$\frac{1}{2}[u_2 \quad u_3]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_2 \\ u_3 \end{bmatrix} - [u_2 \quad u_3]\begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_x L}{2} \end{bmatrix}$
3	$\frac{1}{2}[u_3 \quad u_4]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} \\ -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_3 \\ u_4 \end{bmatrix} - [u_3 \quad u_4]\begin{bmatrix} \frac{q_x L}{2} \\ \frac{q_x L}{2} \end{bmatrix}$

While $u_2^1 = u_2^2$ and $u_3^2 = u_3^3$ the total potential energy of the structure

$$\Pi(u_1, u_2, u_3, u_4) = \frac{1}{2}[u_1 \quad u_2 \quad u_3 \quad u_4]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} & 0 & 0 \\ -\frac{AE}{L} & \frac{2AE}{L} & -\frac{AE}{L} & 0 \\ 0 & -\frac{AE}{L} & \frac{2AE}{L} & -\frac{AE}{L} \\ 0 & 0 & -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} - \\ -[u_1 \quad u_2 \quad u_3 \quad u_4]\begin{bmatrix} \frac{q_x L}{2} \\ q_x L \\ q_x L \\ \frac{q_x L}{2} + F_x \end{bmatrix}. \tag{4.44}$$

Taking the kinematic boundary condition into consideration $u_1 = 0$,

$$\Pi(u_1, u_2, u_3, u_4) = \frac{1}{2}[0 \quad u_2 \quad u_3 \quad u_4]\begin{bmatrix} \frac{AE}{L} & -\frac{AE}{L} & 0 & 0 \\ -\frac{AE}{L} & \frac{2AE}{L} & -\frac{AE}{L} & 0 \\ 0 & -\frac{AE}{L} & \frac{2AE}{L} & -\frac{AE}{L} \\ 0 & 0 & -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} 0 \\ u_2 \\ u_3 \\ u_4 \end{bmatrix} - \\ -[0 \quad u_2 \quad u_3 \quad u_4]\begin{bmatrix} \frac{q_x L}{2} \\ q_x L \\ q_x L \\ \frac{q_x L}{2} + F_x \end{bmatrix}. \tag{4.45}$$

While $u_1 = 0$ the first row and column of the structural stiffness matrix can be eliminated, thus

$$\Pi(u_2, u_3, u_4) = \frac{1}{2}[u_2 \quad u_3 \quad u_4]\begin{bmatrix} \frac{2AE}{L} & -\frac{AE}{L} & 0 \\ -\frac{AE}{L} & \frac{2AE}{L} & -\frac{AE}{L} \\ 0 & -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_2 \\ u_3 \\ u_4 \end{bmatrix} - \\ -[u_2 \quad u_3 \quad u_4]\begin{bmatrix} q_x L \\ q_x L \\ \frac{q_x L}{2} + F_x \end{bmatrix}. \tag{4.46}$$

The Eq. 4.46 can be written in concise form

$$\Pi(\mathbf{q}) = \frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \boldsymbol{q}\mathbf{f}_q, \tag{4.47}$$

where $\mathbf{K}$ is the structural stiffness matrix, $\mathbf{q}$ is the structural nodal vector and $\mathbf{f}_q$ is the structural load vector.
The total potential energy of the structure is the function of the displacement parameters. The minimum of the function according to the principle of minimum total potential energy is sought, thus

$$\min \Pi\,(u_2, u_3, u_4) \;\rightarrow\; \frac{\partial \Pi(u_2, u_3, u_4)}{\partial u_2} = 0;\; \frac{\partial \Pi(u_2, u_3, u_4)}{\partial u_3} = 0;\; \frac{\partial \Pi(u_2, u_3, u_4)}{\partial u_4} = 0, \tag{4.48}$$

or

$$\min \Pi\,(\mathbf{q}) \;\rightarrow\; \frac{\partial \Pi(\mathbf{q})}{\partial \mathbf{q}} = \mathbf{0}. \tag{4.49}$$

$$\frac{\partial \Pi(\mathbf{q})}{\partial \mathbf{q}} = \mathbf{0} = \frac{\partial\left(\frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \boldsymbol{q}\mathbf{f}_q\right)}{\partial \mathbf{q}} = \mathbf{K}\mathbf{q} - \mathbf{f}_q. \tag{4.50}$$

Reordering Eq. 4.50 we get a linear algebraic equation system

$$\mathbf{K}\mathbf{q} = \mathbf{f}_q. \tag{4.51}$$

In detail

$$\begin{bmatrix} \frac{2AE}{L} & -\frac{AE}{L} & 0 \\ -\frac{AE}{L} & \frac{2AE}{L} & -\frac{AE}{L} \\ 0 & -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix}\begin{bmatrix} u_2 \\ u_3 \\ u_4 \end{bmatrix} = \begin{bmatrix} q_x L \\ q_x L \\ \frac{q_x L}{2} + F_x \end{bmatrix}. \tag{4.52}$$

Eq. 4.25 contains three equations which can be solved by the nodal displacement parameters

$$\left.\begin{aligned} \frac{2AE}{L}u_2 - \frac{AE}{L}u_3 &= q_x L \\ -\frac{AE}{L}u_2 + \frac{2AE}{L}u_3 - \frac{AE}{L}u_4 &= q_x L \\ -\frac{AE}{L}u_3 + \frac{AE}{L}u_4 &= \frac{q_x L}{2} + F_x \end{aligned}\right\} \begin{aligned} u_2 &= \frac{5q_x L^2}{2AE} + \frac{F_x L}{AE} \\ u_3 &= \frac{4q_x L^2}{AE} + \frac{2F_x L}{AE}. \\ u_4 &= \frac{9q_x L^2}{2AE} + \frac{3F_x L}{AE} \end{aligned} \tag{4.53}$$

Using the displacement parameters the quantities of interest can be calculated, like normal strains and normal forces.

4.3. Beam element

A beam is a simple but commonly used structural element. It is also geometrically a straight bar of an arbitrary cross section but it deforms only in directions perpendicular to its axis. The element is known as beam element. In beam structures the beams are joined together by welding, thus both forces and moments can be transmitted between the beams. There are two theorems for beam elements, one is Euler-Bernoulli beam theorem, the another is Timoshenko beam theorem. The difference between the two theorems is that the strain energy coming from shear which is neglected by the usage of the Euler-Bernoulli beam theorem, which makes it simpler. Here, the Euler-Bernoulli beam theorem is detailed. Moreover planar beam structures are specified.

The following assumptions are considered during analyzing planar beam structures:

- The beam has straight centerline (neutral axis).
- The deflections of the beam are small in comparison to the specific dimensions of the beam.
- The material of the beam is linear elastic, isotropic and homogeneous.
- The axis y and axis z are the inertial axes.

The basic problem of beams is illustrated in Figure 4.5.

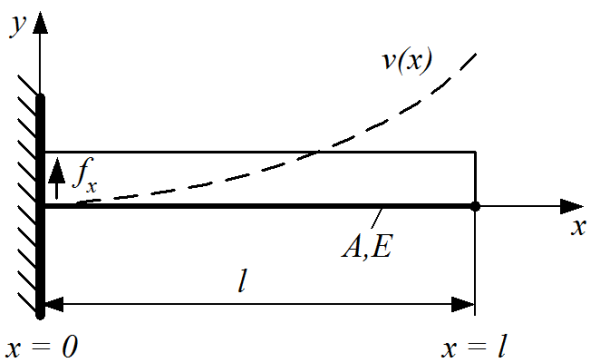


Figure 4.5. Beam structure

4.3.1. The equation system and the boundary conditions for beam problem

The stress arising in bending can be calculated

$$\sigma_x = \frac{M_b}{I_z} y, \tag{4.54}$$

where M_b the bending moment, I_z is the moment of inertia and y is the distance from the neutral axis. The displacement of the centerline can be expressed by

$$\boldsymbol{u} = v\boldsymbol{j} \rightarrow v(x), \tag{4.54}$$

considering small strains

$$\frac{dv}{dx} \ll 1. \tag{4.55}$$

According to the Euler-Bernoulli beam theorem $\gamma_{xy} = 0$ and $\varepsilon_y = \varepsilon_z = 0$.
The angular displacement of the cross section can be derived, thus

$$\varphi(x) = -\frac{dv(x)}{dx}. \tag{4.56}$$

The κ curvature of the neutral axis can also be expressed,

$$\kappa(x) = -\frac{d^2v(x)}{dx^2} = \frac{d\varphi(x)}{dx}. \tag{4.57}$$

The normal strain is

$$\varepsilon_x(x) = \kappa(x)y. \tag{4.58}$$

According to the Hooke's law

$$\sigma_x(x) = E\varepsilon_x(x). \tag{4.59}$$

The Eq. 4.57 can be written using Eq. 4.54 and Eq. 4.58, thus

$$\frac{M_b(x)}{I_z}y = E\kappa(x)y \rightarrow M_b(x) = I_zE\kappa(x) = -I_zE\frac{d^2v(x)}{dx^2}. \tag{4.60}$$

The equilibrium equation according to the Euler-Bernoulli theorem is

$$\frac{d^2M_b(x)}{dx^2} + f_y(x) = 0. \tag{4.61}$$

The kinematical boundary conditions are

$$\begin{gathered} v(0) \\ v(l) \\ \frac{dv(0)}{dx} = -\varphi(0) \\ \frac{dv(l)}{dx} = -\varphi(l) \end{gathered} \tag{4.62}$$

The dynamic boundary conditions are

$$\frac{d^2v(0)}{dx^2} = -\frac{M_b(0)}{I_zE}$$
$$\frac{d^2v(l)}{dx^2} = -\frac{M_b(l)}{I_zE} \tag{4.63}$$

In planar beam elements (see Figure 4.6.) there are two degrees of freedom at a node in its local coordinate system. They can deflect in the direction y and rotate in the plane $x - y$. Therefore, each beam element has a total of four degrees of freedom.

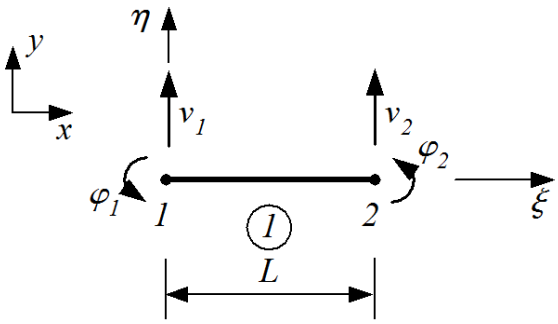


Figure 4.6. Beam element and its local coordinate system

Similar to the truss structures, to develop the finite element equations, shape functions for the interpolation of the variables from the nodal variables have to be developed. Because of the four degrees of freedom there should be four shape functions. To derive the four shape functions in the local coordinates, the displacement in an element is assumed as a third order polynomial of ξ which contains four unknown constants,

$$v^e(\xi) = C_0 + C_1\xi + C_2\xi^2 + C_3\xi^3. \tag{4.64}$$

The third order polynomial is chosen since there are four unknown constants in the polynomial, which can be related to the four nodal degrees of freedom in the beam element.
The displacement field

$$\mathbf{u}^e(\xi) = [v^e(\xi)]. \tag{4.65}$$

The nodes are denoted i and j. The nodal displacement parameter vector

$$\mathbf{q}^e = \begin{bmatrix} \mathbf{q}_i^e \\ \mathbf{q}_j^e \end{bmatrix}, \tag{4.66}$$

where

$$\mathbf{q}_i^e = \begin{bmatrix} v_i \\ \varphi_i \end{bmatrix}^e, \quad \mathbf{q}_j^e = \begin{bmatrix} v_j \\ \varphi_j \end{bmatrix}^e. \tag{4.67}$$

Now,

$$
\begin{aligned}
v^e(\xi=0) &= v_i^e = C_0 \\
\varphi^e(\xi=0) &= -\frac{dv^e(\xi=0)}{d\xi} = \varphi_i^e = -C_1 \\
v^e(\xi=L) &= v_j^e = C_0 + C_1 L + C_2 L^2 + C_3 L^3 \\
\varphi^e(\xi=L) &= -\frac{dv^e(\xi=L)}{d\xi} = \varphi_j^e = -C_1 - 2C_2 L - 3C_3 L^2
\end{aligned}
\tag{4.68}
$$

The displacement in an element can be written by the shape functions

$$
v^e(\xi) = N_1^e(\xi) v_i^e + N_2^e(\xi)\varphi_i^e + N_3^e(\xi) v_j^e + N_4^e(\xi)\varphi_j^e, \tag{4.69}
$$

which can be written in matrix form

$$
[v^e(\xi)] = [N_1^e(\xi) \quad N_2^e(\xi) \quad N_3^e(\xi) \quad N_4^e(\xi)] \begin{bmatrix} v_i^e \\ \varphi_i^e \\ v_j^e \\ \varphi_j^e \end{bmatrix} \tag{4.70}
$$

and

$$
\mathbf{u}^e(\xi) = \mathbf{N}^e(\xi)\mathbf{q}^e. \tag{4.71}
$$

The shape functions (see Figure 4.7.) are found to be

$$
\begin{aligned}
N_1^e(\xi) &= 1 - 3\left(\frac{\xi}{L}\right)^2 + 2\left(\frac{\xi}{L}\right)^3 \\
N_2^e(\xi) &= L\left[-\frac{\xi}{L} + 2\left(\frac{\xi}{L}\right)^2 - \left(\frac{\xi}{L}\right)^3\right] \\
N_3^e(\xi) &= 3\left(\frac{\xi}{L}\right)^2 - 2\left(\frac{\xi}{L}\right)^3 \\
N_4^e(\xi) &= L\left[\left(\frac{\xi}{L}\right)^2 - \left(\frac{\xi}{L}\right)^3\right]
\end{aligned}
\tag{4.72}
$$

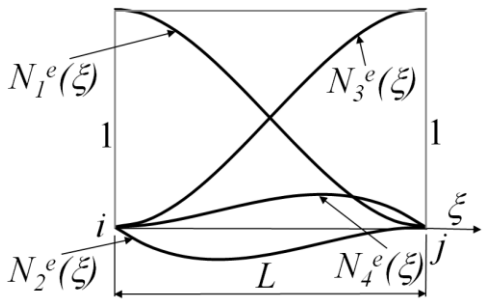


Figure 4.7. The shape functions

where

$$
\begin{aligned}
N_1^e(\xi = 0) &= 1 \\
N_1^e(\xi = L) &= 0 \\
N_2^e(\xi = 0) &= 0 \\
N_2^e(\xi = L) &= 0 \\
N_3^e(\xi = 0) &= 0 \\
N_3^e(\xi = L) &= 1 \\
N_4^e(\xi = 0) &= 0 \\
N_4^e(\xi = L) &= 0
\end{aligned}
\tag{4.73}
$$

With the assumption on the displacements the corresponding element strains can be evaluated,

$$
\boldsymbol{\varepsilon}^e(\xi) = [\kappa^e(\xi)] = \left[-\frac{d^2 v^e(\xi)}{d\xi^2}\right] = \underbrace{\left[-\frac{d^2}{d\xi^2}\right]}_{\mathbf{D}^e} [v^e(\xi)] =
$$

$$
= \underbrace{\begin{bmatrix} -\frac{d^2 N_1^e(\xi)}{d\xi^2} & -\frac{d^2 N_2^e(\xi)}{d\xi^2} & -\frac{d^2 N_3^e(\xi)}{d\xi^2} & -\frac{d^2 N_4^e(\xi)}{d\xi^2} \end{bmatrix}}_{\mathbf{B}^e(\xi)} \begin{bmatrix} v_i^e \\ \varphi_i^e \\ v_j^e \\ \varphi_j^e \end{bmatrix}, \tag{4.74}
$$

$$
\boldsymbol{\varepsilon}^e(\xi) = \mathbf{D}^e \mathbf{N}^e(\xi) \mathbf{q}^e = \mathbf{B}^e(\xi) \mathbf{q}^e, \tag{4.75}
$$

where $\mathbf{D}^e$ is the differential matrix, $\mathbf{B}^e$ is the strain-displacement matrix. The stresses in a finite element are related to the element strains, using Eq. 4.60,

$$
[M_b(\xi)] = \left[-I_\zeta E \frac{d^2 v^e(\xi)}{d\xi^2}\right] = \underbrace{[I_\zeta E]}_{\mathbf{C}^e} [\kappa^e(\xi)], \tag{4.76}
$$

$$
\boldsymbol{\sigma}^e(\xi) = \mathbf{C}^e \mathbf{B}^e(\xi) \mathbf{q}^e, \tag{4.77}
$$

where $\mathbf{C}^e$ is the elasticity matrix.
The stiffness matrix of the beam element

$$
\mathbf{K}^e = \int_L \mathbf{B}^{eT}(\xi)\, \mathbf{C}^e \mathbf{B}^e(\xi) d\xi, \tag{4.78}
$$

$$
\mathbf{K}^e = \frac{I_\zeta E}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} = \begin{bmatrix} \mathbf{K}_{ii} & \mathbf{K}_{ij} \\ \mathbf{K}_{ji} & \mathbf{K}_{jj} \end{bmatrix}^e. \tag{4.79}
$$

The total potential energy of the element

$$
\Pi^e = \frac{1}{2} \begin{bmatrix} \mathbf{q}_i^{eT} & \mathbf{q}_j^{eT} \end{bmatrix} \begin{bmatrix} \mathbf{K}_{ii} & \mathbf{K}_{ij} \\ \mathbf{K}_{ji} & \mathbf{K}_{jj} \end{bmatrix}^e \begin{bmatrix} \mathbf{q}_i^e \\ \mathbf{q}_j^e \end{bmatrix} - \begin{bmatrix} \mathbf{q}_i^{eT} & \mathbf{q}_j^{eT} \end{bmatrix} \begin{bmatrix} \mathbf{f}_i^e \\ \mathbf{f}_j^e \end{bmatrix}. \tag{4.80}
$$

Example 6.
Consider the beam shown in Figure 4.8. Solve for the displacement response.

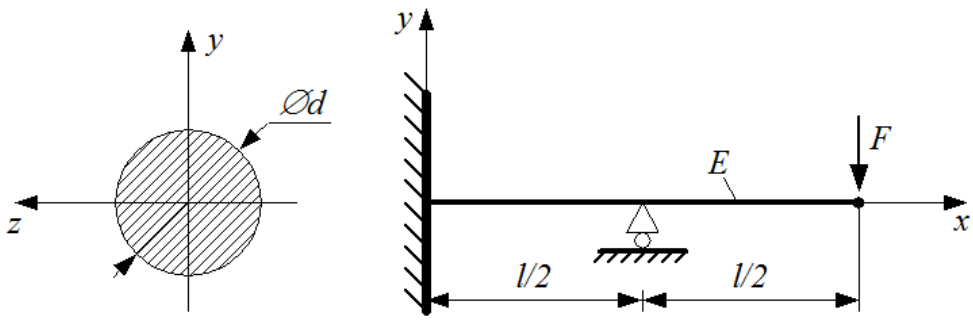


Figure 4.8. A beam loaded by a concentrate force

Data:
$l = 1000mm$
$d = 30mm$
$E = 69000MPa$
$F = 500N$

$v_i = ?\ (i = 1,2,3)$
$\varphi_i = ?\ (i = 1,2,3)$
$F_{1y} = ?$
$M_1 = ?$
$F_{2y} = ?$

The formulation of the beam is a generalization of the Euler-Bernoulli theorem. The cross section of the beam is assumed to be rigid in its own plane so no distortion of the cross section is considered.
We consider the beam as an assemblage of two beam elements. The finite element model of the problem can be seen in Figure 4.9.

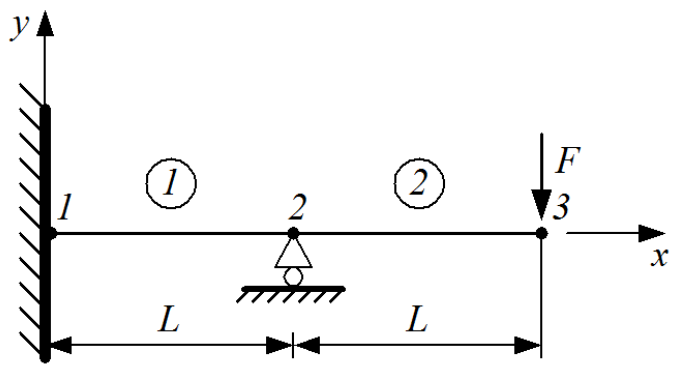


Figure 4.9. Finite element model of the problem

The nodal displacement vector is

$$q_i = \begin{bmatrix} v_i \\ \varphi_i \end{bmatrix}, \qquad i = 1,2,3.$$

First we need to find the stiffness matrix for the two elements. The stiffness matrices of the elements using Eq. 4.79 are

$$\mathbf{K}^1 = \mathbf{K}^2 = \frac{I_z E}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix},$$

where

$$\mathbf{K}^1 = \begin{bmatrix} \mathbf{K}_{11}^1 & \mathbf{K}_{12}^1 \\ \mathbf{K}_{21}^1 & \mathbf{K}_{22}^1 \end{bmatrix}, \qquad \mathbf{K}^2 = \begin{bmatrix} \mathbf{K}_{22}^2 & \mathbf{K}_{23}^2 \\ \mathbf{K}_{32}^2 & \mathbf{K}_{33}^2 \end{bmatrix}.$$

The nodal load vectors are

$$\mathbf{f}^1 = \begin{bmatrix} \mathbf{f}_1^1 \\ \mathbf{f}_2^1 \end{bmatrix}, \qquad \mathbf{f}^2 = \begin{bmatrix} \mathbf{f}_2^2 \\ \mathbf{f}_3^2 \end{bmatrix}.$$

The linear algebraic equation system $\mathbf{Kq} = \mathbf{f}$ can be assembled

$$\begin{bmatrix} \mathbf{K}_{11}^1 & \mathbf{K}_{12}^1 & \mathbf{0} \\ \mathbf{K}_{21}^1 & \mathbf{K}_{22}^1 + \mathbf{K}_{22}^2 & \mathbf{K}_{23}^2 \\ \mathbf{0} & \mathbf{K}_{32}^2 & \mathbf{K}_{33}^2 \end{bmatrix}\begin{bmatrix} \mathbf{q}_1 \\ \mathbf{q}_2 \\ \mathbf{q}_3 \end{bmatrix} = \begin{bmatrix} \mathbf{f}_1^1 \\ \mathbf{f}_2^1 + \mathbf{f}_2^2 \\ \mathbf{f}_3^2 \end{bmatrix},$$

in detail

$$\frac{I_z E}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 \\ -12 & -6L & 24 & 0 & -12 & 6L \\ 6L & 2L^2 & 0 & 8L^2 & -6L & 2L^2 \\ 0 & 0 & -12 & -6L & 12 & -6L \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{bmatrix} v_1 \\ \varphi_1 \\ v_2 \\ \varphi_2 \\ v_3 \\ \varphi_3 \end{bmatrix} = \begin{bmatrix} F_{1y} \\ M_1 \\ F_{2y} \\ 0 \\ -F \\ 0 \end{bmatrix}.$$

The reaction forces F_{1y}, M_1 and F_{2y} are unknowns. According to the kinematical boundary conditions, $v_1 = 0, \varphi_1 = 0$ and $v_2 = 0$.
Applying the kinematical boundary conditions the linear algebraic equation system will be simpler

$$\frac{I_z E}{L^3}\begin{bmatrix} 8L^2 & -6L & 2L^2 \\ -6L & 12 & -6L \\ 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{bmatrix} \varphi_2 \\ v_3 \\ \varphi_3 \end{bmatrix} = \begin{bmatrix} 0 \\ -F \\ 0 \end{bmatrix}.$$

Now we have to solve this equation system for the unknown displacement coordinates from

$$\begin{aligned} 8L^2\varphi_2 - 6Lv_3 + 2L^2\varphi_3 &= 0 \\ \frac{I_z E}{L^3}(-6L\varphi_2 + 12v_3 - 6L\varphi_3) &= -F \\ 2L^2\varphi_2 - 6Lv_3 + 4L^2\varphi_3 &= 0 \end{aligned}$$

equation system. It results

$$\varphi_2 = -\frac{FL^2}{4I_zE} = -0{,}01139rad$$
$$v_3 = -\frac{7FL^3}{12I_zE} = -13{,}289mm$$
$$\varphi_3 = -\frac{3FL^2}{4I_zE} = -0{,}034171rad$$

Finally the reaction forces are calculated by back-substituting the displacement coordinates in the structural equation.

$$F_{1y} = -750N$$
$$M_1 = -125000Nmm$$
$$F_{2y} = 1250N$$

Theoretically, coordinate transformation can also be used to transform beam element from the local coordinate system into the global coordinate system. This transformation is necessary only in the case if the beam structure containing more than one beam element has at least two beam elements of different orientations. This structure is called frame. The coordinate transformation is not detailed in this note.

5. TWO-DIMENSIONAL PROBLEMS OF THE LINEAR ELASTICITY

The analytical investigation of the three-dimensional problems is very difficult. Problems can be modeled in many cases as a two-dimensional problem. Under certain conditions, the state of strains and stresses can be simplified. A general three-dimensional structure analysis can be reduced to a two-dimensional one. In the following two-dimensional problems of linear elasticity are examined in detail.

5.1. Plane strain

Consider an infinitely long prismatic body, shown in Figure 5.1. If the body forces and external forces on the lateral boundaries are independent of the coordinate z and have no component in z direction the displacement field $\boldsymbol{u}$ within the body can be written

$$\boldsymbol{u} = u\boldsymbol{i} + v\boldsymbol{j}, \tag{5.1}$$

where

$$u = u(x, y), \quad v = v(x, y), \quad w \equiv 0. \tag{5.2}$$

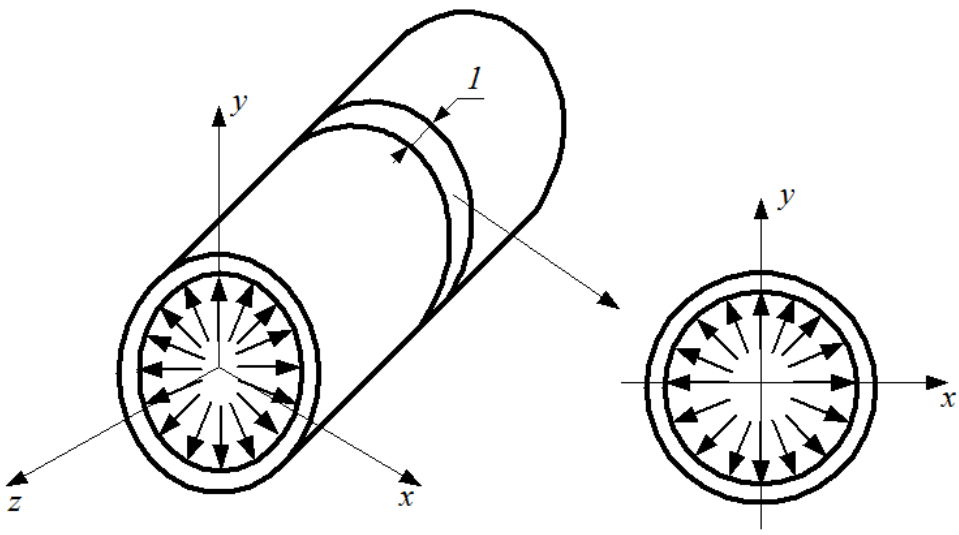


Figure 5.1. Plane strain conditions

This deformation is a state of plane strain. Using the strain-displacement relation the strain tensor is

$$\boldsymbol{A} = \begin{bmatrix} \varepsilon_x & \frac{1}{2}\gamma_{xy} & 0 \\ \frac{1}{2}\gamma_{yx} & \varepsilon_y & 0 \\ 0 & 0 & 0 \end{bmatrix}, \tag{5.3}$$

where

$$
\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x}, \\
\varepsilon_y &= \frac{\partial v}{\partial y}, \\
\varepsilon_z &= \frac{\partial w}{\partial z} \equiv 0, \\
\gamma_{xy} = \gamma_{yx} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}, \\
\gamma_{yz} = \gamma_{zy} &= \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \equiv 0, \\
\gamma_{zx} = \gamma_{xz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \equiv 0.
\end{aligned} \tag{5.4}
$$

Using the stress-strain relations (Hooke's law), the stresses corresponding to this plane

$$
\boldsymbol{T} = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & \sigma_z \end{bmatrix}, \tag{5.5}
$$

where

$$
\begin{aligned}
\sigma_x &= 2G\left[\varepsilon_x + \frac{\nu}{1-2\nu}\left(\varepsilon_x + \varepsilon_y\right)\right], \\
\sigma_y &= 2G\left[\varepsilon_y + \frac{\nu}{1-2\nu}\left(\varepsilon_x + \varepsilon_y\right)\right], \\
\sigma_z &= 2G\left[\frac{\nu}{1-2\nu}\left(\varepsilon_x + \varepsilon_y\right)\right] = \nu\left(\sigma_x + \sigma_y\right), \\
\tau_{xy} &= \tau_{yx} = G\gamma_{xy}, \\
\tau_{yz} &= \tau_{zy} = G\gamma_{yz} \equiv 0, \\
\tau_{zx} &= \tau_{xz} = G\gamma_{zx} \equiv 0.
\end{aligned} \tag{5.6}
$$

It can be seen that σ_z can be easily determined by the σ_x and σ_y.
For plane strain the equilibrium equation has the following form

$$
\begin{aligned}
\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + q_x &= 0, \\
\frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + q_y &= 0.
\end{aligned} \tag{5.7}
$$

Now we can see that in the case of plane strain problem there are 8 unknown parameters ($u, v, \varepsilon_x, \varepsilon_y, \gamma_{xy}, \sigma_x, \sigma_y$ and τ_{xy}) have to be determined with the 8 equations available.
The kinematical boundary condition

$$
\boldsymbol{u} = \tilde{\boldsymbol{u}}, \qquad \boldsymbol{r} \in A_u, \tag{5.8}
$$

where $\tilde{\boldsymbol{u}}$ is the prescribed displacement field (at fixed boundaries $\tilde{\boldsymbol{u}} = \boldsymbol{0}$),

$$
u = \tilde{u}, \qquad v = \tilde{v}. \tag{5.9}
$$

The dynamic boundary condition can be described by

$$\boldsymbol{T} \cdot \boldsymbol{n} = \tilde{\boldsymbol{p}}, \qquad \boldsymbol{r} \in A_p, \tag{5.10}$$

where $\tilde{\boldsymbol{p}}$ is the prescribed force vector. The Eq. 5.10 in scalar notation is

$$\begin{aligned} \sigma_x n_x + \tau_{xy} n_y &= \tilde{p}_x, \\ \tau_{yx} n_x + \sigma_y n_y &= \tilde{p}_y. \end{aligned} \tag{5.11}$$

In plane strain the direction of the stress is perpendicular to the $x - y$ plane that is not zero. However, the strain in that direction is zero, and therefore no contribution to internal work is made by this stress. When computing, the stress in z direction can be explicitly evaluated from the three stress components.

5.2. Plane stress

The plane stress theory applies to domains bounded by two parallel planes separated by a distance which is small compared to the other dimensions of the body as shown in Figure 5.2.

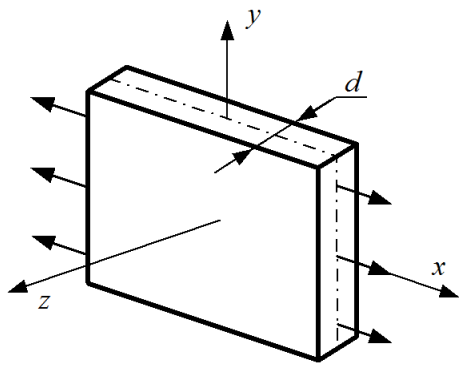


Figure 5.2. Plane stress conditions

For describing the problem the $x - y$ plane is chosen. The domain is bounded by two planes $z = \pm\frac{d}{2}$, see in Figure 5.2. We also assume that $\sigma_z = \tau_{yz} = \tau_{zx} = 0$ on each face.
In the case of plane stress the stress state

$$\boldsymbol{T} = \begin{bmatrix} \sigma_x & \tau_{xy} & 0 \\ \tau_{yx} & \sigma_y & 0 \\ 0 & 0 & 0 \end{bmatrix}, \tag{5.12}$$

where

$$\sigma_x = \sigma_x(x, y), \quad \sigma_y = \sigma_y(x, y), \quad \sigma_z = \tau_{yz} = \tau_{zx} = 0. \tag{5.13}$$

Using the plane stress state, the corresponding strain field can be determined by the general Hooke's law

$$
\begin{aligned}
\varepsilon_x &= \frac{1}{E}(\sigma_x - \nu\sigma_y),\\
\varepsilon_y &= \frac{1}{E}(\sigma_y - \nu\sigma_x),\\
\varepsilon_z &= -\frac{\nu}{E}\left(\sigma_x + \sigma_y\right),\\
\gamma_{xy} &= \frac{\tau_{xy}}{G},\\
\gamma_{yz} &= \frac{\tau_{yz}}{G} \equiv 0,\\
\gamma_{zx} &= \frac{\tau_{zx}}{G} \equiv 0.
\end{aligned}
\tag{5.14}
$$

According to Eq. 5.15 the strain tensor

$$
\boldsymbol{A} = \begin{bmatrix} \varepsilon_x & \frac{1}{2}\gamma_{xy} & 0 \\ \frac{1}{2}\gamma_{yx} & \varepsilon_y & 0 \\ 0 & 0 & \varepsilon_z \end{bmatrix}. \tag{5.15}
$$

The strain-displacement equations for plane stress

$$
\begin{aligned}
\varepsilon_x &= \frac{\partial u}{\partial x},\\
\varepsilon_y &= \frac{\partial v}{\partial y},\\
\varepsilon_z &= \frac{\partial w}{\partial z},\\
\gamma_{xy} = \gamma_{yx} &= \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x},\\
\gamma_{yz} = \gamma_{zy} &= \frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \equiv 0,\\
\gamma_{zx} = \gamma_{xz} &= \frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \equiv 0.
\end{aligned}
\tag{5.16}
$$

The equilibrium equation is the same as the case of plain strain, thus

$$
\begin{aligned}
\frac{\partial \sigma_x}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + q_x &= 0,\\
\frac{\partial \tau_{yx}}{\partial x} + \frac{\partial \sigma_y}{\partial y} + q_y &= 0.
\end{aligned}
\tag{5.17}
$$

Now we can see that in the case of plane stress problem there are 8 unknown parameters ($u, v, \varepsilon_x, \varepsilon_y, \gamma_{xy}, \sigma_x, \sigma_y$ and τ_{xy}) have to determined with the 8 equations available.
The kinematical boundary condition

$$
\boldsymbol{u} = \tilde{\boldsymbol{u}}, \qquad \boldsymbol{r} \in A_u, \tag{5.18}
$$

where $\tilde{\boldsymbol{u}}$ is the prescribed displacement field (at fixed boundaries $\tilde{\boldsymbol{u}} = \boldsymbol{0}$),

$$u = \tilde{u}, \qquad v = \tilde{v}. \tag{5.19}$$

The dynamic boundary condition can be described by

$$\boldsymbol{T} \cdot \boldsymbol{n} = \tilde{\boldsymbol{p}}, \qquad \boldsymbol{r} \in A_p, \tag{5.20}$$

where $\tilde{\boldsymbol{p}}$ is the prescribed force vector. The Eq. 5.10 in scalar notation is

$$\begin{aligned} \sigma_x n_x + \tau_{xy} n_y &= \tilde{p}_x, \\ \tau_{yx} n_x + \sigma_y n_y &= \tilde{p}_y. \end{aligned} \tag{5.21}$$

The only strains and stresses in plane stress problems have to be considered are the three components in the $x - y$ plane. In the case of plane stress, all other components of stress are zero therefore they give no contribution to the internal work.

5.3. Axisymmetric problems

If an axisymmetric solid body is loaded and supported axisymmetrically then the same strain state and stress state occur in every meridian section of the body, see in Figure 5.3. Because of the symmetrical layout the two components of displacements in any plane section of the body along its axis of symmetry define completely the state of strain and the state of stress.

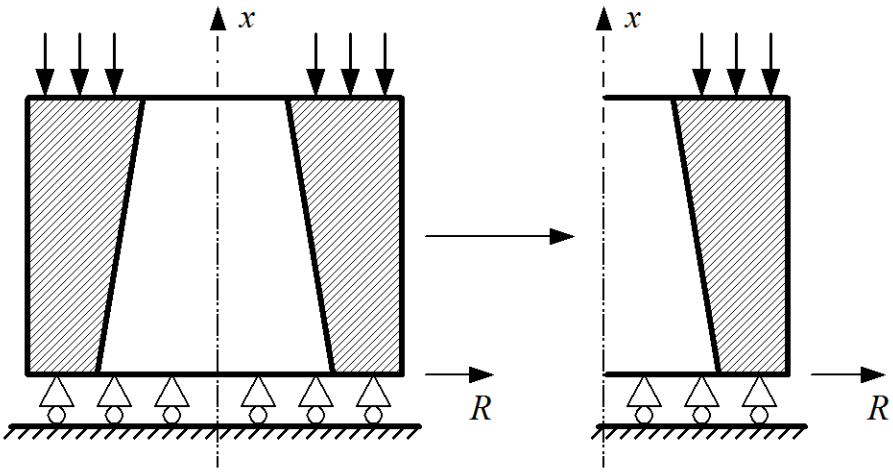


Figure 5.3. Axisymmetric problem conditions

Using polar coordinates R, φ, x the strain tensor

$$\boldsymbol{A} = \begin{bmatrix} \varepsilon_R & 0 & \frac{1}{2}\gamma_{Rx} \\ 0 & \varepsilon_\varphi & 0 \\ \frac{1}{2}\gamma_{xR} & 0 & \varepsilon_x \end{bmatrix}. \tag{5.22}$$

Using the stress-strain relations (Hooke's law), the stresses corresponding to this meridian section

$$\boldsymbol{T} = \begin{bmatrix} \sigma_R & 0 & \tau_{Rx} \\ 0 & \sigma_\varphi & 0 \\ \tau_{xR} & 0 & \sigma_x \end{bmatrix}. \tag{5.23}$$

6. FINITE ELEMENT FORMULATIONS FOR TWO-DIMENSIONAL PROBLEMS

Note that only the formulation for plane strain and plane stress is introduced. For two-dimensional problems the matrix of the displacement vector

$$\mathbf{u} = \underbrace{\begin{bmatrix} u(x,y) \\ v(x,y) \end{bmatrix}}_{(2x1)}. \tag{6.1}$$

The matrix of the strain vector

$$\boldsymbol{\varepsilon} = \underbrace{\begin{bmatrix} \varepsilon_x(x,y) \\ \varepsilon_y(x,y) \\ \gamma_{xy}(x,y) \end{bmatrix}}_{(3x1)}. \tag{6.2}$$

The matrix of the stress vector

$$\boldsymbol{\sigma} = \underbrace{\begin{bmatrix} \sigma_x(x,y) \\ \sigma_y(x,y) \\ \tau_{xy}(x,y) \end{bmatrix}}_{(3x1)}. \tag{6.3}$$

Assuming linear analysis conditions the displacements are infinitesimally small. A two-dimensional finite element is defined by nodes and straight line boundaries. The displacements are approximated element by element,

$$\underbrace{\mathbf{u}^e}_{(2x1)} = \underbrace{\mathbf{N}^e}_{(2xn)} \underbrace{\mathbf{q}^e}_{(nx1)}, \tag{6.4}$$

where $\mathbf{u}^e$ is the displacement field of the element, $\mathbf{N}^e$ is the approximation (shape function) matrix of the element, $\mathbf{q}^e$ is the nodal displacement vector of the element, n is the degrees of freedom of the element. The shape function matrix contains the approximation functions.
The nodal displacement vector of element e is

$$\mathbf{q}^e = \underbrace{\begin{bmatrix} \mathbf{q}_i \\ \mathbf{q}_j \\ \mathbf{q}_k \\ \vdots \\ \mathbf{q}_N \end{bmatrix}^e}_{(2Nx1)}, \tag{6.5}$$

where N is the number of nodes. The nodal displacement vector of element i is

$$\mathbf{q}_i^e = \underbrace{\begin{bmatrix} u_i \\ v_i \end{bmatrix}^e}_{(2x1)}. \tag{6.6}$$

The shape function matrix is predefined to assume shapes of the displacement variations with respect to the coordinates. The general form of the shape function matrix is

$$\mathbf{N}^e = \underbrace{\begin{bmatrix} \mathbf{N}_i \\ \mathbf{N}_j \\ \mathbf{N}_k \\ \vdots \\ \mathbf{N}_N \end{bmatrix}^e}_{(2Nx2)}. \tag{6.7}$$

The shape function block belonging to the ith node of the element e is

$$\mathbf{N}_i^e = \underbrace{\begin{bmatrix} N_{xxi} & N_{xyi} \\ N_{yxi} & N_{yyi} \end{bmatrix}^e}_{(2x2)}. \tag{6.8}$$

Considering the displacements the strain state of an element can be approximated,

$$\boldsymbol{\varepsilon}^e = \underbrace{\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}^e}_{(3x1)} = \underbrace{\begin{bmatrix} \dfrac{\partial u}{\partial x} \\ \dfrac{\partial v}{\partial y} \\ \dfrac{\partial u}{\partial y} + \dfrac{\partial v}{\partial x} \end{bmatrix}^e}_{(3x1)} = \underbrace{\begin{bmatrix} \dfrac{\partial}{\partial x} & 0 \\ 0 & \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial x} \end{bmatrix}^e}_{(3x2)} \underbrace{\begin{bmatrix} u \\ v \end{bmatrix}^e}_{(2x1)}. \tag{6.9}$$

Here

$$\mathbf{D}^e = \underbrace{\begin{bmatrix} \dfrac{\partial}{\partial x} & 0 \\ 0 & \dfrac{\partial}{\partial y} \\ \dfrac{\partial}{\partial y} & \dfrac{\partial}{\partial x} \end{bmatrix}^e}_{(3x2)} \tag{6.10}$$

the differential matrix, thus the strains can be expressed as

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}^e\mathbf{q}^e = \mathbf{B}^e\mathbf{q}^e. \tag{6.11}$$

where $\mathbf{B}^e$ is the strain displacement matrix

$$\mathbf{B}_i^e = \underbrace{\begin{bmatrix} \frac{\partial N_i}{\partial x} & 0 \\ 0 & \frac{\partial N_i}{\partial y} \\ \frac{\partial N_i}{\partial y} & \frac{\partial N_i}{\partial x} \end{bmatrix}^e}_{(3x2)}. \tag{6.12}$$

Considering the strains the stress state of an element can be approximated using the Hooke's law considering isotropic material

$$\boldsymbol{\sigma}^e = \underbrace{\begin{bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{bmatrix}^e}_{(3x1)} = \underbrace{\begin{bmatrix} c_1 & c_2 & 0 \\ c_2 & c_1 & 0 \\ 0 & 0 & c_3 \end{bmatrix}^e}_{(3x3)} \underbrace{\begin{bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{bmatrix}^e}_{(3x1)}, \tag{6.13}$$

where

	Plain strain	Plain stress
c_1	$\frac{E(1-\nu)}{(1-2\nu)(1+\nu)}$	$\frac{E}{1-\nu^2}$
c_2	$\frac{E\nu}{(1-2\nu)(1+\nu)}$	$\frac{E\nu}{1-\nu^2}$
c_3	$\frac{E}{2(1+\nu)} = G$	$\frac{E}{2(1+\nu)} = G$

The elasticity matrix is introduced as

$$\mathbf{C}^e = \underbrace{\begin{bmatrix} c_1 & c_2 & 0 \\ c_2 & c_1 & 0 \\ 0 & 0 & c_3 \end{bmatrix}^e}_{(3x3)}, \tag{6.14}$$

from which the stress is

$$\boldsymbol{\sigma}^e = \mathbf{C}^e \boldsymbol{\varepsilon}^e = \mathbf{C}^e \mathbf{B}^e \mathbf{q}^e. \tag{6.15}$$

According to Hooke's law the stresses in a finite element are related to the element strains, where $\mathbf{C}^e$ is the elasticity matrix of element.
The total potential energy of the element according to Eq. 3.8 is

$$\Pi^e = \frac{1}{2}\int_V \boldsymbol{\varepsilon}^{eT} \boldsymbol{\sigma}^e \, dV - \int_V \mathbf{u}^{eT} \mathbf{q}^e \, dV - \int_{A_p} \mathbf{u}^{eT} \mathbf{p}^e \, dA. \tag{6.16}$$

Using Eq. 6.11 and Eq. 6.15 and taking the constants from the integrals the total potential energy of the element is

$$\Pi^e = \frac{1}{2}\mathbf{q}^{eT}\int_V \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e\, dV\mathbf{q}^e - \mathbf{q}^{eT}\int_V \mathbf{N}^{eT}\mathbf{q}^e\, dV - \mathbf{q}^{eT}\int_{A_p} \mathbf{N}^{eT}\mathbf{p}^e\, dA, \tag{6.17}$$

where

$$\mathbf{K}^e = \int_V \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e\, dV \tag{6.18}$$

is the stiffness matrix of the element. The stiffness matrix of the element is symmetric, thus $\mathbf{K}^e = \mathbf{K}^{eT}$.
Note that in the case of the two-dimensional problem the dimension in z direction is 1. This means

$$dV = dxdydz \;\rightarrow\; dA = dxdy. \tag{6.19}$$

According to Eq. 6.19 the stiffness matrix

$$\mathbf{K}^e = \int_A \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e\, dA. \tag{6.20}$$

The nodal load vector with respect to Eq. 6.19 is

$$\mathbf{f}_q^e = \int_A \mathbf{N}^{eT}\mathbf{q}^e\, dA. \tag{6.21}$$

The nodal load vector with respect to the surface forces is

$$\mathbf{f}_p^e = \int_{A_p} \mathbf{N}^{eT}\mathbf{p}^e\, dA. \tag{6.22}$$

The nodal load vector of the element is

$$\mathbf{f}^e = \mathbf{f}_q^e + \mathbf{f}_p^e. \tag{6.23}$$

Using the notations the total potential energy of the element is

$$\Pi^e = \frac{1}{2}\mathbf{q}^{eT}\mathbf{K}^e\mathbf{q}^e - \mathbf{q}^{eT}\mathbf{f}^e. \tag{6.24}$$

The principle of minimum total potential can be applied only at the examined boundary. It is not valid for individual elements. The total potential energy of the examined boundary is equal with the sum of the potential energy of the elements,

$$\Pi = \sum_{e=1}^{Q} \Pi^e = \frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \mathbf{q}^{eT}\mathbf{f}, \tag{6.25}$$

where Q is the number of finite elements applied for the discretization, $\mathbf{K}$ is the stiffness matrix, $\mathbf{q}$ is the nodal displacement vector and $\mathbf{f}$ is the nodal force vector.
According to the Lagrange variational principle $\delta\Pi = 0$, the Eq. 6.25 can be written

$$\delta\Pi = \delta\left(\frac{1}{2}\mathbf{q}^T\mathbf{K}\mathbf{q} - \mathbf{q}^{eT}\mathbf{f}\right) = \delta\mathbf{q}^T(\mathbf{K}\mathbf{q} - \mathbf{f}) = 0. \tag{6.26}$$

The Eq. 4.25 results in

$$\mathbf{K}\mathbf{q} = \mathbf{f}. \tag{6.27}$$

In linear analysis the finite element system equilibrium equations can be solved.

7. ISOPARAMETRIC FINITE ELEMENTS

The development of element matrices and equations expressed in the global coordinate system becomes a very difficult task. This is why the isoparametric formulation was developed. The isoparametric method leads to a simple computer program formulation applicable for one-, two- and three-dimensional stress analysis. The isoparametric formulation also allows elements to be created which are irregular polynoms having curved sides. Several commercial finite element softwares adapt this formulation for their element libraries.
The idea of the isoparametric finite element formulation is to achieve the relationship between the element displacements at any point and the element nodal point displacements using the shape functions. Instead of evaluating the transformation matrix the element matrices corresponding to the required degrees of freedom are obtained directly. The interpolation of the element coordinates and element displacements using the same interpolation function, which are defined in a local coordinate system is the basis of the isoparametric finite element formulation. Hence, in the isoparametric formulation the element displacements are interpolated in the same way as the geometry. Typical finite elements are illustrated in Figure 7.1.

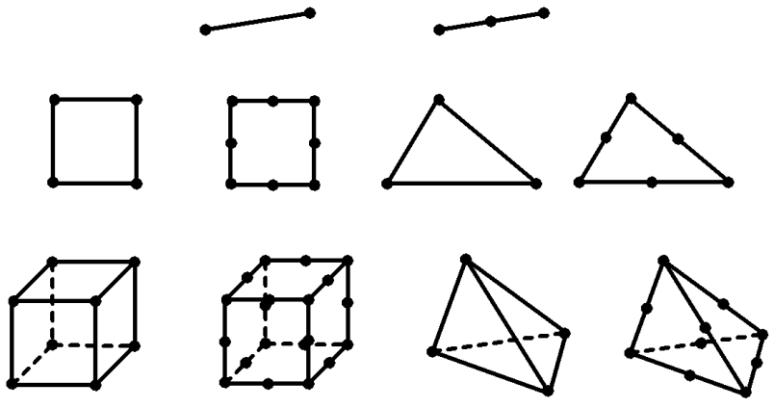


Figure 7.1. Typical finite elements

The basic procedure in the isoparametric finite element formulation is to express the element coordinates and element displacements in the form of interpolations using the local coordinate system of the element. The natural ξ, η, ζ coordinates (depending on the dimension of the problem) are attached to the element with the origin at the center of the element. The ξ, η, ζ axes need to be orthogonal to each other.

7.1. One-dimensional isoparametric mapping

7.1.1. Truss element

The geometric description of a truss element (two node element) and its mapping can be seen in Figure 7.2. There are two nodes at the ends.

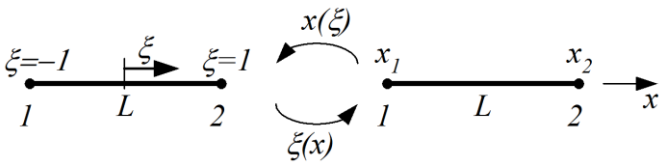


Figure 7.2. The truss element

Considering a truss element, the coordinate interpolation

$$x(\xi) = \sum_{i=1}^{2} N_i x_i = N_1 x_1 + N_2 x_2, \tag{7.1}$$

where the shape functions (see Figure 7.3) are

$$\begin{aligned} N_1(\xi) &= \frac{1}{2}(1-\xi), \\ N_2(\xi) &= \frac{1}{2}(1+\xi). \end{aligned} \tag{7.2}$$

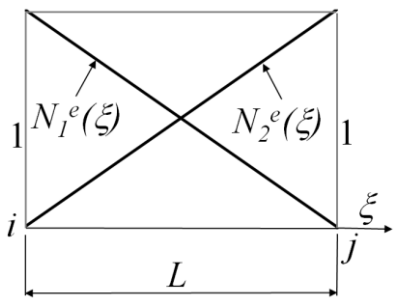


Figure 7.3. Shape functions of the truss element

	$\xi = -1$	$\xi = 1$
Node 1	$N_1(\xi) = 1$	$N_1(\xi) = 0$
Node 2	$N_2(\xi) = 0$	$N_2(\xi) = 1$

The properties of the shape functions:

1.	$\sum_{i=1}^{2} N_i(\xi) = 1 \qquad -1 \le \xi \le 1$
2.	$N_i(\xi_j) = \begin{cases} 1 & if \;\; i = j \\ 0 & if \; i \neq j \end{cases}$ where ξ_j is the coordinate of the jth node $j = 1,2$

The approximation of the displacement field on the element assuming that $\xi = \xi(x)$

$$u^e(x) = \sum_{i=1}^{2} N_i(\xi) u_i^e = N_1(\xi) u_1^e + N_2(\xi) u_2^e, \tag{7.3}$$

where u_i^e is the displacement coordinates of the ith node of the element e. The purpose of the investigation is to determine these nodal values. In matrix form

$$\mathbf{u}^e = \underbrace{[u]^e}_{(1x1)} = \underbrace{[N_1 \quad N_2]}_{(2x1)} \underbrace{\begin{bmatrix} u_1 \\ u_2 \end{bmatrix}^e}_{(2x1)} = \mathbf{N}\mathbf{q}^e, \tag{7.4}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element. Using Eq. 6.11 the strain vector of the element can be obtained

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}\mathbf{q}^e = \mathbf{B}\mathbf{q}^e, \tag{7.5}$$

where the strain-displacement matrix

$$\mathbf{B} = \mathbf{D}^e\mathbf{N} = \underbrace{\left[\frac{\partial}{\partial x}\right]^e}_{(1x1)} \underbrace{[N_1 \quad N_2]}_{(1x2)} = \underbrace{\left[\frac{\partial N_1}{\partial x} \quad \frac{\partial N_2}{\partial x}\right]}_{(2x1)}. \tag{7.6}$$

The derivation of the shape function can be expressed by

$$\frac{\partial N_i(\xi,\eta)}{\partial x} = \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial x}. \tag{7.7}$$

The Eq. 7.7 can be written in matrix form

$$\left[\frac{\partial}{\partial x}\right] = \underbrace{\left[\frac{\partial \xi}{\partial x}\right]}_{\mathbf{J}^{-1}} \left[\frac{\partial}{\partial \xi}\right] = \mathbf{J}^{-1}\left[\frac{\partial}{\partial \xi}\right], \tag{7.8}$$

where $\mathbf{J}$ is the Jacobian. The inverse of the Jacobian has to be known in finite element computations. The Jacobian differential operator matrix is known in possession of the functions $x = x(\xi)$, thus

$$\left[\frac{\partial}{\partial \xi}\right] = \underbrace{\left[\frac{\partial x}{\partial \xi}\right]}_{\mathbf{J}} \left[\frac{\partial}{\partial x}\right] = \mathbf{J}\left[\frac{\partial}{\partial x}\right], \tag{7.9}$$

The Jacobian can be formulated at a known mesh and geometry. The Jacobian maps a differential element from the isoparametric coordinates to the global coordinates. The $\mathbf{J}^{-1}$ can be derived by inverting $\mathbf{J}$,

$$\mathbf{J}^{-1} = \left[\frac{\partial \xi}{\partial x}\right] = \frac{1}{det\mathbf{J}}\left[\frac{\partial x}{\partial \xi}\right], \tag{7.10}$$

where $det\mathbf{J} = J > 0$.

7.1.2. Cable element

The geometric description of a cable element (three node quadratic element) and its mapping can be seen in Figure 7.4. There are two nodes at the ends and one in the middle.

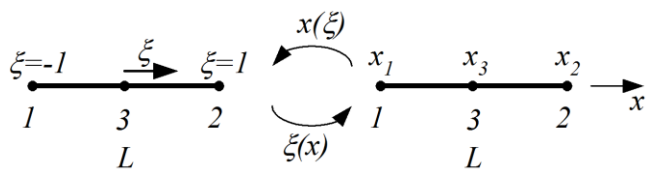


Figure 7.4. The cable element

Considering a cable element, the coordinate interpolation

$$x(\xi) = \sum_{i=1}^{3} N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3, \tag{7.11}$$

where the shape functions (see Figure 7.5) are

$$\begin{gathered} N_1(\xi) = \frac{1}{2}(1-\xi) - \frac{1}{2}(1-\xi^2) = -\frac{\xi(1-\xi)}{2}, \\ N_2(\xi) = \frac{1}{2}(1+\xi) - \frac{1}{2}(1-\xi^2) = \frac{\xi(1+\xi)}{2}, \\ N_3(\xi) = 1-\xi^2. \end{gathered} \tag{7.12}$$

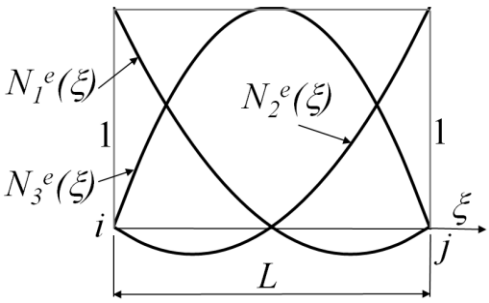


Figure 7.5. Shape functions of the cable element

	$\xi = -1$	$\xi = 1$
Node 1	$N_1(\xi) = 1$	$N_1(\xi) = 0$
Node 2	$N_2(\xi) = 0$	$N_2(\xi) = 1$
Node 3	$N_3(\xi) = 0$	$N_3(\xi) = 0$

The properties of the shape functions:

1.	$\sum_{i=1}^{3} N_i(\xi) = 1 \qquad -1 \le \xi \le 1$
2.	$N_i(\xi_j) = \begin{cases} 1 & if \;\; i = j \\ 0 & if \; i \ne j \end{cases}$ where ξ_j is the coordinate of the jth node $j = 1,2,3$

The approximation of the displacement field on the element assuming that $\xi = \xi(x)$

$$u^e(x) = \sum_{i=1}^{3} N_i(\xi) u_i^e = N_1(\xi) u_1^e + N_2(\xi) u_2^e + N_3(\xi) u_3^e, \tag{7.13}$$

where u_i^e is the displacement coordinates of the ith node of the element e. The purpose of the investigation is to determine these nodal values. In matrix form

$$\mathbf{u}^e = \underbrace{[u]^e}_{(1x1)} = \underbrace{[N_1 \quad N_2 \quad N_3]}_{(3x1)} \underbrace{\begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix}^e}_{(3x1)} = \mathbf{N}\mathbf{q}^e, \tag{7.14}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element. Using Eq. 6.11 the strain vector of the element can be obtained

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}\mathbf{q}^e = \mathbf{B}\mathbf{q}^e, \tag{7.15}$$

where the strain-displacement matrix

$$\mathbf{B} = \mathbf{D}^e\mathbf{N} = \underbrace{\left[\frac{\partial}{\partial x}\right]^e}_{(1x1)} \underbrace{[N_1 \quad N_2 \quad N_3]}_{(3x1)} = \underbrace{\left[\frac{\partial N_1}{\partial x} \quad \frac{\partial N_2}{\partial x} \quad \frac{\partial N_3}{\partial x}\right]}_{(3x1)}. \tag{7.16}$$

The method to derive the derivation of the shape function and the determination of the Jacobian is the same as the method applied for the truss element.

Example 7.

The shape functions of a three node quadratic element get mapped. In the isoparametric coordinates ξ they are polynomials. In the global coordinates x they are in general nonpolynomials. In Figure 7.6 the element and the coordinates of the nodes are given.

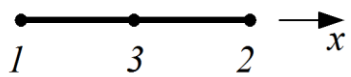


Figure 7.6. Three node quadratic element with its nodal coordinates

Compute the shape functions in the global coordinates!

$$x_1 = 0, \quad x_2 = 5, \quad x_3 = 3$$

The isoparametric mapping $x(\xi)$

$$x(\xi) = \sum_{i=1}^{3} N_i x_i = N_1x_1 + N_2x_2 + N_3x_3 =$$
$$= -\frac{\xi(1-\xi)}{2}\cdot 0 + \frac{\xi(1+\xi)}{2}\cdot 5 + (1-\xi^2)\cdot 3 = 3 + \frac{5\xi}{2} - \frac{\xi^2}{2}$$

The inverse mapping $\xi(x)$

$$\xi(x) = \frac{5-\sqrt{49-8x}}{2}$$

The shape functions in the global coordinates

$$N_1(\xi) == -\frac{\xi(1-\xi)}{2} = 8 - x - \sqrt{49-8x} = N_1(x)$$
$$N_2(\xi) = \frac{\xi(1+\xi)}{2} = \frac{21}{2} - x - \frac{3}{2}\sqrt{49-8x} = N_2(x)$$
$$N_3(\xi) = 1 - \xi^2 = 8x - 48 = N_3(x)$$

7.2. Two-dimensional isoparametric mapping

7.2.1. Linear quadrilateral element

The geometric description of a linear quadrilateral element and its mapping can be seen in Figure 7.7. There are four nodes at the corners of the quadrilateral shape.

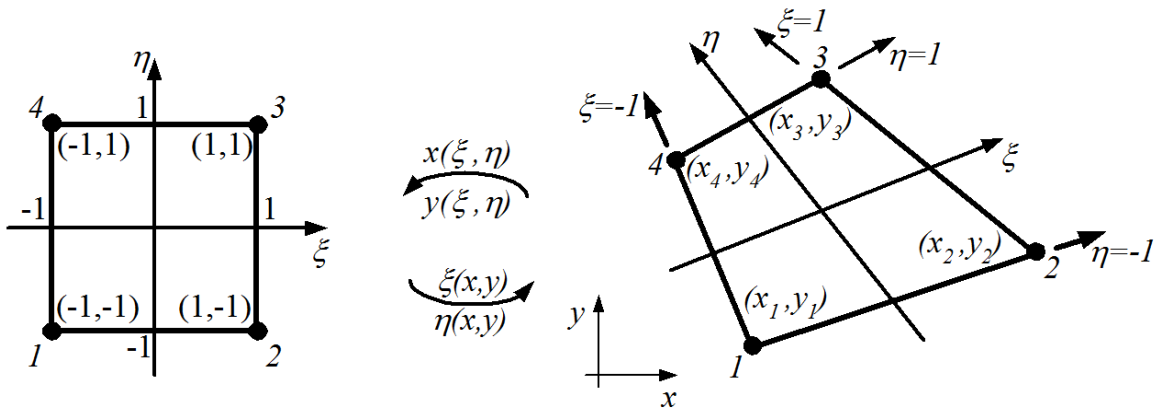


Figure 7.7. Linear quadrilateral isoparametric element

Considering a two-dimensional linear quadrilateral element, the coordinate interpolations

$$
\begin{aligned}
x(\xi,\eta) &= \sum_{i=1}^{4} N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4, \\
y(\xi,\eta) &= \sum_{i=1}^{4} N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4,
\end{aligned}
\tag{7.17}
$$

where the shape functions are

$$
\begin{aligned}
N_1(\xi,\eta) &= \frac{1}{4}(1-\xi)(1-\eta), \\
N_2(\xi,\eta) &= \frac{1}{4}(1+\xi)(1-\eta), \\
N_3(\xi,\eta) &= \frac{1}{4}(1+\xi)(1+\eta), \\
N_4(\xi,\eta) &= \frac{1}{4}(1-\xi)(1+\eta)
\end{aligned}
\tag{7.18}
$$

and linear in the $\xi - \eta$ plane.

	$\xi = -1;\ \eta = -1$	$\xi = 1;\ \eta = -1$	$\xi = 1;\ \eta = 1$	$\xi = -1;\ \eta = 1$
Node 1	$N_1(\xi,\eta) = 1$	$N_1(\xi,\eta) = 0$	$N_1(\xi,\eta) = 0$	$N_1(\xi,\eta) = 0$
Node 2	$N_2(\xi,\eta) = 0$	$N_2(\xi,\eta) = 1$	$N_2(\xi,\eta) = 0$	$N_2(\xi,\eta) = 0$
Node 3	$N_3(\xi,\eta) = 0$	$N_3(\xi,\eta) = 0$	$N_3(\xi,\eta) = 1$	$N_3(\xi,\eta) = 0$
Node 4	$N_4(\xi,\eta) = 0$	$N_4(\xi,\eta) = 0$	$N_4(\xi,\eta) = 0$	$N_4(\xi,\eta) = 1$

The properties of the shape functions:

1.	$\sum_{i=1}^{4} N_i\,(\xi,\eta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j) = \begin{cases} 1 & if\ \ i = j \\ 0 & if\ i \ne j \end{cases}$ where ξ_j, η_j is the coordinate of the jth node $j = 1,2,3,4$

The approximation of the displacement field on the element assuming that $\xi = \xi(x,y)$ and $\eta = \eta(x,y)$

$$\begin{aligned} u^e(x,y) &= \sum_{i=1}^{4} N_i\,(\xi,\eta)u_i^e = N_1(\xi,\eta)u_1^e + N_2(\xi,\eta)u_2^e + N_3(\xi,\eta)u_3^e + N_4(\xi,\eta)u_4^e, \\ v^e(x,y) &= \sum_{i=1}^{4} N_i\,(\xi,\eta)v_i^e = N_1(\xi,\eta)v_1^e + N_2(\xi,\eta)v_2^e + N_3(\xi,\eta)v_3^e + N_4(\xi,\eta)v_4^e, \end{aligned} \tag{7.19}$$

where u_i^e and v_i^e are the displacement coordinates of the ith node of the element e. The purpose of the investigation is to determine these nodal values. In matrix form

$$\mathbf{u}^e = \underbrace{\begin{bmatrix} u \\ v \end{bmatrix}^e}_{(2x1)} = \underbrace{\begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix}}_{(2x8)} \underbrace{\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{bmatrix}^e}_{(8x1)} = \mathbf{N}\mathbf{q}^e, \tag{7.20}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element. Using Eq. 6.11 the strain vector of the element can be obtained

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}\mathbf{q}^e = \mathbf{B}\mathbf{q}^e, \tag{7.21}$$

where the strain-displacement matrix

$$\begin{aligned} \mathbf{B} = \mathbf{D}^e\mathbf{N} &= \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}^e}_{(3x2)} \underbrace{\begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 & 0 & N_4 \end{bmatrix}}_{(2x8)} = \\ &= \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 & \frac{\partial N_4}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} & 0 & \frac{\partial N_4}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} & \frac{\partial N_4}{\partial y} & \frac{\partial N_4}{\partial x} \end{bmatrix} \end{aligned} \tag{7.22}$$

The derivation of the shape function can be expressed by

$$\frac{\partial N_i(\xi,\eta)}{\partial x} = \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial x}$$
$$\frac{\partial N_i(\xi,\eta)}{\partial y} = \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial y} \tag{7.23}$$

The Eq. 7.23 can be written in matrix form

$$\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix}}_{\mathbf{J}^{-1}} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix} = \mathbf{J}^{-1} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix}, \tag{7.24}$$

where $\mathbf{J}$ is the Jacobian. The inverse of the Jacobian has to be known in finite element computations. The Jacobian differential operator matrix is known in possession of the functions $x = x(\xi,\eta)$ and $y = y(\xi,\eta)$, thus

$$\begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}}_{\mathbf{J}} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix} = \mathbf{J} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix}, \tag{7.25}$$

The Jacobian can be formulated at a known mesh and geometry. The $\mathbf{J}^{-1}$ can be derived by inverting $\mathbf{J}$,

$$\mathbf{J}^{-1} = \begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix} = \frac{1}{det\mathbf{J}} \begin{bmatrix} \frac{\partial y}{\partial \eta} & -\frac{\partial y}{\partial \xi} \\ -\frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi} \end{bmatrix}, \tag{7.26}$$

where $det\mathbf{J} = J > 0$. The cases $J < 0$ and $J = 0$ are not allowed in finite element calculations, they make numerical error. Those cases can be when the element is distorted.

Example 8.

The geometry of a linear quadrilateral isoparametric element and the coordinates of the nodes are known, see in Figure 7.8.

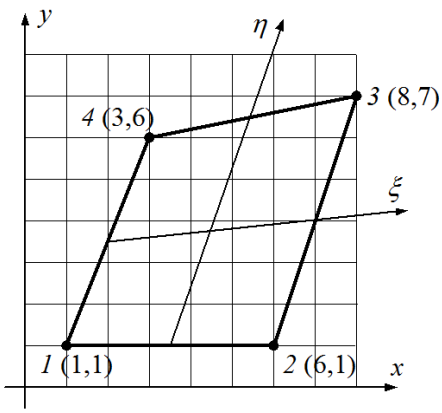


Figure 7.8. The element geometry and the coordinates of the nodes

$x(\xi,\eta) = ?$
$y(\xi,\eta) = ?$
$\mathbf{J} = ?$
$J = ?$

$x(\xi,\eta) = \sum_{i=1}^{4} N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4 =$ $= \frac{1}{4}(1-\xi)(1-\eta)\cdot 1 + \frac{1}{4}(1+\xi)(1-\eta)\cdot 6 +$ $+\frac{1}{4}(1+\xi)(1+\eta)\cdot 8 + \frac{1}{4}(1-\xi)(1+\eta)\cdot 3 = \frac{9}{2} + \frac{5}{2}\xi + \eta$
$y(\xi,\eta) = \sum_{i=1}^{4} N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4 =$ $= \frac{1}{4}(1-\xi)(1-\eta)\cdot 1 + \frac{1}{4}(1+\xi)(1-\eta)\cdot 1 +$ $+\frac{1}{4}(1+\xi)(1+\eta)\cdot 7 + \frac{1}{4}(1-\xi)(1+\eta)\cdot 6 = \frac{15}{4} + \frac{1}{4}\xi + \frac{11}{4}\eta + \frac{1}{4}\xi\eta$

The derivatives of $x(\xi,\eta)$ and $y(\xi,\eta)$ according to ξ and η have to be produced to get the Jacobian matrix,

$\frac{\partial x}{\partial \xi} = \frac{5}{2}$	$\frac{\partial y}{\partial \xi} = \frac{1}{4} + \frac{1}{4}\eta$
$\frac{\partial x}{\partial \eta} = 1$	$\frac{\partial y}{\partial \eta} = \frac{11}{4} + \frac{1}{4}\xi$

Now the Jacobian matrix can be constructed,

$$\mathbf{J} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} = \begin{bmatrix} \frac{5}{2} & \frac{1}{4} + \frac{1}{4}\eta \\ 1 & \frac{11}{4} + \frac{1}{4}\xi \end{bmatrix}$$

$$J(\xi,\eta) = detJ(\xi,\eta) = \frac{\partial x}{\partial \xi}\frac{\partial y}{\partial \eta} - \frac{\partial x}{\partial \eta}\frac{\partial y}{\partial \xi} = \frac{53}{8} + \frac{5}{8}\xi - \frac{1}{4}\eta$$

The condition for $J(\xi,\eta)$ has to be controlled

	$J(\xi,\eta)$
$\xi = -1;\ \eta = -1$	$\frac{25}{4}$
$\xi = 1;\ \eta = -1$	$\frac{15}{2}$
$\xi = 1;\ \eta = 1$	7
$\xi = -1;\ \eta = 1$	$\frac{23}{4}$

Example 9.

The geometry of a linear quadrilateral isoparametric element and the coordinates of the nodes are known, see in Figure 7.9. The element is distorted.

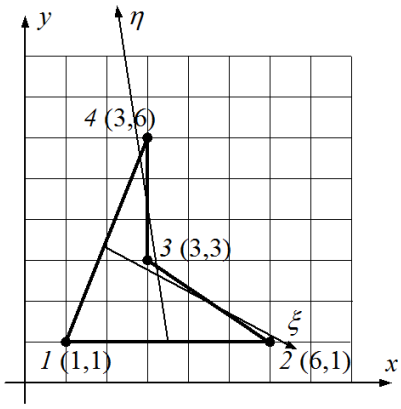


Figure 7.9. The element geometry and the coordinates of the nodes

$x(\xi,\eta) = ?$
$y(\xi,\eta) = ?$
$\mathbf{J} = ?$
$J = ?$

$x(\xi,\eta) = \sum_{i=1}^{4} N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4 =$ $= \frac{1}{4}(1-\xi)(1-\eta)\cdot 1 + \frac{1}{4}(1+\xi)(1-\eta)\cdot 6 +$ $+\frac{1}{4}(1+\xi)(1+\eta)\cdot 3 + \frac{1}{4}(1-\xi)(1+\eta)\cdot 3 = \frac{13}{4} + \frac{5}{4}\xi - \frac{1}{4}\eta - \frac{5}{4}\xi\eta$
$y(\xi,\eta) = \sum_{i=1}^{4} N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4 =$ $= \frac{1}{4}(1-\xi)(1-\eta)\cdot 1 + \frac{1}{4}(1+\xi)(1-\eta)\cdot 1 +$ $+\frac{1}{4}(1+\xi)(1+\eta)\cdot 3 + \frac{1}{4}(1-\xi)(1+\eta)\cdot 6 = \frac{11}{4} - \frac{3}{4}\xi + \frac{7}{4}\eta - \frac{3}{4}\xi\eta$

The derivatives of $x(\xi,\eta)$ and $y(\xi,\eta)$ according to ξ and η have to be produced to get the Jacobian matrix,

$\frac{\partial x}{\partial \xi} = \frac{5}{4} - \frac{5}{4}\eta$	$\frac{\partial y}{\partial \xi} = -\frac{3}{4} - \frac{3}{4}\eta$
$\frac{\partial x}{\partial \eta} = -\frac{1}{4} - \frac{5}{4}\xi$	$\frac{\partial y}{\partial \eta} = \frac{7}{4} - \frac{3}{4}\xi$

Now the Jacobian matrix can be constructed,

$$\mathbf{J} = \begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix} = \begin{bmatrix} \frac{5}{4} - \frac{5}{4}\eta & -\frac{3}{4} - \frac{3}{4}\eta \\ -\frac{1}{4} - \frac{5}{4}\xi & \frac{7}{4} - \frac{3}{4}\xi \end{bmatrix}$$

$$J(\xi,\eta) = detJ(\xi,\eta) = \frac{\partial x}{\partial \xi}\frac{\partial y}{\partial \eta} - \frac{\partial x}{\partial \eta}\frac{\partial y}{\partial \xi} = 2 - \frac{15}{8}\xi - \frac{19}{8}\eta$$

The condition for $J(\xi,\eta)$ has to be controlled

	$J(\xi,\eta)$
$\xi = -1;\ \eta = -1$	$\frac{25}{4}$
$\xi = 1;\ \eta = -1$	$\frac{5}{2}$
$\xi = 1;\ \eta = 1$	$-\frac{25}{4}$
$\xi = -1;\ \eta = 1$	$\frac{3}{2}$

7.2.2. Quadratic quadrilateral element

A eight node quadratic quadrilateral isoparametric element can be seen in Figure 7.10. This is the most widely used element for two-dimensional problems due to its high accuracy in analysis and flexibility in modeling complex geometry, such as curved boundaries.

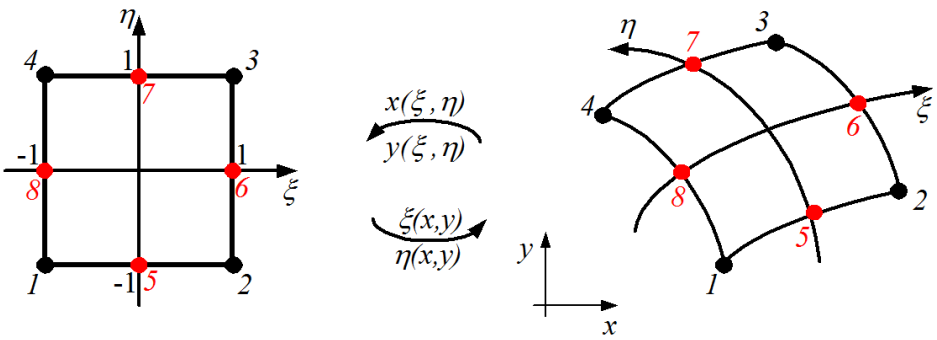


Figure 7.10. Quadratic quadrilateral isoparametric element

There are eight nodes for this element, four corners nodes and four mid-side nodes. Considering a two-dimensional quadratic quadrilateral element, the coordinate interpolations

$$\begin{aligned} x(\xi,\eta) &= \sum_{i=1}^{8} N_i x_i = N_1x_1 + N_2x_2 + N_3x_3 + N_4x_4 + N_5x_5 + N_6x_6 + N_7x_7 + N_8x_8, \\ y(\xi,\eta) &= \sum_{i=1}^{8} N_i y_i = N_1y_1 + N_2y_2 + N_3y_3 + N_4y_4 + N_5y_5 + N_6y_6 + N_7y_7 + N_8y_8. \end{aligned} \tag{7.27}$$

In local coordinate system the eight shape functions are

$$
\begin{aligned}
N_1(\xi,\eta) &= \frac{1}{4}(1-\xi)(1-\eta)(-\xi-\eta-1),\\
N_2(\xi,\eta) &= \frac{1}{4}(1+\xi)(1-\eta)(\xi-\eta-1),\\
N_3(\xi,\eta) &= \frac{1}{4}(1+\xi)(1+\eta)(\xi+\eta-1),\\
N_4(\xi,\eta) &= \frac{1}{4}(1-\xi)(1+\eta)(-\xi+\eta-1),\\
N_5(\xi,\eta) &= \frac{1}{2}(1-\xi^2)(1-\eta),\\
N_6(\xi,\eta) &= \frac{1}{2}(1+\xi)(1-\eta^2),\\
N_7(\xi,\eta) &= \frac{1}{2}(1-\xi^2)(1+\eta),\\
N_8(\xi,\eta) &= \frac{1}{2}(1-\xi)(1-\eta^2).
\end{aligned}
\tag{7.28}
$$

	$\xi=-1;\ \eta=-1$	$\xi=1;\ \eta=-1$	$\xi=1;\ \eta=1$	$\xi=-1;\ \eta=1$
Node 1	$N_1(\xi,\eta)=1$	$N_1(\xi,\eta)=0$	$N_1(\xi,\eta)=0$	$N_1(\xi,\eta)=0$
Node 2	$N_2(\xi,\eta)=0$	$N_2(\xi,\eta)=1$	$N_2(\xi,\eta)=0$	$N_2(\xi,\eta)=0$
Node 3	$N_3(\xi,\eta)=0$	$N_3(\xi,\eta)=0$	$N_3(\xi,\eta)=1$	$N_3(\xi,\eta)=0$
Node 4	$N_4(\xi,\eta)=0$	$N_4(\xi,\eta)=0$	$N_4(\xi,\eta)=0$	$N_4(\xi,\eta)=1$
Node 5	$N_5(\xi,\eta)=0$	$N_5(\xi,\eta)=0$	$N_5(\xi,\eta)=0$	$N_5(\xi,\eta)=0$
Node 6	$N_6(\xi,\eta)=0$	$N_6(\xi,\eta)=0$	$N_6(\xi,\eta)=0$	$N_6(\xi,\eta)=0$
Node 7	$N_7(\xi,\eta)=0$	$N_7(\xi,\eta)=0$	$N_7(\xi,\eta)=0$	$N_7(\xi,\eta)=0$
Node 8	$N_8(\xi,\eta)=0$	$N_8(\xi,\eta)=0$	$N_8(\xi,\eta)=0$	$N_8(\xi,\eta)=0$

The approximation of the displacement field on the element assuming that $\xi=\xi(x,y)$ and $\eta=\eta(x,y)$

$$
\begin{aligned}
u^e(x,y) &= \sum_{i=1}^{8} N_i(\xi,\eta)u_i^e = N_1(\xi,\eta)u_1^e + N_2(\xi,\eta)u_2^e + N_3(\xi,\eta)u_3^e +\\
&+N_4(\xi,\eta)u_4^e + N_5(\xi,\eta)u_5^e + N_6(\xi,\eta)u_6^e + N_7(\xi,\eta)u_7^e + N_8(\xi,\eta)u_8^e,\\
v^e(x,y) &= \sum_{i=1}^{8} N_i(\xi,\eta)v_i^e = N_1(\xi,\eta)v_1^e + N_2(\xi,\eta)v_2^e + N_3(\xi,\eta)v_3^e +\\
&+N_4(\xi,\eta)v_4^e + N_5(\xi,\eta)v_5^e + N_6(\xi,\eta)v_6^e + N_7(\xi,\eta)v_7^e + N_8(\xi,\eta)v_8^e
\end{aligned}
\tag{7.29}
$$

which are the quadratic functions on the element. Strains and stresses of a quadrilateral element are quadratic functions.

7.2.3. Lagrangian quadrilateral element

A nine node Lagrangian quadrilateral isoparametric element can be seen in Figure 7.11. This is also a widely used element. The difference compared to the quadratic quadrilateral element is that it contains a so called bubble function.

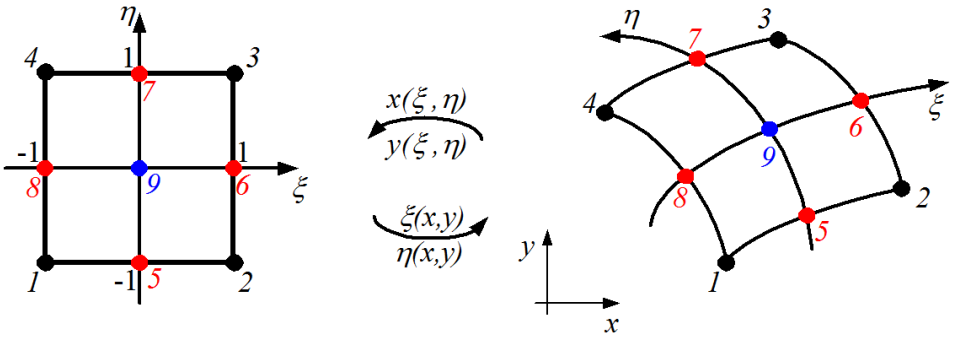


Figure 7.11. The Lagrangian quadrilateral element

Considering a two-dimensional Lagrangian quadrilateral element, the coordinate interpolations

$$
\begin{gathered}
x(\xi,\eta) = \sum_{i=1}^{9} N_i x_i = \\
= N_1x_1 + N_2x_2 + N_3x_3 + N_4x_4 + N_5x_5 + N_6x_6 + N_7x_7 + N_8x_8 + N_9x_9, \\
y(\xi,\eta) = \sum_{i=1}^{9} N_i y_i = \\
N_1y_1 + N_2y_2 + N_3y_3 + N_4y_4 + N_5y_5 + N_6y_6 + N_7y_7 + N_8y_8 + N_9y_9.
\end{gathered} \tag{7.30}
$$

In local coordinate system the nine shape functions are

$$
\begin{aligned}
N_1(\xi,\eta) &= \frac{1}{4}(1-\xi)(1-\eta)(-\xi-\eta-1), \\
N_2(\xi,\eta) &= \frac{1}{4}(1+\xi)(1-\eta)(\xi-\eta-1), \\
N_3(\xi,\eta) &= \frac{1}{4}(1+\xi)(1+\eta)(\xi+\eta-1), \\
N_4(\xi,\eta) &= \frac{1}{4}(1-\xi)(1+\eta)(-\xi+\eta-1), \\
N_5(\xi,\eta) &= \frac{1}{2}(1-\xi^2)(1-\eta), \\
N_6(\xi,\eta) &= \frac{1}{2}(1+\xi)(1-\eta^2), \\
N_7(\xi,\eta) &= \frac{1}{2}(1-\xi^2)(1+\eta), \\
N_8(\xi,\eta) &= \frac{1}{2}(1-\xi)(1-\eta^2), \\
N_9(\xi,\eta) &= \frac{1}{2}(1-\xi^2)(1-\eta^2).
\end{aligned} \tag{7.31}
$$

	$\xi=-1;\ \eta=-1$	$\xi=1;\ \eta=-1$	$\xi=1;\ \eta=1$	$\xi=-1;\ \eta=1$
Node 1	$N_1(\xi,\eta)=1$	$N_1(\xi,\eta)=0$	$N_1(\xi,\eta)=0$	$N_1(\xi,\eta)=0$
Node 2	$N_2(\xi,\eta)=0$	$N_2(\xi,\eta)=1$	$N_2(\xi,\eta)=0$	$N_2(\xi,\eta)=0$
Node 3	$N_3(\xi,\eta)=0$	$N_3(\xi,\eta)=0$	$N_3(\xi,\eta)=1$	$N_3(\xi,\eta)=0$
Node 4	$N_4(\xi,\eta)=0$	$N_4(\xi,\eta)=0$	$N_4(\xi,\eta)=0$	$N_4(\xi,\eta)=1$
Node 5	$N_5(\xi,\eta)=0$	$N_5(\xi,\eta)=0$	$N_5(\xi,\eta)=0$	$N_5(\xi,\eta)=0$

Node 6	$N_6(\xi,\eta)=0$	$N_6(\xi,\eta)=0$	$N_6(\xi,\eta)=0$	$N_6(\xi,\eta)=0$
Node 7	$N_7(\xi,\eta)=0$	$N_7(\xi,\eta)=0$	$N_7(\xi,\eta)=0$	$N_7(\xi,\eta)=0$
Node 8	$N_8(\xi,\eta)=0$	$N_8(\xi,\eta)=0$	$N_8(\xi,\eta)=0$	$N_8(\xi,\eta)=0$
Node 9	$N_9(\xi,\eta)=0$	$N_9(\xi,\eta)=0$	$N_9(\xi,\eta)=0$	$N_9(\xi,\eta)=0$

The approximation of the displacement field of the element assuming that $\xi=\xi(x,y)$ and $\eta=\eta(x,y)$

$$\begin{aligned} u^e(x,y)&=\sum_{i=1}^{9} N_i\,(\xi,\eta)u_i^e = N_1(\xi,\eta)u_1^e+N_2(\xi,\eta)u_2^e+N_3(\xi,\eta)u_3^e+\\ &+N_4(\xi,\eta)u_4^e+N_5(\xi,\eta)u_5^e+N_6(\xi,\eta)u_6^e+N_7(\xi,\eta)u_7^e+N_8(\xi,\eta)u_8^e+N_9(\xi,\eta)u_9^e,\\ v^e(x,y)&=\sum_{i=1}^{9} N_i\,(\xi,\eta)v_i^e = N_1(\xi,\eta)v_1^e+N_2(\xi,\eta)v_2^e+N_3(\xi,\eta)v_3^e+\\ &+N_4(\xi,\eta)v_4^e+N_5(\xi,\eta)v_5^e+N_6(\xi,\eta)v_6^e+N_7(\xi,\eta)v_7^e+N_8(\xi,\eta)v_8^e+N_9(\xi,\eta)v_9^e. \end{aligned} \tag{7.32}$$

7.2.4. Linear triangular element

The geometric description of a linear triangular element and its mapping can be seen in Figure 7.12. The natural way for generating triangular element is by assigning the same global node to two nodes of the linear quadrilateral element.

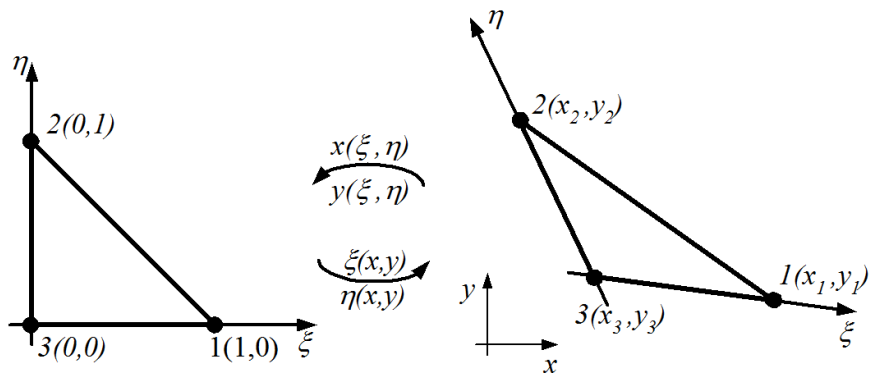


Figure 7.12. Linear triangular isoparametric element

Considering a two-dimensional linear triangular element, the coordinate interpolations

$$\begin{aligned} x(\xi,\eta)&=\sum_{i=1}^{3} N_i x_i = N_1x_1+N_2x_2+N_3x_3,\\ y(\xi,\eta)&=\sum_{i=1}^{3} N_i y_i = N_1y_1+N_2y_2+N_3y_3, \end{aligned} \tag{7.33}$$

where the shape functions are

$$\begin{aligned} N_1(\xi,\eta)&=\xi,\\ N_2(\xi,\eta)&=\eta,\\ N_3(\xi,\eta)&=1-\xi-\eta. \end{aligned} \tag{7.34}$$

The properties of the shape functions:

1.	$\sum_{i=1}^{3} N_i\,(\xi,\eta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j) = \begin{cases} 1 & if\ \ i = j \\ 0 & if\ i \ne j \end{cases}$ where ξ_j, η_j is the coordinate of the jth node $j = 1,2,3$

The approximation of the displacement field of the element assuming that $\xi = \xi(x,y)$ and $\eta = \eta(x,y)$

$$\begin{aligned} u^e(x,y) &= \sum_{i=1}^{3} N_i\,(\xi,\eta)u_i^e = N_1(\xi,\eta)u_1^e + N_2(\xi,\eta)u_2^e + N_3(\xi,\eta)u_3^e, \\ v^e(x,y) &= \sum_{i=1}^{3} N_i\,(\xi,\eta)v_i^e = N_1(\xi,\eta)v_1^e + N_2(\xi,\eta)v_2^e + N_3(\xi,\eta)v_3^e, \end{aligned} \tag{7.35}$$

where u_i^e and v_i^e are the displacement coordinates of the ith node of the element e. The purpose of the investigation is to determine these nodal values. In matrix form

$$\mathbf{u}^e = \underbrace{\begin{bmatrix} u \\ v \end{bmatrix}^e}_{(2x1)} = \underbrace{\begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}}_{(2x6)} \underbrace{\begin{bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{bmatrix}^e}_{(6x1)} = \mathbf{N}\mathbf{q}^e, \tag{7.36}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element. Using Eq. 6.11 the strain vector of the element can be obtained

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}\mathbf{q}^e = \mathbf{B}\mathbf{q}^e, \tag{7.37}$$

where the strain-displacement matrix

$$\begin{aligned} \mathbf{B} = \mathbf{D}^e\mathbf{N} &= \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} \end{bmatrix}^e}_{(3x2)} \underbrace{\begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix}}_{(2x6)} = \\ &= \begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & \frac{\partial N_2}{\partial x} & 0 & \frac{\partial N_3}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \frac{\partial N_2}{\partial y} & 0 & \frac{\partial N_3}{\partial y} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & \frac{\partial N_2}{\partial y} & \frac{\partial N_2}{\partial x} & \frac{\partial N_3}{\partial y} & \frac{\partial N_3}{\partial x} \end{bmatrix} \end{aligned} \tag{7.38}$$

The derivation of the shape function can be expressed by

$$\frac{\partial N_i(\xi,\eta)}{\partial x} = \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial x}$$
$$\frac{\partial N_i(\xi,\eta)}{\partial y} = \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial y} \tag{7.39}$$

The Eq. 7.39 can be written in matrix form

$$\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix}}_{\mathbf{J}^{-1}} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix} = \mathbf{J}^{-1} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix}, \tag{7.40}$$

where $\mathbf{J}$ is the Jacobian. The inverse of the Jacobian has to be known in finite element computations. The Jacobian differential operator matrix is known in possession of the functions $x = x(\xi,\eta)$ and $y = y(\xi,\eta)$, thus

$$\begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \end{bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} \end{bmatrix}}_{\mathbf{J}} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix} = \mathbf{J} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \end{bmatrix}, \tag{7.41}$$

The Jacobian can be formulated at a known mesh and geometry. The $\mathbf{J}^{-1}$ can be derived by inverting $\mathbf{J}$,

$$\mathbf{J}^{-1} = \begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} \end{bmatrix} = \frac{1}{det\mathbf{J}} \begin{bmatrix} \frac{\partial y}{\partial \eta} & -\frac{\partial y}{\partial \xi} \\ -\frac{\partial x}{\partial \eta} & \frac{\partial x}{\partial \xi} \end{bmatrix}, \tag{7.42}$$

where $det\mathbf{J} = J > 0$. The cases $J < 0$ and $J = 0$ are not allowed in finite element calculations, they make numerical error.

7.2.5. Quadratic triangular element

The geometric description of a quadratic triangular element and its mapping can be seen in Figure 7.13.

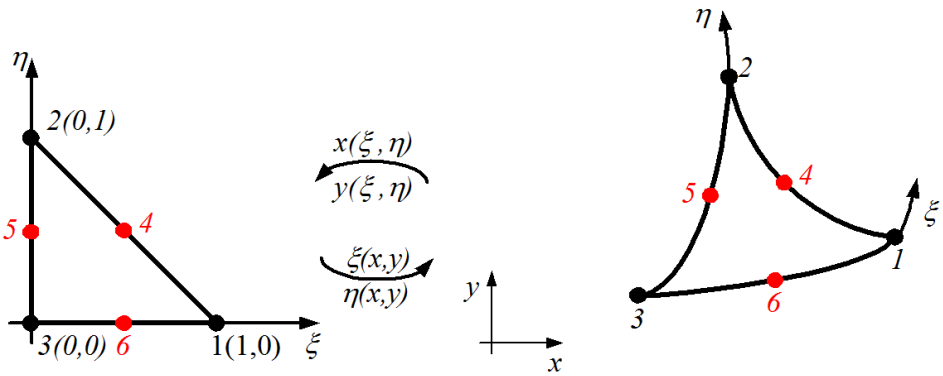


Figure 7.13. Quadratic triangular isoparametric element

Considering a two-dimensional quadratic triangular element, the coordinate interpolations

$$\begin{aligned} x(\xi,\eta) &= \sum_{i=1}^{6} N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4 + N_5 x_5 + N_6 x_6, \\ y(\xi,\eta) &= \sum_{i=1}^{6} N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4 + N_5 y_5 + N_6 y_6, \end{aligned} \tag{7.43}$$

where the shape functions are

$$\begin{gathered} N_1(\xi,\eta) = \xi(2\xi - 1), \\ N_2(\xi,\eta) = \eta(2\eta - 1), \\ N_3(\xi,\eta) = (1 - \xi - \eta)(1 - 2\xi - 2\eta), \\ N_4(\xi,\eta) = 4\xi\eta, \\ N_5(\xi,\eta) = \eta(1 - \xi - \eta), \\ N_6(\xi,\eta) = \xi(1 - \xi - \eta). \end{gathered} \tag{7.44}$$

The properties of the shape functions:

1.	$\sum_{i=1}^{6} N_i\,(\xi,\eta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j) = \begin{cases} 1 & if\ \ i = j \\ 0 & if\ i \ne j \end{cases}$ where ξ_j, η_j is the coordinate of the jth node $j = 1, \dots, 6$

The approximation of the displacement field of the element assuming that $\xi = \xi(x,y)$ and $\eta = \eta(x,y)$

$$\begin{gathered} u^e(x,y) = \sum_{i=1}^{3} N_i\,(\xi,\eta) u_i^e = \\ = N_1(\xi,\eta)u_1^e + N_2(\xi,\eta)u_2^e + N_3(\xi,\eta)u_3^e + N_4(\xi,\eta)u_4^e + N_5(\xi,\eta)u_5^e + N_6(\xi,\eta)u_6^e, \\ v^e(x,y) = \sum_{i=1}^{3} N_i\,(\xi,\eta) v_i^e = \\ = N_1(\xi,\eta)v_1^e + N_2(\xi,\eta)v_2^e + N_3(\xi,\eta)v_3^e + N_4(\xi,\eta)v_4^e + N_5(\xi,\eta)v_5^e + N_6(\xi,\eta)v_6^e, \end{gathered} \tag{7.45}$$

where u_i^e and v_i^e are the displacement coordinates of the ith node of the element e.

7.3. Three-dimensional isoparametric mapping

7.3.1. Eight node brick element

The geometric description of an eight node brick element and its mapping can be seen in Figure 7.14. The brick element is also known as hexahedron. A hexahedron is topologically equivalent to a cube. It has eight corners. These kinds of elements are widely used in finite element simulations of three-dimensional solids. These elements have 24 degrees of freedom

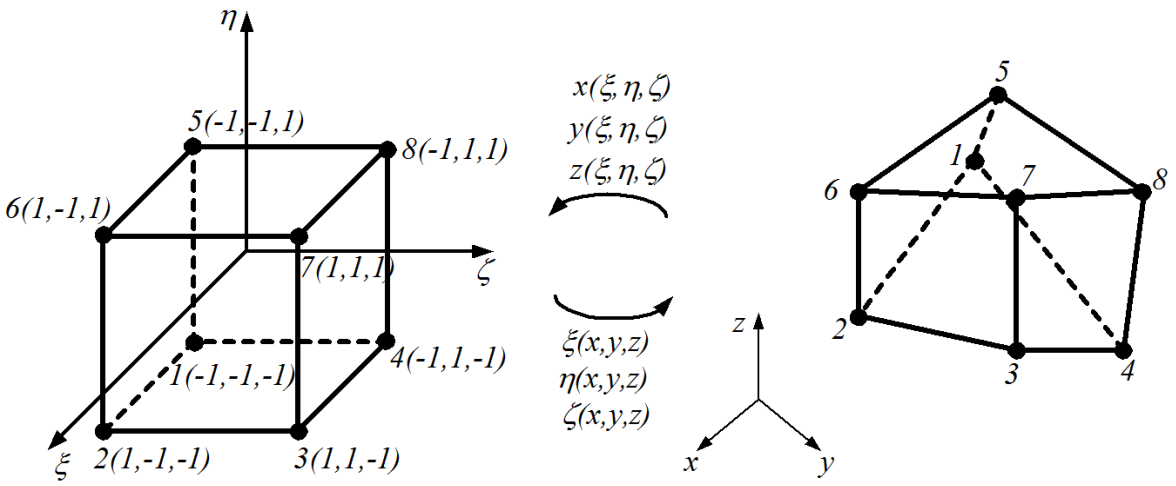


Figure 7.14. Eight node brick element

Considering a three-dimensional eight node brick element, the coordinate interpolations

$$
\begin{gathered}
x(\xi,\eta,\zeta) = \sum_{i=1}^{8} N_i x_i = \\
= N_1x_1 + N_2x_2 + N_3x_3 + N_4x_4 + N_5x_5 + N_6x_6 + N_7x_7 + N_8x_8, \\
y(\xi,\eta,\zeta) = \sum_{i=1}^{8} N_i y_i = \\
= N_1y_1 + N_2y_2 + N_3y_3 + N_4y_4 + N_5y_5 + N_6y_6 + N_7y_7 + N_8y_8, \\
z(\xi,\eta,\zeta) = \sum_{i=1}^{8} N_i z_i = \\
= N_1z_1 + N_2z_2 + N_3z_3 + N_4z_4 + N_5z_5 + N_6z_6 + N_7z_7 + N_8z_8,
\end{gathered}
\tag{7.46}
$$

where the shape functions are

$$
\begin{aligned}
N_1(\xi,\eta,\zeta) &= \frac{1}{8}(1-\xi)(1-\eta)(1-\zeta), \\
N_2(\xi,\eta,\zeta) &= \frac{1}{8}(1+\xi)(1-\eta)(1-\zeta), \\
N_3(\xi,\eta,\zeta) &= \frac{1}{8}(1+\xi)(1+\eta)(1-\zeta), \\
N_4(\xi,\eta,\zeta) &= \frac{1}{8}(1-\xi)(1+\eta)(1-\zeta), \\
N_5(\xi,\eta,\zeta) &= \frac{1}{8}(1-\xi)(1-\eta)(1+\zeta), \\
N_6(\xi,\eta,\zeta) &= \frac{1}{8}(1+\xi)(1-\eta)(1+\zeta), \\
N_7(\xi,\eta,\zeta) &= \frac{1}{8}(1+\xi)(1+\eta)(1+\zeta), \\
N_8(\xi,\eta,\zeta) &= \frac{1}{8}(1-\xi)(1+\eta)(1+\zeta).
\end{aligned}
\tag{7.47}
$$

The properties of the shape functions:

1.	$\sum_{i=1}^{8} N_i\,(\xi,\eta,\zeta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\\ \zeta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j,\zeta_j) = \begin{cases}1 & if\ \ i = j\\ 0 & if\ i \ne j\end{cases}$ where ξ_j,η_j,ζ_j is the coordinate of the jth node $j = 1,\dots,8$

The approximation of the displacement field of the element assuming that $\xi = \xi(x,y,z)$, $\eta = \eta(x,y,z)$ and $\zeta = \zeta(x,y,z)$

$$\begin{aligned}
u^e(x,y,z) &= \sum_{i=1}^{8} N_i\,(\xi,\eta,\zeta)u_i^e = N_1(\xi,\eta,\zeta)u_1^e + N_2(\xi,\eta,\zeta)u_2^e + N_3(\xi,\eta,\zeta)u_3^e + \\
&+N_4(\xi,\eta,\zeta)u_4^e + N_5(\xi,\eta,\zeta)u_5^e + N_6(\xi,\eta,\zeta)u_6^e + N_7(\xi,\eta,\zeta)u_7^e + N_8(\xi,\eta,\zeta)u_8^e,\\
v^e(x,y,z) &= \sum_{i=1}^{8} N_i\,(\xi,\eta,\zeta)v_i^e = N_1(\xi,\eta,\zeta)v_1^e + N_2(\xi,\eta,\zeta)v_2^e + N_3(\xi,\eta,\zeta)v_3^e + \\
&+N_4(\xi,\eta,\zeta)v_4^e + N_5(\xi,\eta,\zeta)v_5^e + N_6(\xi,\eta,\zeta)v_6^e + N_7(\xi,\eta,\zeta)v_7^e + N_8(\xi,\eta,\zeta)v_8^e,\\
w^e(x,y,z) &= \sum_{i=1}^{8} N_i\,(\xi,\eta,\zeta)w_i^e = N_1(\xi,\eta,\zeta)w_1^e + N_2(\xi,\eta,\zeta)w_2^e + N_3(\xi,\eta,\zeta)w_3^e + \\
&+N_4(\xi,\eta,\zeta)w_4^e + N_5(\xi,\eta,\zeta)w_5^e + N_6(\xi,\eta,\zeta)w_6^e + N_7(\xi,\eta,\zeta)w_7^e + N_8(\xi,\eta,\zeta)w_8^e,
\end{aligned} \tag{7.48}$$

where u_i^e, v_i^e and w_i^e are the displacement coordinates of the ith node of the element e. The purpose of the investigation is to determine these nodal values. In matrix form

$$\mathbf{u}^e = \underbrace{\begin{bmatrix} u\\ v\\ w\end{bmatrix}^e}_{(3x1)} = \underbrace{\begin{bmatrix} N_1 & 0 & 0 & \cdots & N_8 & 0 & 0\\ 0 & N_1 & 0 & \cdots & 0 & N_8 & 0\\ 0 & 0 & N_1 & \cdots & 0 & 0 & N_8\end{bmatrix}}_{(3x24)} \underbrace{\begin{bmatrix} u_1\\ v_1\\ w_1\\ \vdots\\ u_8\\ v_8\\ w_8\end{bmatrix}^e}_{(24x1)} = \mathbf{N}\mathbf{q}^e, \tag{7.49}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element. Using Eq. 6.11 the strain vector of the element can be obtained

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}\mathbf{q}^e = \mathbf{B}\mathbf{q}^e, \tag{7.50}$$

where the strain-displacement matrix using Eq. 4.10

$$\mathbf{B} = \mathbf{D}^e\mathbf{N} = \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix}^e}_{(6x3)} \underbrace{\begin{bmatrix} N_1 & 0 & 0 & \cdots & N_8 & 0 & 0 \\ 0 & N_1 & 0 & \cdots & 0 & N_8 & 0 \\ 0 & 0 & N_1 & \cdots & 0 & 0 & N_8 \end{bmatrix}}_{(3x24)} =$$

$$= \underbrace{\begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & 0 & \cdots & \frac{\partial N_8}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \cdots & 0 & \frac{\partial N_8}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_1}{\partial z} & \cdots & 0 & 0 & \frac{\partial N_8}{\partial z} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & 0 & \cdots & \frac{\partial N_8}{\partial y} & \frac{\partial N_8}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial z} & \frac{\partial N_1}{\partial y} & \cdots & 0 & \frac{\partial N_8}{\partial z} & \frac{\partial N_8}{\partial y} \\ \frac{\partial N_1}{\partial z} & 0 & \frac{\partial N_1}{\partial x} & \cdots & \frac{\partial N_8}{\partial z} & 0 & \frac{\partial N_8}{\partial x} \end{bmatrix}}_{(6x24)}. \tag{7.51}$$

The derivation of the shape function can be expressed by

$$\begin{aligned} \frac{\partial N_i(\xi,\eta,\zeta)}{\partial x} &= \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial x} + \frac{\partial N_i}{\partial \zeta}\frac{\partial \zeta}{\partial x}, \\ \frac{\partial N_i(\xi,\eta,\zeta)}{\partial y} &= \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial y} + \frac{\partial N_i}{\partial \zeta}\frac{\partial \zeta}{\partial y}, \\ \frac{\partial N_i(\xi,\eta,\zeta)}{\partial z} &= \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial z} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial z} + \frac{\partial N_i}{\partial \zeta}\frac{\partial \zeta}{\partial z}. \end{aligned} \tag{7.52}$$

The Eq. 7.52 can be written in matrix form

$$\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} & \frac{\partial \zeta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} & \frac{\partial \zeta}{\partial y} \\ \frac{\partial \xi}{\partial z} & \frac{\partial \eta}{\partial z} & \frac{\partial \zeta}{\partial z} \end{bmatrix}}_{\mathbf{J}^{-1}} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{bmatrix} = \mathbf{J}^{-1} \begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{bmatrix}, \tag{7.53}$$

where $\mathbf{J}$ is the Jacobian. The inverse of the Jacobian has to be known in finite element computations. The Jacobian differential operator matrix is known in possession of the functions $x = x(\xi,\eta,\zeta)$, $y = y(\xi,\eta,\zeta)$ and $z = z(\xi,\eta,\zeta)$, thus

$$\begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{bmatrix} = \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix}}_{\mathbf{J}} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix} = \mathbf{J} \begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix}, \tag{7.54}$$

The Jacobian can be formulated at a known mesh and geometry.

7.3.2. 20-node quadratic brick element

The geometric description of a 20-node brick element and its mapping can be seen in Figure 7.15. It has eight corner nodes and 12 side nodes which are located at the midpoints of the sides. These elements have 60 degrees of freedom.

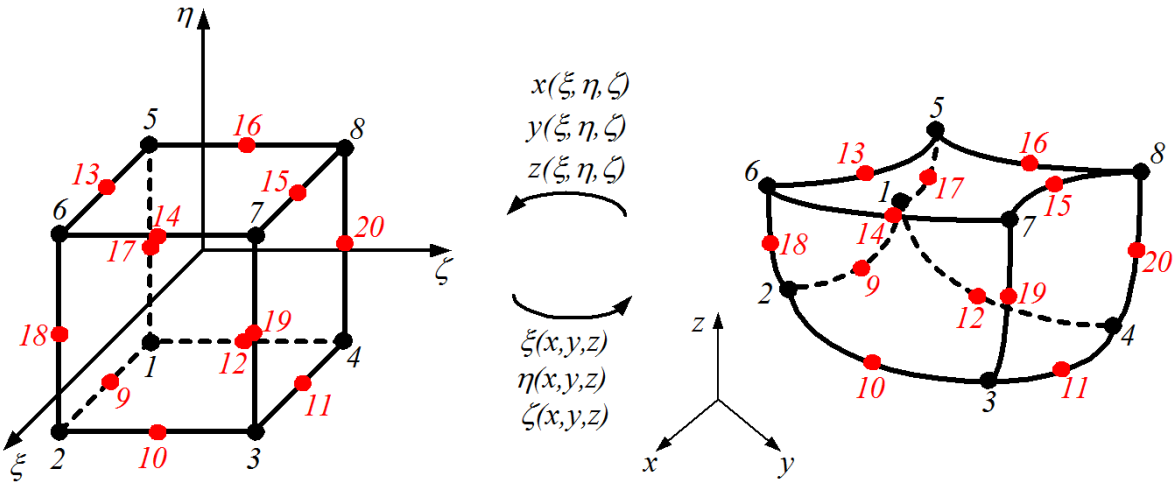


Figure 7.15. 20-node brick element

Considering a three-dimensional 20-node brick element, the coordinate interpolations

$$\begin{aligned} x(\xi,\eta,\zeta) &= \sum_{i=1}^{20} N_i x_i\,, \\ y(\xi,\eta,\zeta) &= \sum_{i=1}^{20} N_i y_i\,, \\ z(\xi,\eta,\zeta) &= \sum_{i=1}^{20} N_i z_i\,. \end{aligned} \tag{7.55}$$

where the shape functions can be grouped. For the corner nodes $(i = 1,2,3,\dots,8)$

$$N_i(\xi,\eta,\zeta) = \frac{1}{8}(1+\xi\xi_i)(1+\eta\eta_i)(1+\zeta\zeta_i)(\xi\xi_i+\eta\eta_i+\zeta\zeta_i-2). \tag{7.56}$$

For the midside nodes ($i = 9,11,15,13$)

$$N_i(\xi,\eta,\zeta) = \frac{1}{4}(1-\xi^2)(1+\eta\eta_i)(1+\zeta\zeta_i). \tag{7.57}$$

For the midside nodes ($i = 10,12,16,14$)

$$N_i(\xi,\eta,\zeta) = \frac{1}{4}(1-\eta^2)(1+\xi\xi_i)(1+\zeta\zeta_i). \tag{7.58}$$

For the midside nodes ($i = 17,18,19,20$)

$$N_i(\xi,\eta,\zeta) = \frac{1}{4}(1-\zeta^2)(1+\xi\xi_i)(1+\eta\eta_i). \tag{7.59}$$

The properties of the shape functions:

1.	$\sum_{i=1}^{20} N_i\,(\xi,\eta,\zeta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\\ \zeta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j,\zeta_j) = \begin{cases}1 & if\ \ i = j\\ 0 & if\ i \ne j\end{cases}$ where ξ_j,η_j,ζ_j is the coordinate of the jth node $j = 1,\dots,20$

The approximation of the displacement field of the element assuming that $\xi = \xi(x,y,z)$, $\eta = \eta(x,y,z)$ and $\zeta = \zeta(x,y,z)$

$$\begin{aligned} u^e(x,y,z) &= \sum_{i=1}^{20} N_i\,(\xi,\eta,\zeta)u_i^e,\\ v^e(x,y,z) &= \sum_{i=1}^{20} N_i\,(\xi,\eta,\zeta)v_i^e,\\ w^e(x,y,z) &= \sum_{i=1}^{20} N_i\,(\xi,\eta,\zeta)w_i^e, \end{aligned} \tag{7.60}$$

where u_i^e, v_i^e and w_i^e are the displacement coordinates of the ith node of the element e.

7.3.3. 27-node triquadratic brick element

A 27-node triquadratic brick element can also be constructed by adding 7 plus nodes. 6 nodes are on each face center and 1 interior node (bubble function) at the brick center point. These elements have 81 degrees of freedom.

7.3.4. Four node tetrahedron element

The geometric description of a four node tetrahedron element and its mapping can be seen in Figure 7.16. It has four corners. These elements have 12 degrees of freedom.

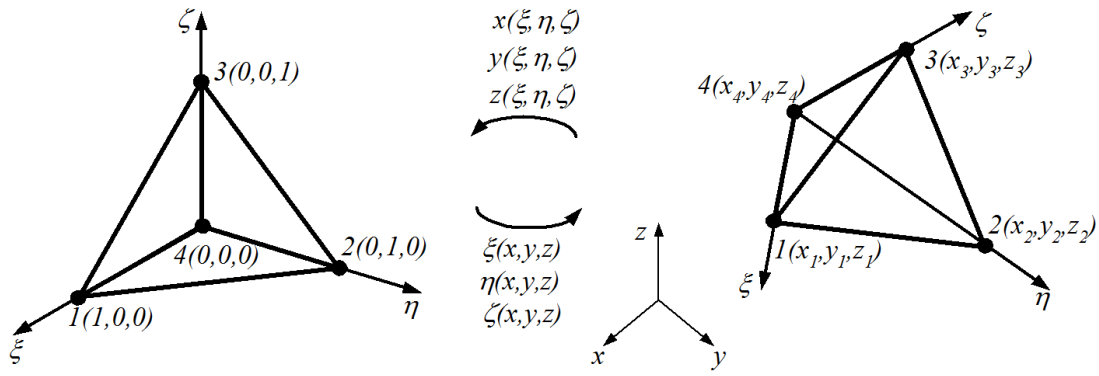


Figure 7.16. Four node tetrahedron element

Considering a three-dimensional four node tetrahedron element, the coordinate interpolations

$$
\begin{aligned}
x(\xi,\eta,\zeta) &= \sum_{i=1}^{4} N_i x_i = N_1 x_1 + N_2 x_2 + N_3 x_3 + N_4 x_4, \\
y(\xi,\eta,\zeta) &= \sum_{i=1}^{4} N_i y_i = N_1 y_1 + N_2 y_2 + N_3 y_3 + N_4 y_4, \\
z(\xi,\eta,\zeta) &= \sum_{i=1}^{4} N_i z_i = N_1 z_1 + N_2 z_2 + N_3 z_3 + N_4 z_4,
\end{aligned}
\tag{7.61}
$$

where the shape functions are

$$
\begin{aligned}
N_1(\xi,\eta,\zeta) &= \xi, \\
N_2(\xi,\eta,\zeta) &= \eta, \\
N_3(\xi,\eta,\zeta) &= \zeta, \\
N_4(\xi,\eta,\zeta) &= 1 - \xi - \eta - \zeta.
\end{aligned}
\tag{7.62}
$$

The properties of the shape functions:

1.	$\sum_{i=1}^{4} N_i\,(\xi,\eta,\zeta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\\ \zeta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j,\zeta_j) = \begin{cases} 1 & if\ \ i = j \\ 0 & if\ i \ne j \end{cases}$ where ξ_j,η_j,ζ_j is the coordinate of the jth node $j = 1,2,3,4$

The approximation of the displacement field of the element assuming that $\xi = \xi(x,y,z)$, $\eta = \eta(x,y,z)$ and $\zeta = \zeta(x,y,z)$

$$
\begin{aligned}
u^e(x,y,z) &= \sum_{i=1}^{4} N_i\,(\xi,\eta,\zeta) u_i^e = \\
&= N_1(\xi,\eta,\zeta)u_1^e + N_2(\xi,\eta,\zeta)u_2^e + N_3(\xi,\eta,\zeta)u_3^e + N_4(\xi,\eta,\zeta)u_4^e, \\
v^e(x,y,z) &= \sum_{i=1}^{4} N_i\,(\xi,\eta,\zeta) v_i^e = \\
&= N_1(\xi,\eta,\zeta)v_1^e + N_2(\xi,\eta,\zeta)v_2^e + N_3(\xi,\eta,\zeta)v_3^e + N_4(\xi,\eta,\zeta)v_4^e, \\
w^e(x,y,z) &= \sum_{i=1}^{4} N_i\,(\xi,\eta,\zeta) w_i^e = \\
&N_1(\xi,\eta,\zeta)w_1^e + N_2(\xi,\eta,\zeta)w_2^e + N_3(\xi,\eta,\zeta)w_3^e + N_4(\xi,\eta,\zeta)w_4^e,
\end{aligned}
\tag{7.63}
$$

where u_i^e, v_i^e and w_i^e are the displacement coordinates of the ith node of the element e. The purpose of the investigation is to determine these nodal values. In matrix form

$$\mathbf{u}^e = \underbrace{\begin{bmatrix} u \\ v \\ w \end{bmatrix}^e}_{(3x1)} = \underbrace{\begin{bmatrix} N_1 & 0 & 0 & \cdots & N_4 & 0 & 0 \\ 0 & N_1 & 0 & \cdots & 0 & N_4 & 0 \\ 0 & 0 & N_1 & \cdots & 0 & 0 & N_4 \end{bmatrix}}_{(3x12)} \underbrace{\begin{bmatrix} u_1 \\ v_1 \\ w_1 \\ \vdots \\ u_4 \\ v_4 \\ w_4 \end{bmatrix}^e}_{(12x1)} = \mathbf{N}\mathbf{q}^e, \tag{7.64}$$

where $\mathbf{q}^e$ is the nodal displacement vector of the element. Using Eq. 6.11 the strain vector of the element can be obtained

$$\boldsymbol{\varepsilon}^e = \mathbf{D}^e\mathbf{u}^e = \mathbf{D}^e\mathbf{N}\mathbf{q}^e = \mathbf{B}\mathbf{q}^e, \tag{7.65}$$

where the strain-displacement matrix using Eq. 4.10

$$\mathbf{B} = \mathbf{D}^e\mathbf{N} = \underbrace{\begin{bmatrix} \frac{\partial}{\partial x} & 0 & 0 \\ 0 & \frac{\partial}{\partial y} & 0 \\ 0 & 0 & \frac{\partial}{\partial z} \\ \frac{\partial}{\partial y} & \frac{\partial}{\partial x} & 0 \\ 0 & \frac{\partial}{\partial z} & \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} & 0 & \frac{\partial}{\partial x} \end{bmatrix}^e}_{(6x3)} \underbrace{\begin{bmatrix} N_1 & 0 & 0 & \cdots & N_4 & 0 & 0 \\ 0 & N_1 & 0 & \cdots & 0 & N_4 & 0 \\ 0 & 0 & N_1 & \cdots & 0 & 0 & N_4 \end{bmatrix}}_{(3x12)} =$$

$$= \underbrace{\begin{bmatrix} \frac{\partial N_1}{\partial x} & 0 & 0 & \cdots & \frac{\partial N_4}{\partial x} & 0 & 0 \\ 0 & \frac{\partial N_1}{\partial y} & 0 & \cdots & 0 & \frac{\partial N_4}{\partial y} & 0 \\ 0 & 0 & \frac{\partial N_1}{\partial z} & \cdots & 0 & 0 & \frac{\partial N_4}{\partial z} \\ \frac{\partial N_1}{\partial y} & \frac{\partial N_1}{\partial x} & 0 & \cdots & \frac{\partial N_4}{\partial y} & \frac{\partial N_4}{\partial x} & 0 \\ 0 & \frac{\partial N_1}{\partial z} & \frac{\partial N_1}{\partial y} & \cdots & 0 & \frac{\partial N_4}{\partial z} & \frac{\partial N_4}{\partial y} \\ \frac{\partial N_1}{\partial z} & 0 & \frac{\partial N_1}{\partial x} & \cdots & \frac{\partial N_4}{\partial z} & 0 & \frac{\partial N_4}{\partial x} \end{bmatrix}}_{(6x12)}. \tag{7.66}$$

The derivation of the shape function can be expressed by

$$
\begin{aligned}
\frac{\partial N_i(\xi,\eta,\zeta)}{\partial x} &= \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial x} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial x} + \frac{\partial N_i}{\partial \zeta}\frac{\partial \zeta}{\partial x},\\
\frac{\partial N_i(\xi,\eta,\zeta)}{\partial y} &= \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial y} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial y} + \frac{\partial N_i}{\partial \zeta}\frac{\partial \zeta}{\partial y},\\
\frac{\partial N_i(\xi,\eta,\zeta)}{\partial z} &= \frac{\partial N_i}{\partial \xi}\frac{\partial \xi}{\partial z} + \frac{\partial N_i}{\partial \eta}\frac{\partial \eta}{\partial z} + \frac{\partial N_i}{\partial \zeta}\frac{\partial \zeta}{\partial z}.
\end{aligned}
\tag{7.67}
$$

The Eq. 7.67 can be written in matrix form

$$
\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix}
= \underbrace{\begin{bmatrix} \frac{\partial \xi}{\partial x} & \frac{\partial \eta}{\partial x} & \frac{\partial \zeta}{\partial x} \\ \frac{\partial \xi}{\partial y} & \frac{\partial \eta}{\partial y} & \frac{\partial \zeta}{\partial y} \\ \frac{\partial \xi}{\partial z} & \frac{\partial \eta}{\partial z} & \frac{\partial \zeta}{\partial z} \end{bmatrix}}_{\mathbf{J}^{-1}}
\begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{bmatrix}
= \mathbf{J}^{-1}
\begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{bmatrix},
\tag{7.68}
$$

where $\mathbf{J}$ is the Jacobian. The inverse of the Jacobian has to be known in finite element computations. The Jacobian differential operator matrix is known in possession of the functions $x = x(\xi,\eta,\zeta)$, $y = y(\xi,\eta,\zeta)$ and $z = z(\xi,\eta,\zeta)$, thus

$$
\begin{bmatrix} \frac{\partial}{\partial \xi} \\ \frac{\partial}{\partial \eta} \\ \frac{\partial}{\partial \zeta} \end{bmatrix}
= \underbrace{\begin{bmatrix} \frac{\partial x}{\partial \xi} & \frac{\partial y}{\partial \xi} & \frac{\partial z}{\partial \xi} \\ \frac{\partial x}{\partial \eta} & \frac{\partial y}{\partial \eta} & \frac{\partial z}{\partial \eta} \\ \frac{\partial x}{\partial \zeta} & \frac{\partial y}{\partial \zeta} & \frac{\partial z}{\partial \zeta} \end{bmatrix}}_{\mathbf{J}}
\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix}
= \mathbf{J}
\begin{bmatrix} \frac{\partial}{\partial x} \\ \frac{\partial}{\partial y} \\ \frac{\partial}{\partial z} \end{bmatrix},
\tag{7.69}
$$

The Jacobian can be formulated at a known mesh and geometry.

7.3.5. 10-node quadratic tetrahedron element

The geometric description of a 10-node quadratic tetrahedron element and its mapping can be seen in Figure 7.17. It has four corner nodes and 6 side nodes which are located at the midpoints of the sides. These elements have 30 degrees of freedom.

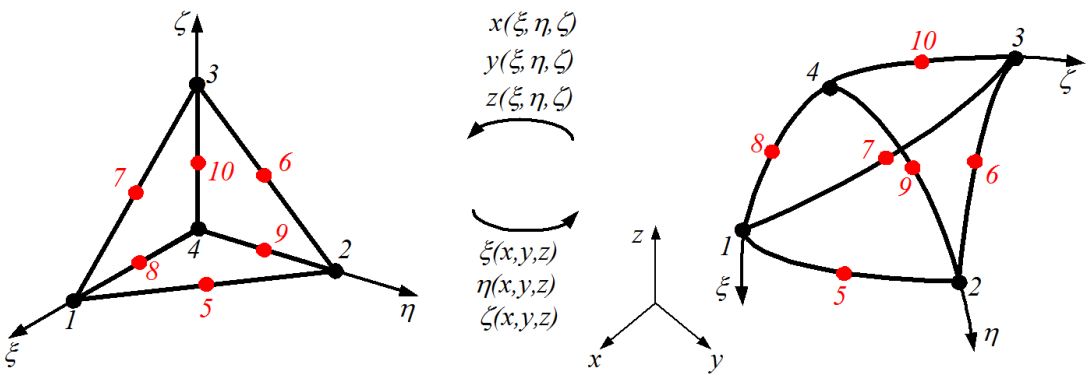


Figure 7.17. 10-node quadratic tetrahedron element

Considering a three-dimensional 10-node quadratic tetrahedron element, the coordinate interpolations

$$
\begin{aligned}
x(\xi,\eta,\zeta) &= \sum_{i=1}^{10} N_i x_i\,, \\
y(\xi,\eta,\zeta) &= \sum_{i=1}^{10} N_i y_i\,, \\
z(\xi,\eta,\zeta) &= \sum_{i=1}^{10} N_i z_i\,,
\end{aligned}
\tag{7.70}
$$

where the shape functions are

$$
\begin{aligned}
N_1(\xi,\eta,\zeta) &= \xi, \\
N_2(\xi,\eta,\zeta) &= \eta, \\
N_3(\xi,\eta,\zeta) &= \zeta, \\
N_4(\xi,\eta,\zeta) &= 1-\xi-\eta-\zeta, \\
N_5(\xi,\eta,\zeta) &= 4\xi\eta, \\
N_6(\xi,\eta,\zeta) &= 4\eta\zeta, \\
N_7(\xi,\eta,\zeta) &= 4\zeta\xi, \\
N_8(\xi,\eta,\zeta) &= \xi(1-\xi-\eta-\zeta), \\
N_9(\xi,\eta,\zeta) &= \eta(1-\xi-\eta-\zeta), \\
N_{10}(\xi,\eta,\zeta) &= \zeta(1-\xi-\eta-\zeta).
\end{aligned}
\tag{7.71}
$$

The properties of the shape functions:

1.	$\sum_{i=1}^{10} N_i\,(\xi,\eta,\zeta) = 1 \qquad -1 \le \begin{matrix}\xi\\ \eta\\ \zeta\end{matrix} \le 1$
2.	$N_i(\xi_j,\eta_j,\zeta_j) = \begin{cases} 1 & if\ \ i=j \\ 0 & if\ i \ne j \end{cases}$ where ξ_j,η_j,ζ_j is the coordinate of the jth node $j = 1,2,\ldots,10$

The approximation of the displacement field of the element assuming that $\xi = \xi(x,y,z)$, $\eta = \eta(x,y,z)$ and $\zeta = \zeta(x,y,z)$

$$
\begin{aligned}
u^e(x,y,z) &= \sum_{i=1}^{10} N_i\,(\xi,\eta,\zeta) u_i^e, \\
v^e(x,y,z) &= \sum_{i=1}^{10} N_i\,(\xi,\eta,\zeta) v_i^e, \\
w^e(x,y,z) &= \sum_{i=1}^{10} N_i\,(\xi,\eta,\zeta) w_i^e,
\end{aligned}
\tag{7.72}
$$

where u_i^e, v_i^e and w_i^e are the displacement coordinates of the ith node of the element e.

7.3.6. 14-node triquadratic tetrahedron element

A 14-node triquadratic tetrahedron element can also be constructed by adding 4 plus nodes. 3 nodes are on each face center and 1 interior node (bubble function) at the tetrahedron center point. These elements have 42 degrees of freedom.

8. NUMERICAL INTEGRATION

In the governing equations of the finite element method definite integrals (depending on the problem line-, surface- or volume integral) appear. For the determination of the stiffness matrices and the load vector of isoparametric elements numerical integration is needed.
For the element stiffness matrix

$$\mathbf{K}^e = \int_V \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, dV, \tag{8.1}$$

where

$$dV = dxdydz. \tag{8.2}$$

The nodal load vector

$$\mathbf{f}_p^e = \int_{A_p} \mathbf{N}^{eT}\mathbf{p}^e \, dA, \tag{8.3}$$

where

$$dA = dxdy \cdot 1. \tag{8.4}$$

Numerical integration types are implemented in the finite element packages. The aim is to calculate these integrals as proper as it possible. There are numerous methods for numerical integration like Newton-Cotes formulas, Gaussian quadrature, etc. Gaussian quadrature is preferred to Newton-Cotes formulas for finite element applications because they have fewer function evaluations for a given order. Using Gaussian quadrature the weights and evaluation points are determined so the integration accuracy is as high as its order is. Here, the Gaussian quadrature is detailed.
In general, the integral of function $f(x)$ can be produced numerically in the form of amount. Each term of the amount is a multiplication of a weight factor and a function value, thus

$$\int_{x=a}^{b} f(x)dx = \sum_{i=1}^{n_{int}} w_i f(x_i) + E, \tag{8.5}$$

where w_i is the weights belongs to the integration point, x_i is the coordinate of the integration point, n_{int} is the number of evaluation point and E is the error.

8.1. Gaussian quadrature

The common integration types use equally spaced sampling points. The Gaussian quadrature is based on optimizing the position of the sampling points and also optimizing the weights. The Gaussian quadrature formula is

$$\int_{x=a}^{b} f(x)dx \cong \sum_{k=1}^{n_{int}} w_k f(x_k). \tag{8.6}$$

It means that the function is evaluated at several sampling points n_{int} (Gauss points). The Gaussian method applies sampling points so required number points are needed to get the best solution. The sampling points are located symmetrically with respect to the center of the interval. The calculation is exact if the integrand is a polynomial of degree $2n-1$ or less. The accuracy of the numerical integration depends on how well the polynomial fits the given curve.
In three-dimensional case using isoparametric finite elements the

$$dV = dxdydz = Jd\xi d\eta d\zeta. \tag{8.7}$$

In two-dimensional case using isoparametric finite elements the

$$dA = dxdy \cdot 1 = Jd\xi d\eta. \tag{8.8}$$

In one-dimensional case (see Figure 8.1.) the Gauss-type integration

$$\int_a^b f(x)dx = \int_{\xi=-1}^{\xi=1} f[x(\xi)]Jd\xi \cong \sum_{k=1}^{n_{int}} w_k f[x(\xi_k)]J, \tag{8.9}$$

where $J = (b-a)/2 > 0$, ξ_k are the integration points, w_k are the weights.

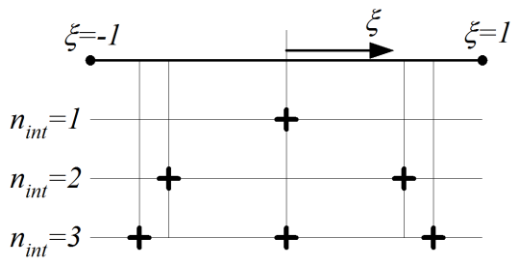


Figure 8.1. Numerical integration in one-dimensional case

The Gaussian coordinates and Gaussian weights for the interval $a = -1$ to $b = 1$ is given below:

n_{int}	ξ_k	w_k
1	0.000000000000000	2.000000000000000
2	-0.577350269189626	1.000000000000000
	0.577350269189626	1.000000000000000
3	-0.774596669241483	0.555555555555555
	0.000000000000000	0.888888888888888
	0.774596669241483	0.555555555555555

Table 8.1. The Gaussian coordinates and Gaussian weights

Example 10.
Numerical calculation of a one-dimensional integral. The $f(x)$ function is given. Determine the exact value of the integral of given interval and compare the result with the results by using one-point, two-point and three-point formulas.

Data:
$f(x) = sinx$
$a = 0, b = \pi/2$

The exact value of the integral:

$$I = \int_0^{\pi} sinx\, dx = [-cosx]_0^{\pi} = -cos\pi - (-cos0) = -(-1) - (-1) = 2$$

In general, the Gaussian quadrature leads to

$$\int_a^b f(x)\, dx = \int_{\xi=-1}^{\xi=1} f[x(\xi)]Jd\xi \cong \sum_{k=1}^{n_{int}} w_k f[x(\xi_k)]J,$$

where $J = (b-a)/2 = \pi/2$.

10a, The usage of one-point formula
($n_{int} = 1$), where $\xi_1 = 0$ and $w_1 = 2$:

$$I_{n_{int}=1} = \int_{\xi=-1}^{\xi=1} sin[x(\xi)]Jd\xi \cong \sum_{k=1}^{1} w_k sin[x_k(\xi_k)]J = w_1 sin[x(\xi_1)]J$$

The mapping

$$x(\xi) = \sum_{i=1}^{2} N_i x_i = N_1 x_1 + N_2 x_2 = \frac{1}{2}(1-\xi)0 + \frac{1}{2}(1+\xi)\pi = \frac{\pi}{2}(1+\xi)$$

For $\xi_1 = 0$ the $x(\xi_1) = \pi/2$, thus

$$I_{n_{int}=1} = w_1 sin[x(\xi_1)]J = 2 \cdot sin\frac{\pi}{2} \cdot \frac{\pi}{2} = \pi = 3{,}14 \neq 2$$

The error is 54%.

10b, The usage of two-point formula
($n_{int} = 2$), where $\xi_1 = -0.577350269189626$, $\xi_2 = 0.577350269189626$ and $w_1 = 1$, $w_2 = 1$:

$$I_{n_{int}=2} = \int_{\xi=-1}^{\xi=1} sin[x(\xi)]Jd\xi \cong \sum_{k=1}^{2} w_k sin[x_k(\xi_k)]J = w_1 sin[x(\xi_1)]J + w_2 sin[x(\xi_2)]J$$

The mapping

$$x(\xi) = \sum_{i=1}^{2} N_i x_i = N_1 x_1 + N_2 x_2 = \frac{1}{2}(1-\xi)0 + \frac{1}{2}(1+\xi)\pi = \frac{\pi}{2}(1+\xi)$$

For $\xi_1 = -0.577350269189626$ the $x(\xi_1) = 38.03847577°$ and
for $\xi_1 = 0.577350269189626$ the $x(\xi_2) = 141.9615242°$, thus

$$I_{n_{int}=2} = w_1 sin[x(\xi_1)]J + w_2 sin[x(\xi_2)]J = 1.935819575 \neq 2$$

The error is 3.31%.

10c, The usage of three-point formula
($n_{int} = 3$), where $\xi_1 = --0.774596669241483$, $\xi_2 = 0$, $\xi_3 = 0.774596669241483$, and $w_1 = 0.555555555555555$, $w_2 = 0.888888888888888$, $w_3 = 0.555555555555555$:

$$I_{n_{int}=3} = \int_{\xi=-1}^{\xi=1} sin[x(\xi)]Jd\xi \cong \sum_{k=1}^{3} w_k sin[x_k(\xi_k)]J$$

The mapping

$$x(\xi) = \sum_{i=1}^{2} N_i x_i = N_1 x_1 + N_2 x_2 = \frac{1}{2}(1-\xi)0 + \frac{1}{2}(1+\xi)\pi = \frac{\pi}{2}(1+\xi)$$

For $\xi_1 = -0.774596669241483$ the $x(\xi_1) = 20.28629977°$ and
for $\xi_2 = 0$ the $x(\xi_2) = \pi/2$ and
for $\xi_3 = 0.774596669241483$ the $x(\xi_3) = 159.7137002°$, thus

$$I_{n_{int}=3} = w_1 sin[x(\xi_1)]J + w_2 sin[x(\xi_2)]J + w_3 sin[x(\xi_3)]J = 2.001388914 \approx 2$$

The error is 0.069%.

In two-dimensional case the Gauss-type integration over quadrilateral domain

$$\begin{aligned} \int_A f(x,y)dA &= \int_x \int_y f(x,y)dxdy = \\ &= \int_{\xi=-1}^{\xi=1}\int_{\eta=-1}^{\eta=1} f[x(\xi,\eta),y(\xi,\eta)]J(\xi,\eta)d\xi d\eta \cong \\ &\cong \sum_{k=1}^{n_{int}}\sum_{l=1}^{n_{int}} w_k w_l f[x(\xi_k,\eta_l),y(\xi_k,\eta_l)]J(\xi_k,\eta_l), \end{aligned} \tag{8.10}$$

where ξ_k and η_l are the integration points, w_k and w_l are the weights. Using *2x2* integration order (four-point Gauss rule) the location of the integration points can be seen in Figure 8.2.

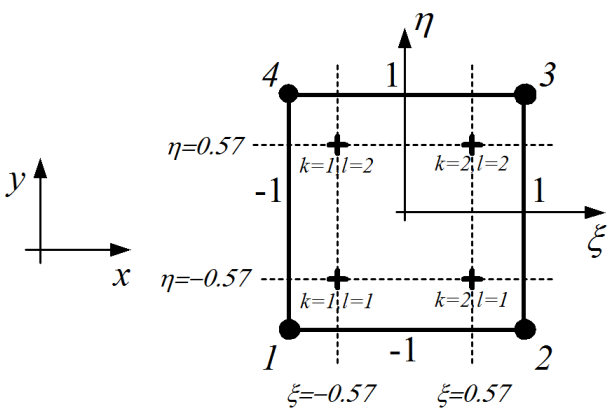


Figure 8.2. Numerical integration in two-dimensional case for quadrilateral element

In this case the integral can be approximated with

$$\sum_{k=1}^{2}\sum_{l=1}^{2} w_k w_l f[x(\xi_k,\eta_l), y(\xi_k,\eta_l)]J(\xi_k,\eta_l) =$$
$$= w_1 w_1 f[x(\xi_1,\eta_1), y(\xi_1,\eta_1)]J(\xi_1,\eta_1) + w_1 w_2 f[x(\xi_1,\eta_2), y(\xi_1,\eta_2)]J(\xi_1,\eta_2) =$$
$$= w_2 w_1 f[x(\xi_2,\eta_1), y(\xi_2,\eta_1)]J(\xi_2,\eta_1) + w_2 w_2 f[x(\xi_2,\eta_2), y(\xi_2,\eta_2)]J(\xi_2,\eta_2), \tag{8.11}$$

where $\xi_1 = \eta_1 = -0.577350269189626$, $\xi_2 = \eta_2 = 0.577350269189626$ and $w_1 = w_2 = 1$.
In two-dimensional case the Gauss-type integration over triangular domain

$$\int_A f(x,y)dA = \int_x \int_y f(x,y)dxdy \cong$$
$$\cong \sum_{k=1}^{n_{int}} w_k f[x(\xi_k,\eta_k), y(\xi_k,\eta_k)]J(\xi_k,\eta_k), \tag{8.12}$$

where ξ_k and η_k are the integration points, w_k are the weights. Using three-point Gauss rule the location of the integration points can be seen in Figure 8.3.

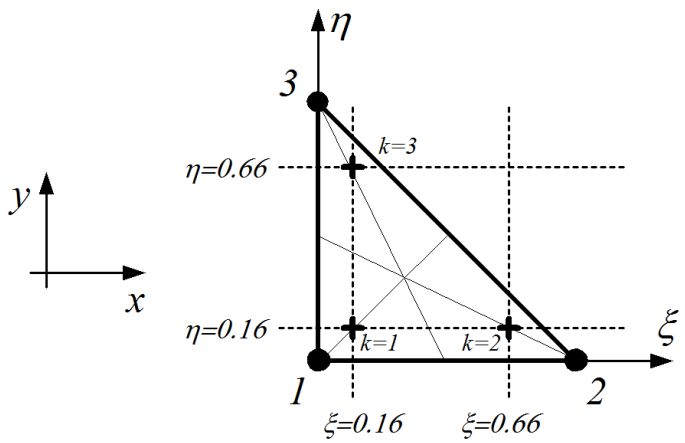


Figure 8.3. Numerical integration in two-dimensional case for triangular element

In this case the integral can be approximated with

$$\sum_{l=1}^{3} w_k f[x(\xi_k,\eta_k), y(\xi_k,\eta_k)]J(\xi_k,\eta_k) =$$
$$= w_1 f[x(\xi_1,\eta_1), y(\xi_1,\eta_1)]J(\xi_1,\eta_1) + w_2 f[x(\xi_2,\eta_2), y(\xi_2,\eta_2)]J(\xi_2,\eta_2) =$$
$$= w_3 f[x(\xi_3,\eta_3), y(\xi_3,\eta_3)]J(\xi_3,\eta_3), \tag{8.13}$$

where $\xi_1 = \eta_1 = \xi_3 = \eta_3 = 0.166666666666667$, $\xi_2 = \eta_2 = 0.666666666666667$ and $w_1 = w_2 = w_3 = 0.333333333333333$.
In three-dimensional case the Gauss-type integration over eight node brick element

$$\int_V f(x,y,z)dV = \int_x \int_y \int_z f(x,y,z)dxdydz =$$
$$= \int_{\xi=-1}^{\xi=1}\int_{\eta=-1}^{\eta=1}\int_{\zeta=-1}^{\zeta=1} f[x(\xi,\eta,\zeta), y(\xi,\eta,\zeta), z(\xi,\eta,\zeta)]J(\xi,\eta,\zeta)d\xi d\eta d\zeta \cong \tag{8.14}$$
$$\cong \sum_{k=1}^{n_{int}}\sum_{l=1}^{n_{int}}\sum_{m=1}^{n_{int}} w_k w_l w_m f[x(\xi_k,\eta_l,\zeta_m), y(\xi_k,\eta_l,\zeta_m), y(\xi_k,\eta_l,\zeta_m)]J(\xi_k,\eta_l,\zeta_m),$$

where ξ_k, η_l and ζ_m are the integration points, w_k, w_l and w_m are the weights. Using *2x2x2* integration order (eight-point Gauss rule) the Gaussian coordinates $\xi_1 = \eta_1 = \zeta_1 = -0.577350269189626$, $\xi_2 = \eta_2 = \zeta_2 = 0.577350269189626$ and $w_1 = w_2 = 1$.

8.2. Evaluation of the stiffness matrix by Gaussian quadrature

For two-dimensional quadrilateral element the element stiffness matrix using Eq. 8.1 and Eq. 8.4

$$\mathbf{K}^e = \int_A \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, dA = \int_x \int_y \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, dxdy, \tag{8.15}$$

where the integrand is a function of x and y and nodal coordinate values. The element stiffness matrix for a quadrilateral element can be evaluated in terms of a local set of coordinates ξ, η with limits from *-1* to *1* within the element,

$$\mathbf{K}^e = \int_{-1}^{1}\int_{-1}^{1} \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, J d\xi d\eta. \tag{8.16}$$

The integrand can be evaluated by numerical integration in the same manner. Using *2x2* integration order (four-point Gauss rule) we have

$$\begin{gathered}\mathbf{K}^e = \int_{-1}^{1}\int_{-1}^{1} \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, J d\xi d\eta \cong \\ \cong \mathbf{B}(\xi_1,\eta_1)^{eT}\mathbf{C}^e\mathbf{B}(\xi_1,\eta_1)^e J(\xi_1,\eta_1) w_1 w_1 + \\ + \mathbf{B}(\xi_1,\eta_2)^{eT}\mathbf{C}^e\mathbf{B}(\xi_1,\eta_2)^e J(\xi_1,\eta_2) w_1 w_2 + \\ + \mathbf{B}(\xi_2,\eta_1)^{eT}\mathbf{C}^e\mathbf{B}(\xi_2,\eta_1)^e J(\xi_2,\eta_1) w_2 w_1 + \\ + \mathbf{B}(\xi_2,\eta_2)^{eT}\mathbf{C}^e\mathbf{B}(\xi_2,\eta_2)^e J(\xi_2,\eta_2) w_2 w_2,\end{gathered} \tag{8.17}$$

where $\xi_1 = \eta_1 = -0.577350269189626$, $\xi_2 = \eta_2 = 0.577350269189626$ and $w_1 = w_2 = 1$. Substituting the weights in Eq. 8.17 we get

$$\begin{gathered}\mathbf{K}^e = \int_{-1}^{1}\int_{-1}^{1} \mathbf{B}^{eT}\mathbf{C}^e\mathbf{B}^e \, J d\xi d\eta \cong \\ \cong \mathbf{B}(\xi_1,\eta_1)^{eT}\mathbf{C}^e\mathbf{B}(\xi_1,\eta_1)^e J(\xi_1,\eta_1) + \\ + \mathbf{B}(\xi_1,\eta_2)^{eT}\mathbf{C}^e\mathbf{B}(\xi_1,\eta_2)^e J(\xi_1,\eta_2) + \\ + \mathbf{B}(\xi_2,\eta_1)^{eT}\mathbf{C}^e\mathbf{B}(\xi_2,\eta_1)^e J(\xi_2,\eta_1) + \\ + \mathbf{B}(\xi_2,\eta_2)^{eT}\mathbf{C}^e\mathbf{B}(\xi_2,\eta_2)^e J(\xi_2,\eta_2).\end{gathered} \tag{8.18}$$

The next step is to determine the Jacobian at every Gaussian point, than to determine the strain-displacement matrices at every Gaussian point and to execute the multiplication.

9. GENERAL PURPOSE FINITE ELEMENT PROGRAMS

A finite element analysis consists of three separated modules. The first is the preprocessing, the second is the solving (processing) and the third is the postprocessing, see in Figure 9.1.

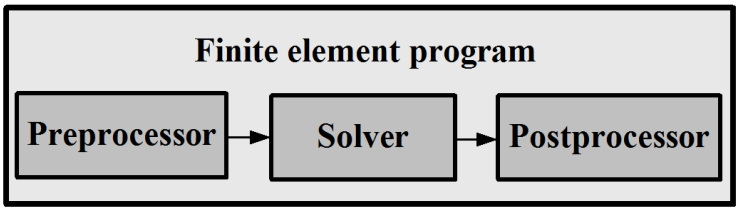


Figure 9.1. Modules of the finite element analysis

The preprocessing involves the developing an appropriate finite element mesh, assign suitable material properties and apply boundary conditions in the form of loads and constrains.
Basic steps of the preprocessing module:

1. The geometry construction of the structure or part
   a, geometry import
   b, geometry construction
      points,
      lines,
      surfaces,
      volumes.
2. The finite element meshing of the structure or part
      nodes,
      element.
3. The material information of the structure or part
4. Loadings of the structure or part
5. Supports of the structure or part

The solving stage involves the stiffness generation and solution of equations and results in the evaluation of nodal variables. This is a black box operation. Here, the governing equations are assembled into matrix form and are solved numerically.
Basic steps of the solving module:

1. The construction of the element stiffness matrices and nodal force vectors
2. The construction of the global stiffness matrix and global force vector
3. Taking the kinematical boundary conditions into consideration
4. Solution of the linear algebraic equation system which results in the nodal displacements
5. Computation of the strains and stresses in each node or at inner element points

The postprocessing stage deals with the representation of results. Once the solution is verified, the quantities of interest can be examined.
Typical steps of the postprocessing module in structural analysis:
1. Representation of the deformed configuration
2. Representation of the stresses, loading, reaction forces

9.1. Introduction to Femap 9.3

The Femap is a CAD independent Windows-native pre- and postprocessor. It is CAD independent which means it offers geometry access to CAD systems such as Solid Edge, NX, Catia, SolidWorks, etc. The Femap is integrated in NX Nastran, but also supports industry standard solvers; MSC Nastran, Abaqus, Ansys, MSC Marc, LS-DYNA, etc.
The line element library of the Femap is summarized in Table 9.1.

Rod	Uniaxial element with tension and compression. No shear and bending.	Line, connecting two nodes.
Tube		Line, connecting two nodes.
Curved tube		Arc, connecting two nodes.
Bar	Uniaxial element with tension, compression, torsion and bending.	Line, connecting two nodes.
Beam	Uniaxial element with tension, compression, torsion and bending.	Line, connecting two or three nodes.
Curved beam		Arc, connecting two nodes.
Link		Line, connecting two nodes.
Spring	Stiffness and damper element.	Line, connecting two nodes.
DOF spring		Connects two nodes.
Gap		Line, connecting two nodes.
Plot only		Line, connecting two nodes.

Table 9.1. Line element library

The plane element library of the Femap is summarized in Table 9.2.

Shear panel	Resists only shear forces.	Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Membrane		Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Bending	Resists only bending forces.	Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Plate		Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Laminate		Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Plane strain	Used to model very thick solids which have a constant cross section	Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Axisymmetric shell		Linear and parabolic lines defined by two or three nodes.
Planar plot only		Linear triangular and linear quadrilateral.

Table 9.2. Plane element library

The volume element library of the Femap is summarized in Table 9.3.

Axisymmetric	Two dimensional element used to represent volumes of revolution	Planar, three-noded or six-noded triangle, four-noded or eight noded quadrilateral.
Solid	Three dimensional solid element used to represent any three dimensional structure.	Four-noded or ten-noded tetrahedron, eight-noded or twenty-noded brick.

Table 9.3. Volume element library

The material library of the Femap is summarized in Table 9.4.

Isotropic	Constant properties in all directions.
Orthotropic	Material properties are direction dependent.
Anisotropic	More general than orthotropic with the same properties.
Hyperelastic	Materials subject to large deformations.

Table 9.4. Material library

The loads library of the Femap is summarized in Table 9.5.

Body or global	Acceleration	Translational (gravity) and rotational.
	Velocity	Rotational
	Thermal	Default temperature.
Nodal	Force/moment	
	Displacement	
	Velocity	
	Temperature	
	Heat generation	Heat energy/unit volume
	Heat flux	Heat energy/unit area
Elemental	Distributed	Load/length
	Pressure	
	Temperature	
	Heat generation	Heat energy/unit volume
	Heat flux	Heat energy/unit area
	Convection	
	Radiation	
Geometry-based	Points	
	Lines	
	Surfaces	

Table 9.5. Loads library

In Femap nodal constraints, geometry based constraints and even constraint equations can be created. To prevent nodes from moving in any of six degrees of freedom (translations and rotations) constraints can be applied. Geometry based constraints have three options; fixed, pinned or no rotations.

10. NUMERICAL EXAMPLES

Numerical example 1. (Prismatic bar - programming)
Consider the case of a uniform prismatic bar which is loaded by its own weight. The bar is fixed at one end. The material of the bar is steel. The problem is illustrated in Figure 10.1.

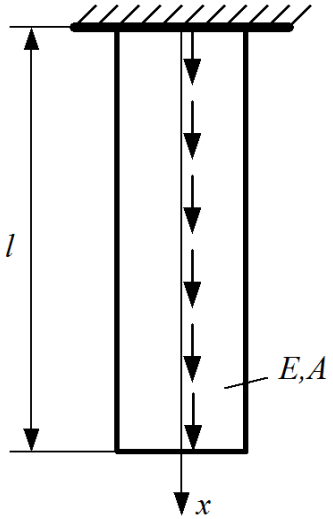


Figure 10.1. The uniform prismatic bar

<u>Data:</u>
$d = 30mm$
$E = 2{,}1 \cdot 10^5 \frac{N}{mm^2}$
$l = 2000mm$
$\varrho = 7800 \frac{kg}{m^3}$

Comparison of the displacement and the stress when using 1, 2, 4 or 8 finite elements.

Source code written in Scilab 3.1:

```
//input data (Nelem has to be changed only; for 1, 2, 4 or 8)//
Nelem=1;
Nnodes=Nelem+1;
E=2.1e5;
A=7.0685e2;
g=9.81;
ro=7.8e-6;
L=2000;
pm=A*ro*g;
Le=L/Nelem;
Kg=zeros(Nnodes,Nnodes);
fg=zeros(Nnodes,1);
q=zeros(Nnodes,1);
s=zeros(Nnodes,1);
```

```
//cycle for the elements//
for ie=1:Nelem
        Ke=zeros(2,2);
        fe=zeros(2,1);
        Ke(1,1)=A*E/Le;
        Ke(1,2)=-A*E/Le;
        Ke(2,1)=-A*E/Le;
        Ke(2,2)=A*E/Le;
        fe(1,1)=pm*Le/2;
        fe(2,1)=pm*Le/2;
        ik=ie;
        iv=ie+1;
        jk=ie;
        jv=ie+1;
        Kg(ik:iv,jk:jv)=Kg(ik:iv,jk:jv)+Ke;
        fg(ik:iv,1)=fg(ik:iv,1)+fe;
end

//boundary conditions//
Kg(:,1)=zeros(Nnodes,1);
Kg(1,:)=zeros(1,Nnodes);
Kg(1,1)=1;
fg(1,1)=0;
Kgi=inv(Kg);
q=Kgi*fg;
for i=1:Nnodes
        s(i,1)=(i-1)*Le;
end
subplot(2,1,1)
plot2d(s,q)
B=zeros(1,2);
B(1,1)=-1/Le;
B(1,2)=1/Le;
qe=zeros(2,1);

//cycle for the elements//
for ie=1:Nelem
        qe(1,1)=q(ie,1);
        qe(2,1)=q(ie+1,1);
        xe(1,1)=s(ie,1);
        xe(2,1)=s(ie+1,1);
        sige=E*B*qe
        ye(1,1)=sige;
        ye(2,1)=sige;
        subplot(2,1,2)
        plot2d(xe,ye)
end
```

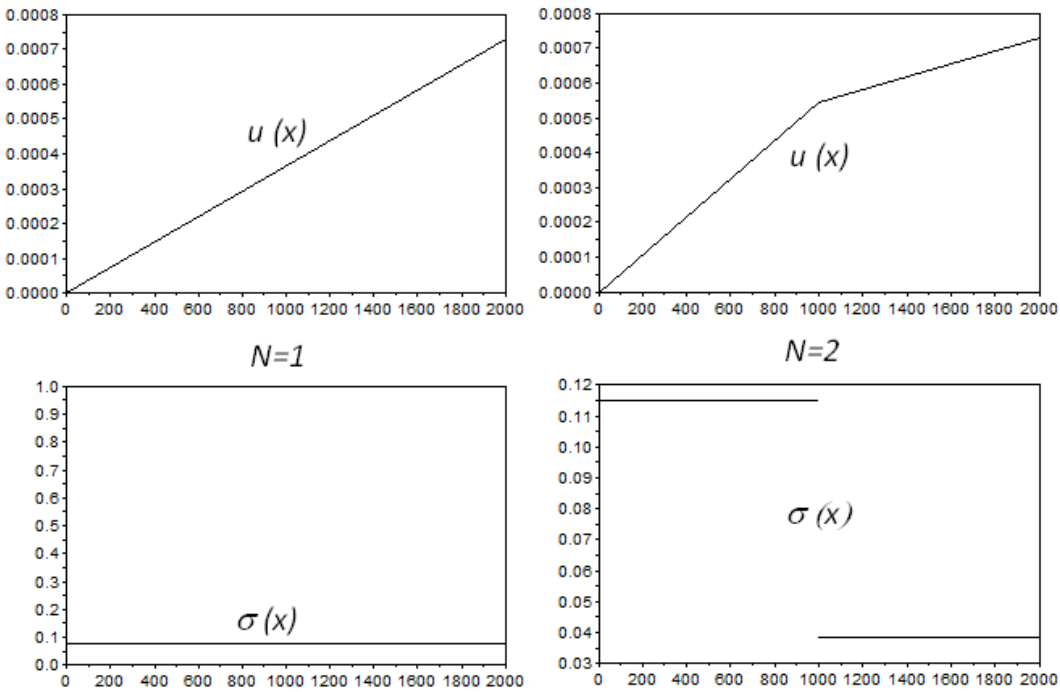


Figure 10.2. The results for $N = 1$ and $N = 2$

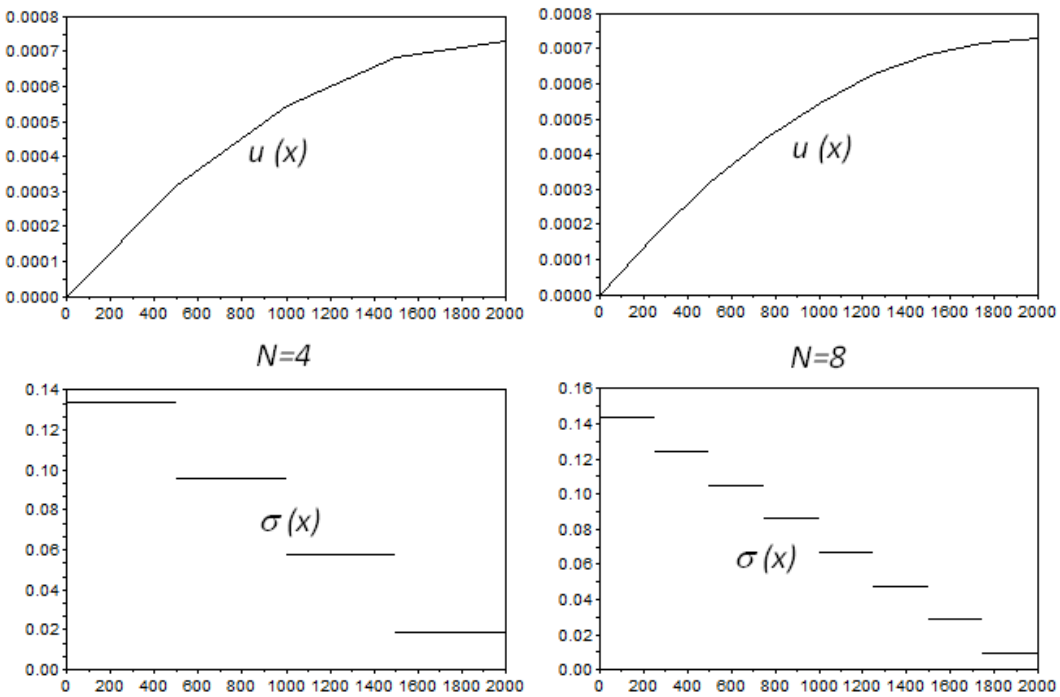


Figure 10.3. The results for $N = 4$ and $N = 8$

Numerical example 2 (Prismatic bar – software)
Uniform prismatic bar having circular cross section is loaded by its own weight and by distributed force system along the flange. The bar is fixed at the other end. The material of the bar is aluminum. The problem is illustrated in Figure 2.9. The bar is modeled with three line elements along its length. The first task is to construct the finite element model then analyze it with the Femap 9.3 finite element program. The postprocessing task is to evaluate the nodal displacements and the normal forces. Finally, illustrate the normal forces and the reactions.

Data:
$d = 30mm$
$E = 6{,}9 \cdot 10^4 \frac{N}{mm^2}$
$l = 1000mm$
$\varrho = 2700 \frac{kg}{m^3}$
$F_x = 5N$

Defining model Geometry:
//The first step is to define the geometry from which the finite element model can be built up. The simple geometry is better to be constructed in the Femap then importing a CAD geometry.//
Geometry/Curve-Line/Project Points…
In Locate- Enter First Location for Projected Line dialog box set
X=0; Y=0; Z=0, then Click OK
In Locate- Enter Second Location for Projected Line dialog box set
X=1000; Y=0; Z=0, then Click OK, then Cancel
Hit Ctrl+A key to Autoscale the graphics window

Figure 10.4. The prismatic bar constructed with a simple line

Defining the Material and Property
//Before meshing it is necessary to define the element property and the material. The defined property contains the type of the chosen finite element and also the assigned constitutive law.//

Defining the Material
//The material of the bar is aluminum which is a linear, isotropic and homogeneous material. The Hooke's law is valid so it is enough to give the Young's modulus and the Poisson ratio. For considering the body load in the finite element model the density of the bar has to be given.//
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set

Title: Aluminum
Young Modulus, E: 6,9E4
Poisson's Ratio, nu: 0,33
Mass Density: 2,7E-6
Click OK (material 1 created), then Cancel

Defining the Property

//After defining the material the property of the chosen beam elements has to be given. The beam element can be applied in the case of mechanical problems modeled with line elements. The Femap 9.3 contains general cross sections can be used. Here circular cross section is applied.//

Model/Property
In Define Property – PLATE Element Type dialog box
Click Elem/Property Type
In Element/Property Type dialog box choose Beam (line Elements)
Click OK
In Define Property – BEAM Element Type dialog box
Give a Title such as: 30mm diameter BEAM
Material: 1..Aluminum
Click Shape button
In Cross Section Defining dialog box
Shape: Circular Bar
Radius: 15
Click Draw section, then Click OK
In Define Property – BEAM Element Type dialog box
Click OK (Property 1 created), then Cancel

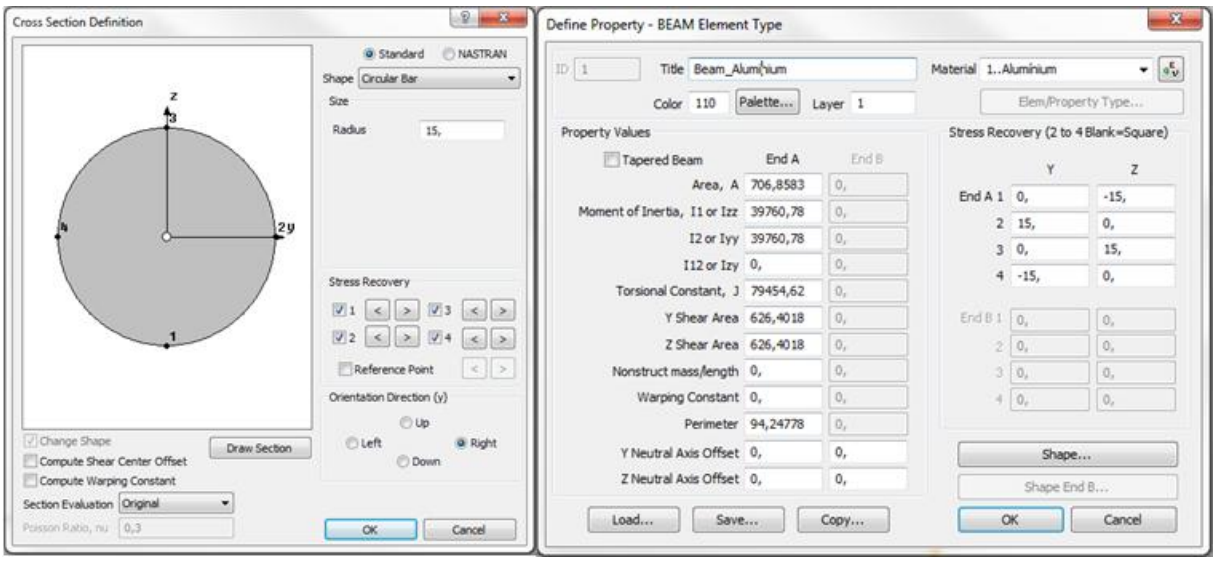


Figure 10.5. The defined cross section and property

Meshing the Model

//The next step is the meshing. Firstly the element number along the curve has to be set. The orientation of the element also has to be given. The orientation vector cannot be parallel to the meshed line. Finally, the nodal numbers are illustrated.//

Mesh/Mesh control/Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box

Select the defined curve in the graphics window, then click OK
In Mesh Size Along Curve dialog box
Select Number of Elements, then write 3 in field
click OK, then Cancel

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s)to Mesh dialog box
Select the defined curve in the graphics window, then click OK
In Geometry Mesh Options dialog box set
Property: 1..30 mm diameter BEAM, then Click OK
In Vector Locate – Define Element Orientation Vector dialog box set
In Base fields: X=0; Y=0; Z=0
In Tip fields: X=0; Y=1; Z=0
Click OK (3 Elements Created)
View/ Visibility
In Visibility dialog box Select Entity/ Label Tab
Pick Labels, then Click to All Off Button
In Mesh section Select Node and Element, then Click Done

Applying Constraints
Model/ Constraint/ On Point
In New Constraint Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Point(s) to Select dialog box
Choose the point 1 on the left end of the rod , then click OK
In Create Constraint on Geometry dialog box
Title: Support
Select Fixed, then Click OK, then Click Cancel
View/ Visibility
In Visibility dialog box Select Entity/ Label Tab
Pick Labels, then Click to All Off Button
In Mesh section Select Node and Element, then Click Done

Applying Load
//The boundary conditions are set. Now the external forces and the force comes from the body load have to be defined.//
Model/ Load/ On Point
In New Load Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Point(s) to Select dialog box
Choose the point 2 on the right end of the rod , then click OK
In Create Loads on Points dialog box
Title: húzás; Choose Force and set Load value to: Fx=5
Click OK, then Cancel
Model/Load/Body
In Create Body Loads dialog box
Translational Accel/Gravity (lenght/time/time) select Active
Set Ax value to 9,81
Click OK
Select Loads Visibility

Hit Ctrl+G key to Regenerate the graphics window
//Now the direction and magnitude of the gravity can be seen in the coordinate system.//

Figure 10.6. The finite element model of the prismatic bar

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the Results
//The important data from the finite element analysis are now the nodal displacements and the normal forces. Now the results have to be listed.//
List/Output/Standard…
In Select Output Set(s) to list dialog box
Select 1..NX NASTRAN Case 1, then Click OK
In List Formatted Output dialog box
Title: Elmozdulások
Sort Field: Choose 2..T1 Translation from the drop down list
Format ID: Select 0..NASTRAN Displacement, then click OK
In Entity Selection – Select Node(s) to List dialog box
Click Select All button, then Click OK

Displacements			
Point ID	T1	T2	T3
1	0	0	0
2	1,408022E-4	0	0
3	2,389522E-4	0	0
4	2,944501E-4	0	0

Table 10.1. The nodal displacement values

List/Output/Standard…
In Select Output Set(s) to list dialog box
Select 1..NX NASTRAN Case 1, then Click OK
In List Formatted Output dialog box
Title: Normal forces
Sort Field: Choose 3022..Beam EndA Axial Forces from the drop down list
Format ID: Select 0..NASTRAN CBEAM Forces, then click OK
In Entity Selection – Select Element(s) to List dialog box

Click Select All button, then Click OK

Forces in beam elements			
Element ID	Moments	Shears	Axial forces
3	0	0	8,120426E+0 [*N*]
2	0	0	1,436128E+1 [*N*]
1	0	0	2,060213E+1 [*N*]

Table 10.2. The normal force values

Illustrate the element Axial Forces
View/Select (or hit F5 button)
In View Select dialog box
Deformed Style: Deform
Contour Style: Criteria
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Output Sets: NX NASTRAN Case 1
Deform: 1..Total Translation
Contour: 3022..Beam EndA Axial Force
Click OK all dialog boxes

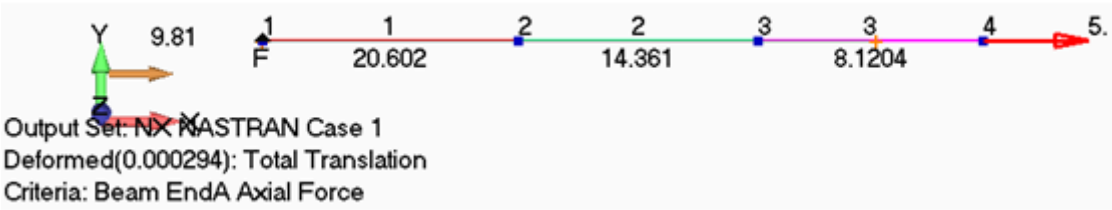


Figure 10.7. The representation of the normal forces in the elements

Numerical example 3 (Analyzing a welded structure)
In this example a welded structure is analyzed using beam elements.

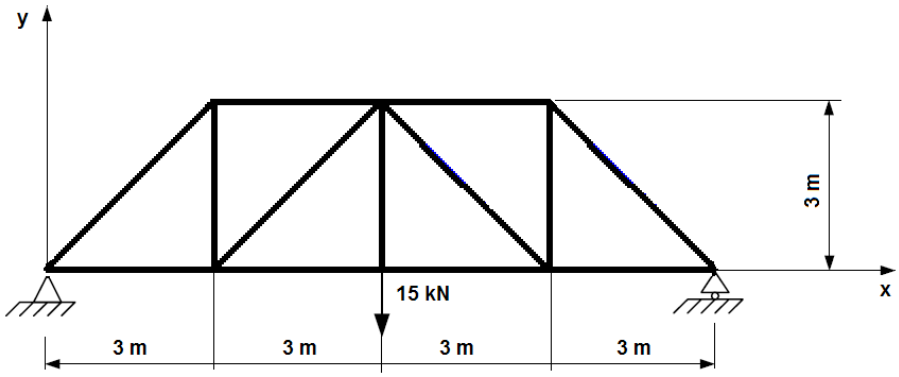


Figure 10.8. The geometry of the structure

<u>Data:</u>
Cross section properties of the circular tube:
Diameter: 200*mm*
Thickness: 7*mm*
Cross section properties of the hollow section:
Height: 250*mm*
Width: 150*mm*
Thickness: 7*mm*
Material: S235 steel
Force: *F=15kN*

Defining model Geometry:
//In the engineering practice millimeter is used, but due to the sizes of structure the geometry parameters of the structure are given in metres.//
Geometry/Point
In Locate - Enter Coordinates or Select with Cursor dialog box set
X=0; Y=0; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=3; Y=0; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=3; Y=3; Z=0, Click OK then Cancel
Hit Ctrl+A key to Autoscale the graphics window

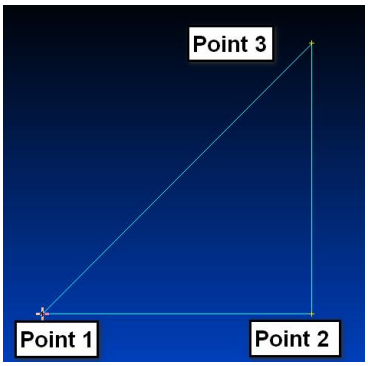


Figure 10.9. The left side of the structure

Geometry/Curve-Line/Points…
In Create Line from Points dialog box
Pick point 1 then point 2
Click OK (curve 1 created)
In Create Line from Points dialog box
Pick point 2 then point 3
Click OK (curve 2 created)
In Create Line from Points dialog box
Pick point 3 then point 1
Click OK (curve 3 created), then Cancel

Geometry/Copy/Curve…
In Entity Selection – Select Curve(s) to Copy dialog box
Select All
Click OK
In Generation Options dialog box
Repetitions: 1
Click OK
In Vector Locate – Select Vector to Copy Along dialog box
Base: X=0; Y=0; Z=0
Tip: X=3; Y=0; Z=0
Click OK

Geometry/Curve-Line/Points…
In Create Line from Points dialog box
Pick point 3 then point 6
Click OK (curve 7 created)

Geometry/Reflect/Curve…
In Entitty Selection – Select Curve(s) to Reflect dialog box
Select all
Pick Remove
Pick curve 5 (or write 5 in ID, than Click More)
Click OK
In Generation Options
Click OK
In Plane Locate – Select Reflection Plane
Select Base X field, then hit Ctrl+P key (in this way you can select points in the graphics window)
In Base field select point 5
In Point 1 field select point 6
In Point 2 field write X=6;Y=0; Z=1
Click OK

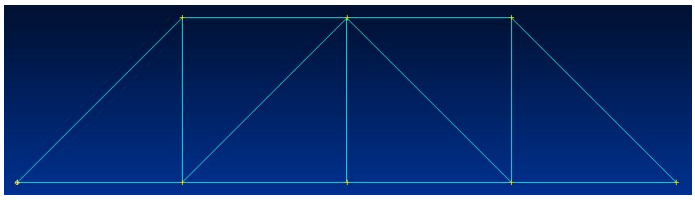


Figure 10.10. The geometry constructed in the Femap

Defining the Material and Property
//Before meshing it is necessary to define the element property and the material. The defined property contains the type of the chosen finite element and also the assigned constitutive law.//

Defining the Material
//The material of the structure is S235 steel which is a linear, isotropic and homogeneous material. The Hooke's law is valid so it is enough to give the Young's modulus and the Poisson ratio. For considering the body load in the finite element model the density of the bar has to be given.//
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set
Title: S235 Steel
Young Modulus, E: 2,06e11 (remember we chosed *m* for unit)
Poisson's Ratio, nu: 0,3
Mass Density: 7,85e3
Click OK (material 1 created), then Cancel

Defining the Property
//After defining the material the property of the given chosen beam elements has to be given. The beam element can be applied in the case of mechanical problems modeled with line elements. The Femap 9.3 contains general cross sections can be used. Here circular tube and rectangular tube cross sections are applied.//
Model/Property
In Define Property – PLATE Element Type dialog box
Click Elem/Property Type
In Element/Property Type dialog box choose Beam (line Elements)
Click OK
In Define Property – BEAM Element Type dialog box
Give a Title such as: 200mm diameter BEAM
Material: 1..S235 Steel
Click Shape button
In Cross Section Defining dialog box
Shape: Circular Tube
Radius: 0,1
Thickness: 0,007
Click Draw section, then Click OK
In Define Property – BEAM Element Type dialog box
Click OK (Property 1 created)

In Define Property – BEAM Element Type dialog box
Title: 250x150x7 Beam
Material: 1..S235 Steel
Click Shape button
In Cross Section Defining dialog box

Shape: Rectangular Tube
Size: Height= 0,25; Width= 0,15; Thickness= 0,007
Click Draw section, then Click OK
In Define Property – BEAM Element Type dialog box
Click OK (Property 2 created), then click Cancel

Meshing the Model
Mesh/Mesh control/Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Select All, then click OK
In Mesh Size Along Curve dialog box
Select Number of Elements, then write 1 in field,
click OK, then Cancel

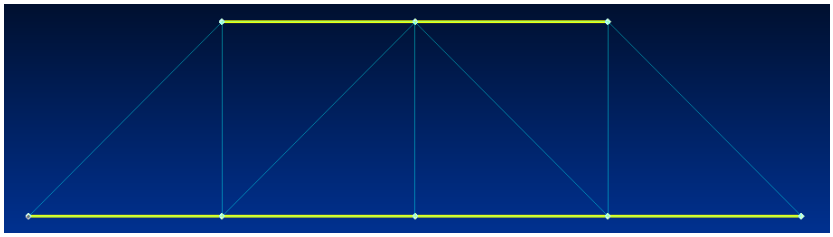


Figure 10.11. Creating the *250x150x7* Beam mesh

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s) to Mesh dialog box
Pick the highlighted curves (Curve IDs 1,4,7,8,11,13)
Click OK
In Geometry Mesh Options dialog box set
Property: 2..250x150x7 Beam, then Click OK
In Vector Locate – Define Element Orientation Vector dialog box
Click Methods, then select Global Axis
In Vector Global Axis – Define Element Orientation Vector dialog box
Select Y Axis, then Click OK (6 Element(s) Created)
F6 key (view options)
In View Options dialog box
Category: Labels, Entities, and Color
Option: select Element Orientation/Shape
Element Shape: select 3..Show Cross Section
Click OK

//Zoom and Rotate the model to see the cross section better//
Modify/ Update Elements/ Line Element Orientation
In Entity Selection – Select Element(s) to Update Orientation
Select All, then click OK
In Update Element Orientation dialog box
Select Vector, then click OK
In Vector Global Axis – Define Element Orientation Vector dialog box
Select Z Axis, then click OK
Ctrl+G key to Regenerate the window

Creating the 200mm diameter Beam Mesh

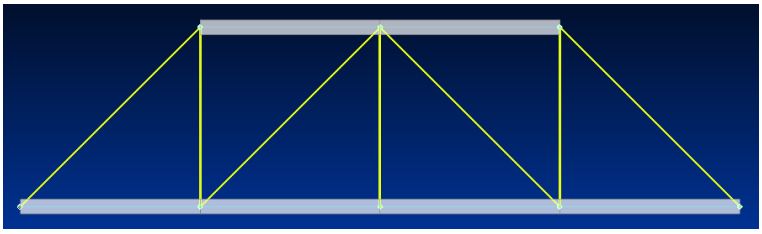


Figure 10.12. Creating the *200mm* Beam mesh

F8 key/View Rotate dialog box
Click XY_Top button, then Click OK

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s) to Mesh dialog box
Pick the highlighted curves (Curve IDs 2, 3, 5, 6, 9, 10, 12)
Click OK
In Geometry Mesh Options dialog box set
Property: 1..200mm diameter BEAM, then Click OK
In Vector Global Axis – Define Element Orientation Vector dialog box
Click Methods, then select Normal to view
In Vector Normal to view – Define Element Orientation Vector dialog box
Click OK (7 Element(s) Created)

Merging Coincident Nodes
Tools/ Check/ Coincident Nodes
In Entity Selection dialog box
Select All, then Click Ok
In Check/Merge Coincident dialog box
Action: Merge
Keep ID: Automatic
Move To: Current Location
Click OK (8 Node(s) Merged

After finishing the meshing the graphic window should look like this:

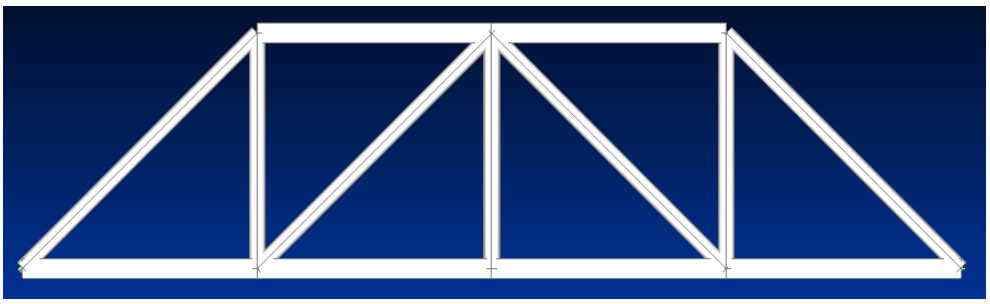


Figure 10.13. The meshed truss structure

Applying Constraints
Model/ Constraint/ Nodal
In New Constraint Set dialog box
Title: (give a title), then click OK

In Entity Selection – Enter Node(s) to Select dialog box
Pick point 5, then click OK
In Create Nodal Constraint/ DOF Dialog box
Title: Support1
Select Coord Sys: 0.. Basic Rectangular
Select TX; TY; TZ; RX; RY, then Click OK

In Entity Selection – Enter Node(s) to Select dialog box
Pick point 11, then click OK
In Create Nodal Constraint/ DOF Dialog box
Title: Support2
Select Coord Sys: 0.. Basic Rectangular
Select TY; TZ; RX; RY
Click OK, then Click Cancel

Applying Load
Model/ Load/ Nodal
In New Load Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Node(s) to Select dialog box
Pick point 10, then click OK
In Create Loads on Nodes dialog box
Title: Loading; Force FY=-15000
Click OK, then Cancel
Model/Load/Body
In Create Body Loads dialog box
Translational Accel/Gravity (lenght/time/time) select Active
Ay value= -9,81
Click OK

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the Results
F5 key (view select)
In View Select dialog box
Deformed Style: Deform
Contour Style Criteria
Click Deformed and Contour Data
Select PostProcessing Data dialog box

Output vector; Deform: 1..Total Translation
Contour: 3164..Beam EndA Max Comb Stress
Click OK all dialog box

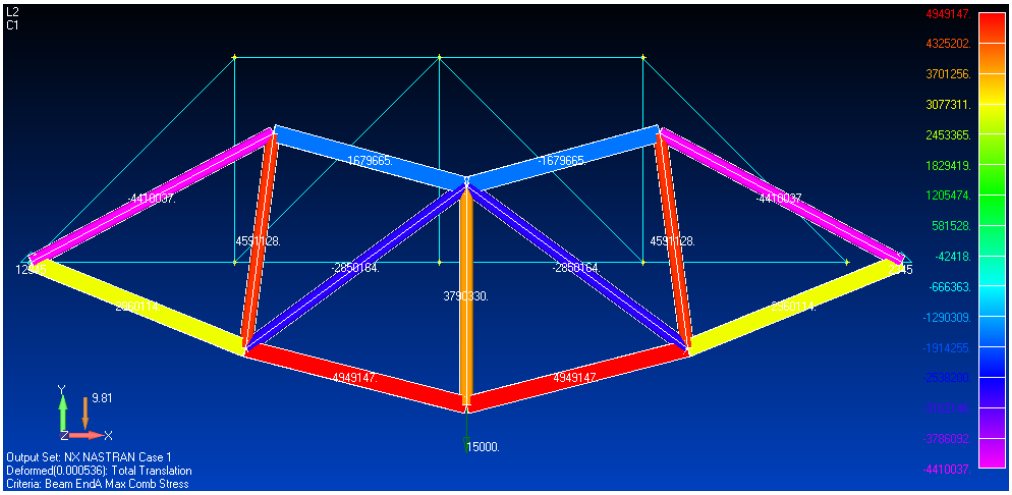


Figure 10.14. The stress state

Numerical example 4 (Statically indetermined beam)
Consider the beam shown in Figure 4.8. Solve for the displacement response.

Data:
$l = 1000mm$
$d = 30mm$
$E = 69000MPa$
$F = 500N$

Defining model Geometry:
Geometry/Curve-Line/Project Points…
In Locate- Enter First Location for Projected Line dialog box set
X=0; Y=0; Z=0, then Click OK
In Locate- Enter Second Location for Projected Line dialog box set
X=1000; Y=0; Z=0, then Click OK and finally CANCEL
Hit Ctrl+A key to Autoscale the graphics window

Defining the Material and Property
//Before meshing it is necessary to define the element property and the material. The defined property contains the type of the chosen finite element and also the assigned constitutive law.//

Defining the Material
//The material of the beam is aluminum which is a linear, isotropic and homogeneous material. The Hooke's law is valid so it is enough to give the Young's modulus and the Poisson ratio.
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set
Title: Alumínium
Young Modulus, E: 6,9E4Poisson's Ratio, nu: 0,33
Click OK (material 1 created), then Cancel

Defining the Property
//After defining the material the property of the given chosen beam elements has to be given. The beam element can be applied in the case of mechanical problems modeled with line elements. The Femap 9.3 contains general cross sections can be used. Here circular tube and rectangular tube cross sections are applied.//
Model/Property
In Define Property – PLATE Element Type dialog box
Click Elem/Property Type
In Element/Property Type dialog box choose Beam (line Elements)
Click OK
In Define Property – BEAM Element Type dialog box
Give a Title such as: 30mm diameter BEAM
Material: 1..Aluminum
Click Shape button

In Cross Section Defining dialog box
Shape: Circular Bar
Radius: 15
Click Draw section, then Click OK
In Define Property – BEAM Element Type dialog box
Click OK (Property 1 created), then Cancel

Meshing the Model
Mesh/Mesh control/Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Select the defined curve in the graphics window, then click OK
In Mesh Size Along Curve dialog box
Select Number of Elements, then write 2 in field
click OK, then Cancel

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s)to Mesh dialog box
Select the defined curve in the graphics window, then click OK
In Geometry Mesh Options dialog box set
Property: 1..30 mm diameter BEAM, then Click OK
In Vector Locate – Define Element Orientation Vector dialog box set
In Base fields: X=0; Y=0; Z=0
In Tip fields: X=0; Y=1; Z=0
Click OK (2 Elements Created)

Applying Constraints
Model/ Constraint/ Nodal
In New Constraint Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Nodal(s) to Select dialog box
Choose the nodel 1 on the left end of the rod , then click OK
In Create Nodal Constraints/DOF dialog box
Title: Support1
Select Fixed, then Click OK
In Entity Selection – Enter Node(s) to Select dialog box
Pick node 2 on the middle of the rod , then click OK
In Create Nodal Constraint/ DOF Dialog box
Title: Support2
Select Coord Sys: 0.. Basic Rectangular
Select TY, Click OK, then Click Cancel

Applying Load
Model/ Load/ Nodal
In New Load Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Nodal(s) to Select dialog box
Pick node 3 on the right end of the rod , then click OK
In Create Loads on Nodes dialog box
Title: Hajlítás; Choose Force and set Load value to: Fy=-500
Click OK, then Cancel

Analyzing the Model

//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the Results

//The important data from the finite element analysis are now the nodal displacements and the normal forces. Now the results have to be listed.//
List/Output/Results to Data Table (If this is option is unavailable, Turn on it Tools/Data Table)
OK to Unlock Data Table? Click Yes
In Send Results to Data Table dialog box
Select Output Sets: in Columns; Output Vectors: in Columns; Nodes/Elements: in Rows
In Coordinate System (Nodal Output Only) Select 0..Basic Rectangular
In Results to Add to Data Table dialog box
Pick Outputs Sets: 1..NX NASTRAN Case 1
Pick Output Vectors: 3..T2 Translation and 8..R3 Rotation; Click OK
In Entity Selection – Select Node(s) to Report
Choose Select All Button, then Click OK
//Open the Data Table and you should see the following results://

ID	CSys ID	X	Y	Z	1..NX NASTRAN Case 1, 3..T2 Translation	1..NX NASTRAN Case 1, 8..R3 Rotation
1	0	0	0	0	0	0
2	0	500	0	0	0	-0.01145968 [*rad*]
3	0	1000	0	0	-13.33895 [*mm*]	-0.03424085 [*rad*]

Table 10.3. The nodal displacements and rotations

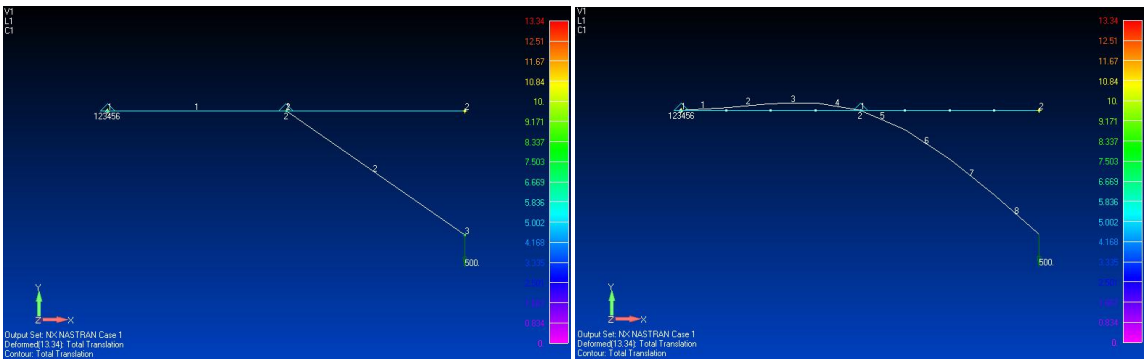


Figure 10.15. The deformed neutral axis of the beam (N=2 and N=8)

Numerical example 5 (Usage of hybrid model to analyze a hydraulic lift)
The task is to choose the proper hydraulic cylinder. The aim is that the hydraulic lift should be able to lift 500kg weight. For this purpose the proper hydraulic cylinder has to be chosen. The deformation and the stress distribution of the structure are also evaluated. After determining the stress resultant diagrams the dangerous cross section are identified.

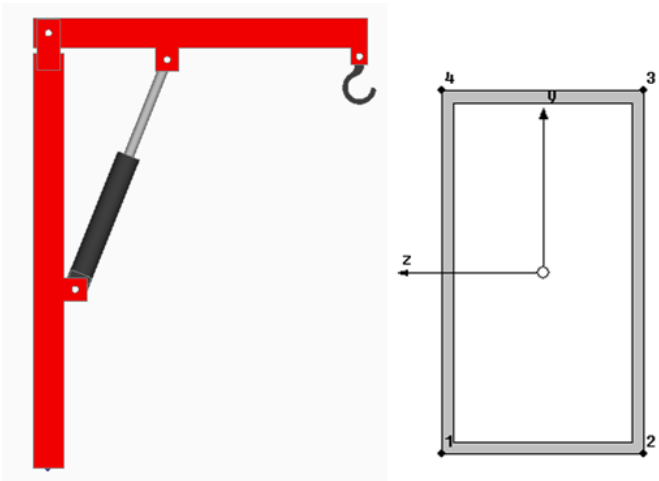


Figure 10.16. The hydraulic lift and its cross section

<u>Data:</u>
$F = mg$
$m = 500\ kg$
$a = 50\ mm$
$b = 90\ mm$
$v = 3\ mm$
$d = 30mm$
$E = 206000\ MPa$
$\nu = 0{,}3$
$\sigma_{meg} = 200$ MPa

When designing the proper mechanical model has to be constructed. The hydraulic cylinder is a standard part so its sizing is not necessary. The hydraulic cylinder is substituted with one rod element which has connections at both ends. The other part of the structure is modeled by beam elements.

Defining model Geometry:
Geometry/Point
In Locate - Enter Coordinates or Select with Cursor dialog box set
X=0; Y=0; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=0; Y=500; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=0; Y=1200; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=300; Y=1200; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=900; Y=1200; Z=0, Click OK, then Cancel
Hit Ctrl+A key to Autoscale the graphics window

Geometry/Curve-Line/Points…
In Create Line from Points dialog box
Pick point 1 then point 2 on the graphics window
Click OK (curve 1 created)
In Create Line from Points dialog box
Pick point 2 then point 3
Click OK (curve 2 created)
In Create Line from Points dialog box
Pick point 3 then point 4
Click OK (curve 3 created)
In Create Line from Points dialog box
Pick point 4 then point 5
Click OK (curve 4 created)
Pick point 2 then point 4
Click OK (curve 5 created), then CANCEL

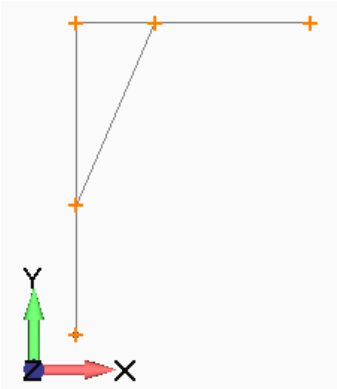


Figure10.17. The geometry of the mechanical model

Defining the Material
//The material of the structure is S235 steel which is a linear, isotropic and homogeneous material. The Hooke's law is valid so it is enough to give the Young's modulus and the Poisson ratio.
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set
Title: S235 Steel
Young Modulus, E: 2,06E5
Poisson's Ratio, nu: 0,3
Click OK (material 1 created), then Cancel

Defining the Property
//After defining the material the property of the given chosen beam elements has to be given. The beam element can be applied in the case of mechanical problems modeled with line elements. The Femap 9.3 contains general cross sections can be used. Here rectangular tube cross sections is applied for the structure and circular rod for the cylinder.//

Model/Property
In Define Property – PLATE Element Type dialog box
Click Elem/Property Type
In Element/Property Type dialog box
Choose Beam (line Elements), then Click OK
In Define Property – BEAM Element Type dialog box
Give a Title such as: BEAM 90x50x3
Material: 1..S235 Steel
Click Shape button
In Cross Section Defining dialog box
Shape: Rectangular Tube
Height: 90
Width: 50
Thickness: 3
Orientation Direction (y): Pick Up,
Click Draw section, then Click OK
In Define Property – BEAM Element Type dialog box
Click OK (Property 1 created)

In Define Property – BEAM Element Type dialog box
Click Elem/Property Type
In Element/Property Type dialog box
Choose Rod (line Elements), then Click OK
In Define Property – ROD Element Type dialog box
Title: 30 diameter ROD
Material: 1..S235 Steel
Area, A: 707
Click OK (Property 2 created), then click Cancel

Meshing the Model
Mesh/Mesh control/Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Pick to Curve: 1,2,3,4 on the graphics window, then click OK
In Mesh Size Along Curve dialog box
Select Element Size: 50
Click OK, then Cancel

In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Pick to Curve: 5 on the graphics window, then click OK
In Mesh Size Along Curve dialog box
Select Number of Elements: 1
Click OK, then Cancel

View/ Options (or hit F6 key)
In view Options dialog box
Category: Labels, Entities and Color
Options: Element – Orientation/ Shape
Element Shape: 3..Show Cross Section
Click OK

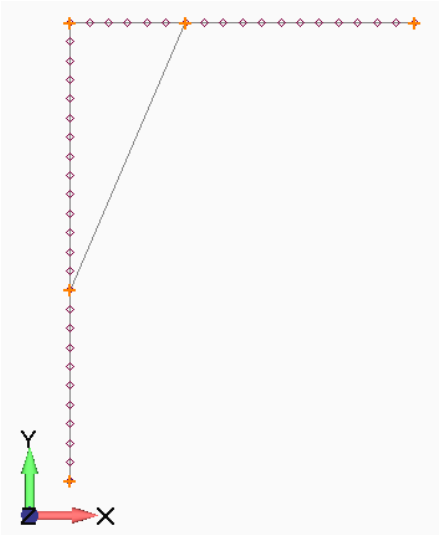


Figure 10.18. The defined nodes on the frame

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s) to Mesh dialog box
Pick curve 1,2 on the graphics window, then click OK
In Geometry Mesh Options dialog box set
Property: 1..BEAM 90x50x3, then Click OK
In Vector Locate – Define Element Orientation Vector dialog box set
Click to Methods Button and Select Global Axis from the drop down list
In Vector Global Axis – Define Element Orientation Vector
Direction: Negative; X Axis
Click Preview Button, then OK (24 Elements Created)

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s) to Mesh dialog box
Pick curve 3,4 on the graphics window, then click OK
In Geometry Mesh Options dialog box set
Property: 1..BEAM 90x50x3, then Click OK
In Vector Global Axis – Define Element Orientation Vector
Direction: Positive; Y Axis
Click OK (18 Elements Created)

Mesh/Geometry/Curve
In Entity Selection – Select Curve(s) to Mesh dialog box
Pick curve 5 on the graphics window, then click OK
In Geometry Mesh Options dialog box set
Property: 2..30 diameter ROD
Click OK (1 element Created)

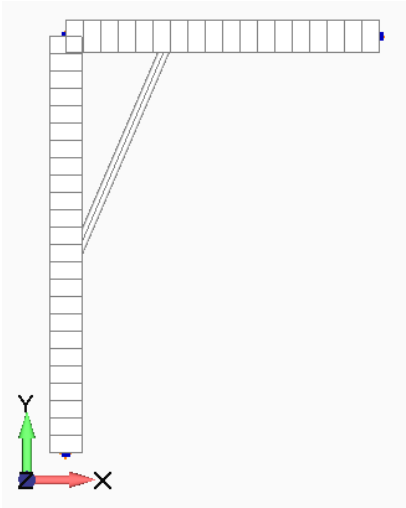


Figure 10.19. The meshed model

Merging Coincident Nodes
Tools/ Check/ Coincident Nodes
In Entity Selection dialog box
Select All, then Click Ok
In Check/Merge Coincident dialog box
Tolerance: 1
Click Preview Button (You had to see 3 coincident nodes)
In Preview Coincident dialog box, Click to Done
In Check/Merge Coincident dialog box, Click to OK (3 Nodes Merged)

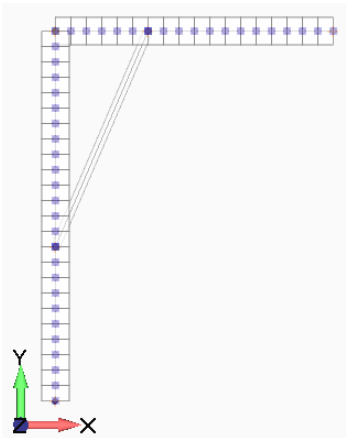


Figure 10.20. Coincident nodes

Releasing Beam Element Rotation
View/ Options (or hit F6 key)
In view Options dialog box
Category: Labels, Entities and Color
Options: Element – Directions, Pick Show Direction
Options: Element – Orientation/ Shape, then Element Shape: 0..Line/Plane Only
Options: Element – Beam Y-Axis, Pick Show Y Axis
Click OK

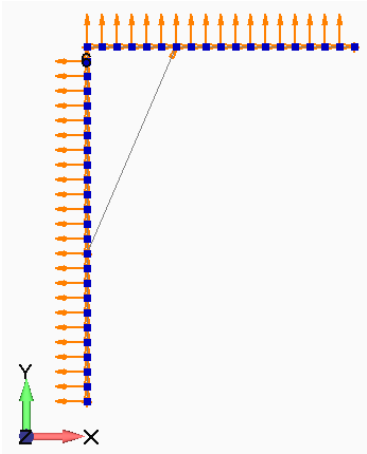


Figure 10.21. The local coordinate systems of the elements

Modify/ Update Elements/ Beam or Bar Releases
In Entity Selection – Select Element(s) to Update Releases
Pick the element 25 on the graphics window, then Click to OK
In Define Element Releases
End A: Select RZ, then click OK

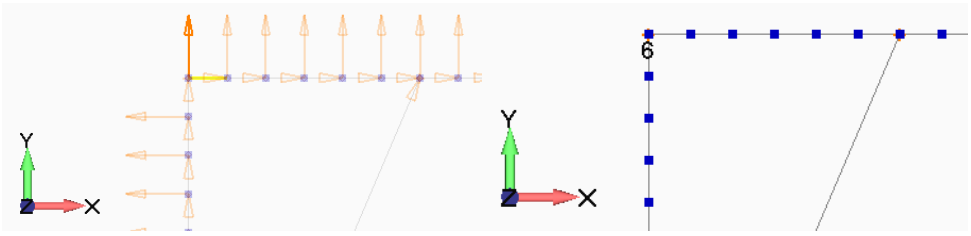


Figure 10.22. The element No 25

Applying Constraints
Model/ Constraint/ On Point
In New Constraint Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Point(s) to Select dialog box
Pick point 1, then click OK
In Create Constraint on Geometry Dialog box
Title: Support
Standard Types: Fixed
Click OK, then Cancel

Applying Load
Model/ Load/ On Point
In New Load Set dialog box
Title: (give a Title), then click OK
In Entity Selection – Enter Point(s) to Select dialog box
Pick point 5, then click to OK
In Create Loads on Points dialog box
Title: Loading
Select Force
Load Value: FY= -5000
Click OK, then Cancel

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click Next 8 times
In Nastran Output Requests dialog box
Uncheck: Applied Load
Check: Displacement, Constraint Force, Force and Stess
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the Results
View/ Rotate/ Modell (or hit F8 key)
In view rotate dialog box
Click to XY Top Button, then OK
Hit Ctrl+A key
View/Select (or hit F5 button)
In View Select dialog box
Deformed Style: Deform
Contour Style: Criteria
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Output Sets: NX NASTRAN Case 1
Deform: 1..Total Translation
Contour: 1..Total Translation
Click OK all dialob boxes
View/ Options
Category: Labels, Entities and Color
Options: Element – Orientation/ Shape, then Element Shape: 3..Show Cross Section
Category: PostProcessing
Options: Citeria – Elements that Pass
Label Mode: 0..No Label
Click OK

View/Select (or hit F5 button)
In View Select dialog box
Deformed Style: None – Model Only
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Deform: 1..Total Translation
Contour: 3164..Beam EndA Max Comb Stress
Click OK all dialob boxes

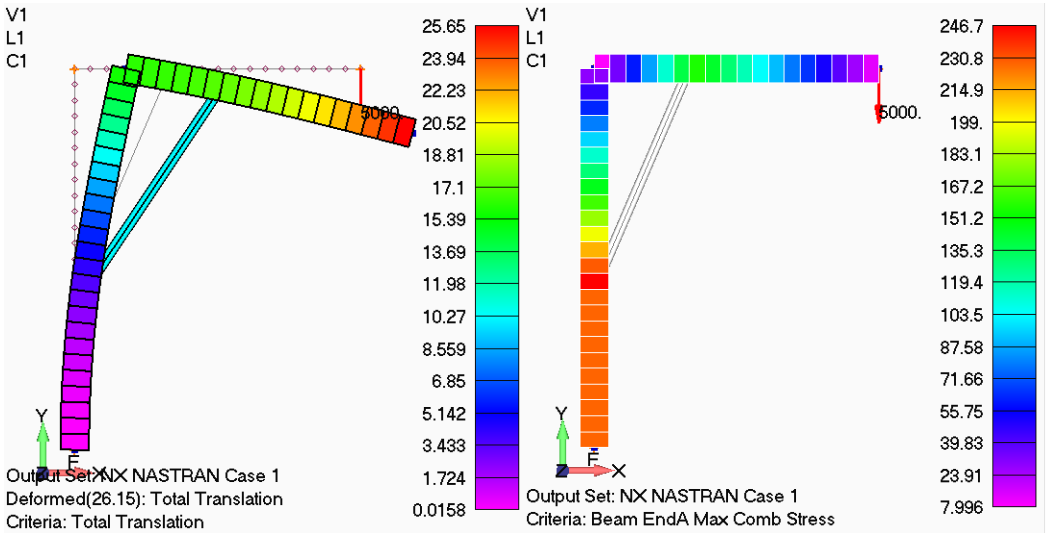


Figure 10.23. The deformation of the structure and the stress distribution

//The allowable stress for the material is 200MPa.//
View/ Options
In View Options dialog box
Category: PostProcessing
Options: Criteria Limits
Limits Mode: 1..Above Maximum
Minimum: 0; Maximum: 200
Click OK

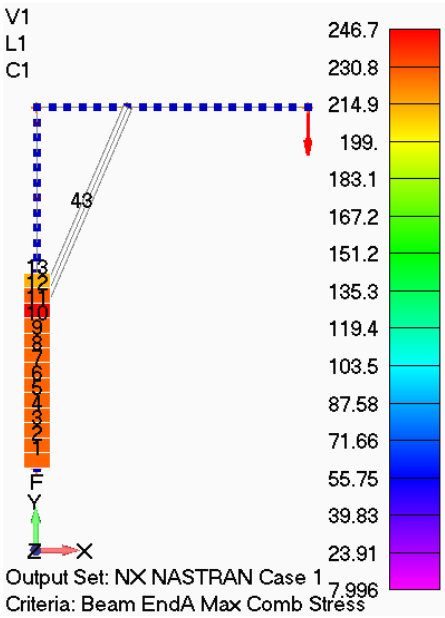


Figure 10.24. The dangerous structural elements

//Now the normal force is determined in the rod element.//
View/ Select
In View Select dialog box
Deformed Style: None – Model Only
Contour Style: Criteria
Click to Deformed and Contour Data… Button

In Select PostProcessing Data dialog box
Contour: 3036..Rod Axial Force
Click OK all dialob boxes

//The next step is the illustration of the moment diagram.//
View/ Select
In View Select dialog box
Deformed Style: None – Model Only
Contour Style: Beam Diagram
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Contour: 3014..Beam EndA Plane1 Moment
Click OK all dialog boxes

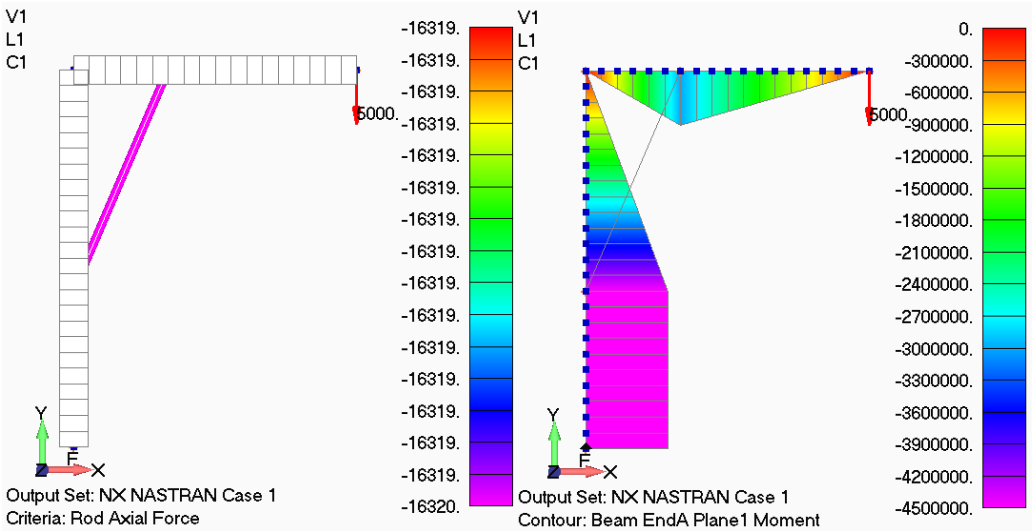


Figure 10.25. The normal force in the cylinder and the moment diagram of the frame

View/ Select
In View Select dialog box
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Contour: 3018..Beam EndA Pl1 Shear Force
Click OK all dialob boxes
View/ Select
In View Select dialog box
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Contour: 3022..Beam EndA Axial Force
Click OK all dialob boxes

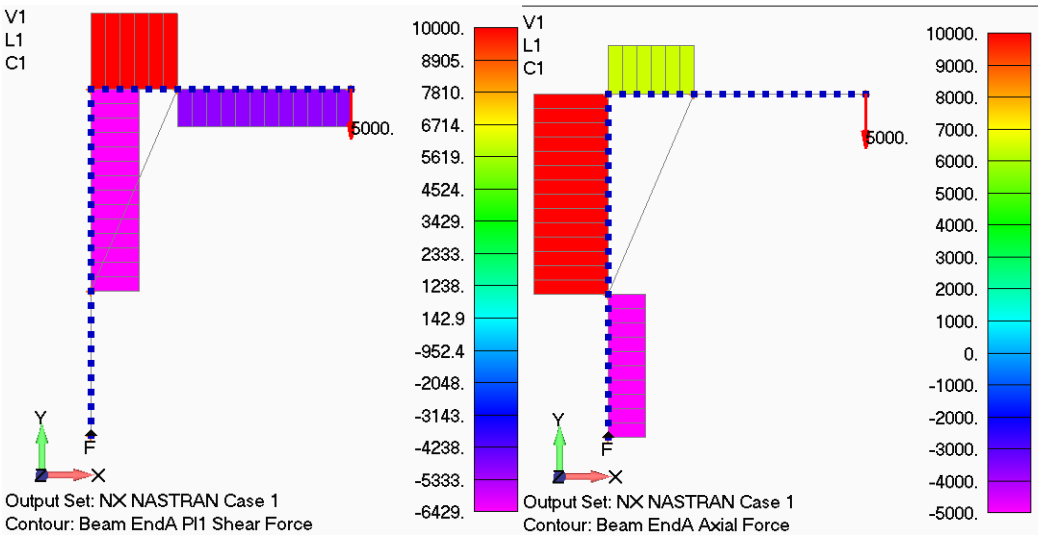


Figure 10.26. Shear force and normal force diagrams

View/ Select
In View Select dialog box
Contour Style: None – Model Only
Click OK
View/ Advenced Post/ Beam Cross Section
In Beam Cross Section Stress Control dialog box
Elements: Select Single
Pick Element 42 (the last element at the right side of the model)
Show Stress: Select 4..Axial Stress from the drop down list
In location section: Move the slider bar between End A and End B

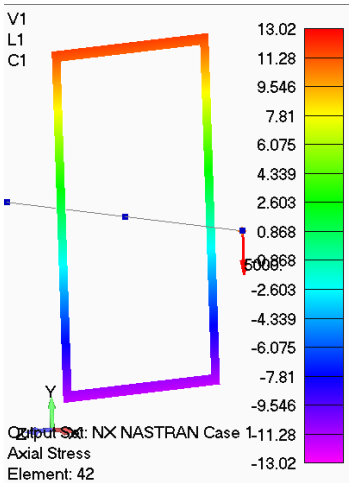


Figure 10.27. The normal stress distribution along the cross section

Select Screen Space
Show Stress: 0.. von Mises Stress
Click OK

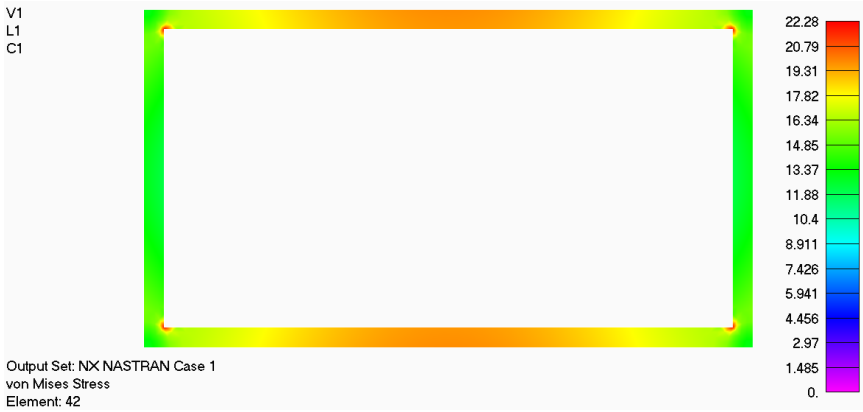


Figure 10.28. The equivalent stress distribution along the cross section

//List Forces on the Dangerous Cross-Section//
List/Output/Results to Data Table (If this is option is unavailable, Turn on it Tools/Data Table)
OK to Unlock Data Table? Click Yes
In Send Results to Data Table dialog box
Select Output Sets: in Columns; Output Vectors: in Columns; Nodes/Elements: in Rows
In Coordinate System (Nodal Output Only) Select 0..Basic Rectangular
In Results to Add to Data Table dialog box
Pick Outputs Sets: 1..NX NASTRAN Case 1
Pick Output Vectors: 3014..Beam EndA Plane 1 Moment, 3018..Beam endA Pl1Shear Force, 3022..Beam EndA Axial Force; then Click OK
In Entity Selection – Select Element(s) to Report
Pick to element 11 (above the dangerous cross-section)
Open the Data Table and you had to see the following results:

Element ID	Beam EndA Plane1 Moment	Beam EndA Plane1 Shear Force	Beam EndA Axial Force
11	-4500000 [*Nmm*]	-6428.571 [*N*]	10000 [*N*]

Table 10.4. Resultants on the dangerous cross section

Numerical example 6. (Plate with a hole)
An aluminum plate with a hole is subjected to tension.

Data:
Sides $a = 80mm, b = 30mm$.
Hole diameter $d = 20mm$.
The thickness of the plate: $t = 2mm$.
The Young's modulus: $E = 69\ GPa$.
Poisson ration: $\nu = 0.33$.
Load $p = 30\ N/mm^2$.

Defining model geometry
//Because of the symmetry in geometry and boundary conditions the key geometry is a quarter model of the plate with a hole.//
Geometry/Curve-Line/Project points
In Locate-Enter First Location for Projected Line dialog box set
X=10; Y=0; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=15; Y=0; Z=0
Click OK (curve 1 created)
In Locate-Enter First Location for Projected Line dialog box set
X=15; Y=0; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=15; Y=40; Z=0
Click OK (curve 2 created)
In Locate-Enter First Location for Projected Line dialog box set
X=15; Y=40; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=0; Y=40; Z=0
Click OK (curve 3 created)
In Locate-Enter First Location for Projected Line dialog box set
X=0; Y=40; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=0; Y=10; Z=0
Click OK (curve 4 created), then click Cancel.

Geometry/Curve-Arc/Center-Start-End
In Locate-Enter Location at Center of Arc dialog box set
X=0; Y=0; Z=0
Click OK
In Locate-Enter Location at Start of Arc dialog box set
X=10; Y=0; Z=0
Click OK
In Locate-Enter Location at End of Arc dialog box set
X=0; Y=10; Z=0
Click OK (curve 5 created), then click Cancel.

Geometry/Boundary Surface/From Curves
In Entity Selection-Select Curve(s) on Closed Boundary dialog box click
Select All, then click OK (boundary 1 created), then click Cancel.

Creating Material
Model/Material
In Define Material dialog box click
Type,
In Material Type dialog box choose Isotropic
Click OK
In Define Material-ISOTROPIC Dialog box set
Title=aluminum
Youngs Modulus, E=69000
Poisson's Ration, nu=0.33
Click OK (Material 1 created), then click Cancel.

Defining Property
Model/Property
In Define Property dialog box click
Elem/Property Type...,
In Element/Property Type dialog box choose Plane Strain
Click OK
In Define Property-PLANE STRAIN Element Type dialog box set
Thicknesses, Tavg or T1=2
Click OK (Property 1 created), then click Cancel.

Applying Load
Model/Load/On Curve
In Entity Selection-Enter Curve(s) to Select dialog box set
ID=3 to=3 by=1 (or pick curve 3)
Click OK
In Create Loads on Curves dialog box choose Force Per Length and set
FY=30
Click OK (Load 1 created), then click Cancel.

Constraining the Model
Model/Constraint/On Curve
In Entity Selection-Enter Curve(s) to Select dialog box set
ID=1 to=1 by=1 (or pick curve 1)
Click OK
In Create Constraints on Geometry dialog box choose Arbitrary in CSyS and choose TY
Click OK
In Entity Selection Enter Curve(s) to Select dialog box set
ID=4 to=4 by=1 (or pick curve 4)
Click OK
In Create Constraints on Geometry dialog box choose Arbitrary in CSyS and choose TX
Click OK, then click Cancel.

Meshing the Model
Mesh/Mesh Control/Default Size
In Default Mesh Size dialog box set
Size=1
Click OK
Mesh/Geometry/Surface
In Entity Selection-Select Surfaces to Mesh dialog box click
Select All, then click OK
In Automesh Surfaces dialog box set Property to 1..PLAIN STRAIN Property
Click OK.

The graphic window should look like this:

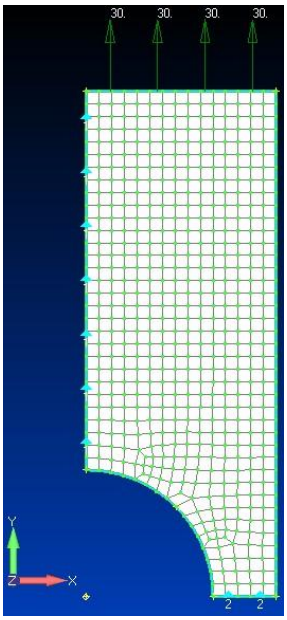


Figure 10.29. The finite element model of the plate with hole

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click Next 8 times
In Nastran Output Requests dialog box
Uncheck Applied Load, Constraint Force, Force and Check Strain
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the results

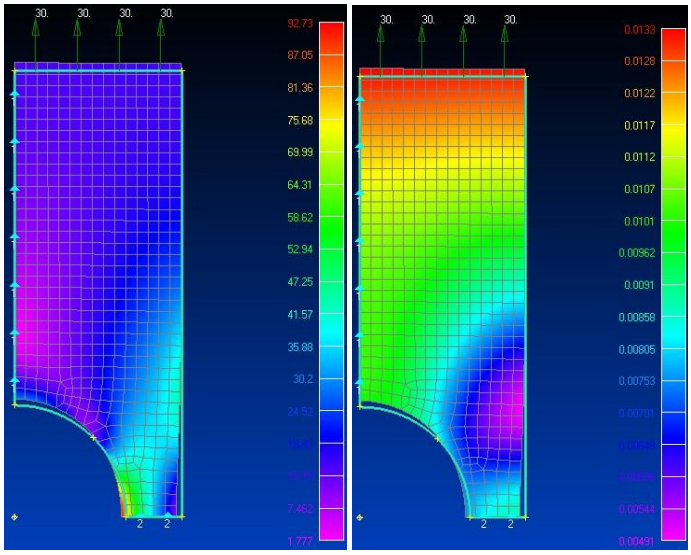


Figure 10.30. The calculated stress and total translation

Numerical example 7. (Tube under internal pressure)
The mechanical model of a tube with thick walls is shown in Figure 10.31. The tube is subjected to internal pressure. The axial deformation of the cylinder is constrained. The finite element model consists of 8 finite elements with 4 nodes per element. All radial displacements are free.

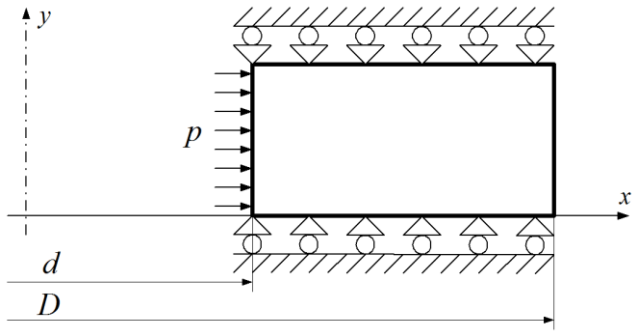


Figure 10.31. The mechanical model of the tube loaded by internal pressure

<u>Data:</u>
Outer diameter $D = 800mm$
Inner diameter $d = 600mm$
The Young's modulus: $E = 210\ GPa$.
Poisson ration: $\nu = 0.3$.
Internal pressure $p = 100bar = 10MPa$.

Defining model geometry
Geometry/Curve-Line/Project points
In Locate-Enter First Location for Projected Line dialog box set
X=300; Y=0; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=400; Y=0; Z=0
Click OK (curve 1 created)
In Locate-Enter First Location for Projected Line dialog box set
X=400; Y=0; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=400; Y=50; Z=0
Click OK (curve 2 created)
In Locate-Enter First Location for Projected Line dialog box set
X=400; Y=50; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=300; Y=50; Z=0
Click OK (curve 3 created)
In Locate-Enter First Location for Projected Line dialog box set
X=300; Y=50; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=300; Y=0; Z=0
Click OK (curve 4 created), then click Cancel.

Geometry/Boundary Surface/From Curves
In Entity Selection-Select Curve(s) on Closed Boundary dialog box click
Select All, then click OK (Boundary 1 created), then click Cancel.

Creating Material
Model/Material
In Define Material dialog box click
Type,
In Material Type dialog box choose Isotropic
Click OK
In Define Material-ISOTROPIC Dialog box set
Title=steel
Youngs Modulus, E=210000
Poisson's Ration, nu=0.3
Click OK (Material 1 created), then click Cancel.

Defining Property
Model/Property
In Define Property dialog box click
Elem/Property Type...,
In Element/Property Type dialog box choose Axisymmetric
Click OK
OK to show Axisymmetric Axis in all Views?
Click Yes
In Define Property-AXISYMMETRIC Element Type dialog box set
Title=axisymmetric

Material, choose 1..steel
Click OK (Property 1 created), then click Cancel.

Meshing the Model
Mesh/Mesh Control/Size along curve
In Entity Selection-Select Curve(s) to Set Mesh Size dialog box set
Select Curve 1 and Curve 3
Click OK
In Mesh Size Along Curves dialog box set
Number of Elements box, set 8
Click OK
In Entity Selection-Select Curve(s) to Set Mesh Size dialog box set
Select Curve 2 and Curve 4
Click OK
In Mesh Size Along Curves dialog box set
Number of Elements box, set 1
Click OK, then click Cancel.
Mesh/Geometry/Surface
In Entity Selection-Select Surfaces to Mesh dialog box click
Select All, then click OK
In Automesh Surfaces dialog box set Property to 1..axisymmetric Property
Click OK.

Applying Load
Model/Load/Elemental
In New Load Set dialog box set
Title: loading
Click OK
In Entity Selection-Enter Element(s) to Select dialog box set
Pick element 1
Click OK
In Create Loads on Elements dialog box choose Pressure and set
Pressure=10
Click OK
In Face Selection dialog box set
Pick Face 3
Click OK (Load 1 created), then click Cancel.

Constraining the Model
Model/Constraint/Create/Manage Set
In Constraint Set Manager dialog box set
New Constraint Set…
Title: constraints
Click OK
Click Done.
Modify/Update Other/Perm Constraint…
In Entity Selection-Select Node(s) to Update Permanent Constraints dialog box set
Select All, then click OK
In Update Nodal Permament Constraints dialog box pick
TY, TZ, RX, RY, RZ, then click OK

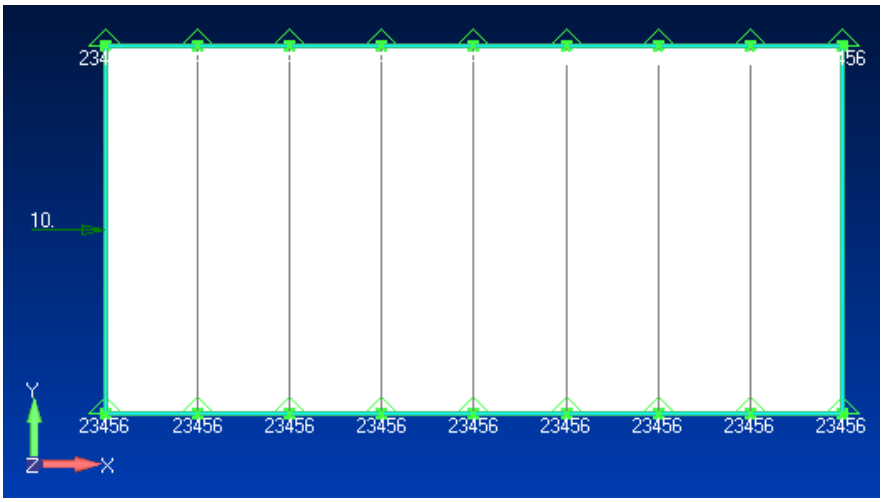


Figure 10.32. The finite element model of the tube

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box

Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click Next 8 times
Click OK, then analyze (In Analysis Set Manager dialog box)
OK to Flip Model?
Click Yes
OK to force all element normals to lie along the correct global axis?
Click Yes
In Scale Factor for Axisym Forces dialog box set
Factor=1, then click OK
OK to Save Model Now?
Click Yes
close the NX Nastran Analysis Monitor

Postprocessing the results
View/Rotate/Model
In View Rotate dialog box pick
ZX Front, then click OK
//Now we represent the total translation then the stress distribution of the cylinder.//
View/Select
In View Select dialog box set
Deform Style: Deform
Contour Style: Contour
Click Deformed and Contour Data…
In Select PostProcessing Data dialog box set
Output Set: 1..NX NASTRAN Case 1
Output Vectors
Deformation: 1..Total Translation
Contour: 1.. Total Translation
Click OK
Click OK
View/Select
In View Select dialog box set
Deform Style: Deform
Contour Style: Contour
Click Deformed and Contour Data…
In Select PostProcessing Data dialog box set
Output Set: 1..NX NASTRAN Case 1
Output Vectors
Deformation: 1..Total Translation
Contour: 6035.. Axisym Von Mises Stress
Click OK
Click OK

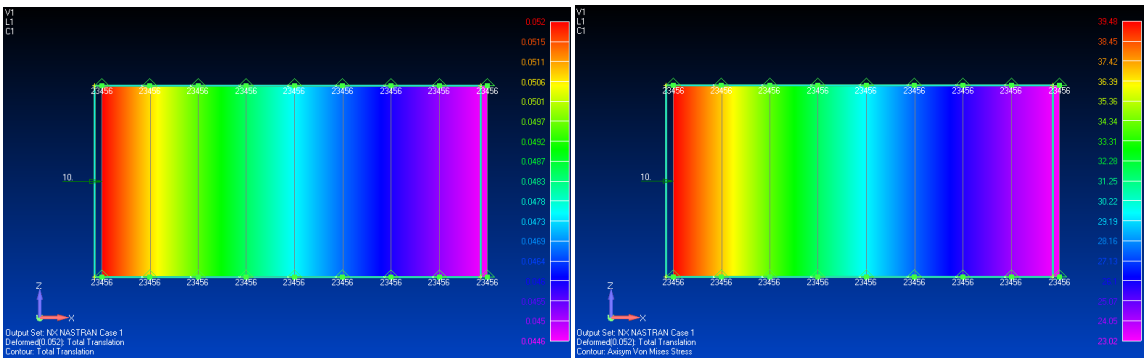


Figure 10.33. Total translation and the stress distribution of the tube

ID	1..NX NASTRAN Case 1, 1..Total Translation
1	0.05198631 [*mm*]
2	0.05198631 [*mm*]
3	0.05068693 [*mm*]
4	0.04951809 [*mm*]
5	0.04846529 [*mm*]
6	0.04751609 [*mm*]
7	0.04665979 [*mm*]
8	0.04588709 [*mm*]
9	0.04518991 [*mm*]
10	0.04456116 [*mm*]
11	0.04456116 [*mm*]
12	0.04518991 [*mm*]
13	0.04588709 [*mm*]
14	0.04665979 [*mm*]
15	0.04751609 [*mm*]
16	0.04846529 [*mm*]
17	0.04951809 [*mm*]
18	0.05068693 [*mm*]

Table 10.5. Total translation values at nodes

ID	1..NX NASTRAN Case 1, 6028..Axisym Radial Stress	1..NX NASTRAN Case 1, 6029..Axisym Azimuth Stress	1..NX NASTRAN Case 1, 6035..Axisym Von Mises Stress
1	-9.076963 [*MPa*]	34.79367 [*MPa*]	38.33964 [*MPa*]
2	-7.390267 [*MPa*]	33.10605 [*MPa*]	35.44596 [*MPa*]
3	-5.890926 [*MPa*]	31.60596 [*MPa*]	32.878 [*MPa*]
4	-4.552184 [*MPa*]	30.26659 [*MPa*]	30.58937 [*MPa*]
5	-3.351897 [*MPa*]	29.06578 [*MPa*]	28.54168 [*MPa*]
6	-2.271611 [*MPa*]	27.98506 [*MPa*]	26.70294 [*MPa*]
7	-1.295845 [*MPa*]	27.00893 [*MPa*]	25.04629 [*MPa*]
8	-0.4115381 [*MPa*]	26.12431 [*MPa*]	23.54908 [*MPa*]

Table 10.6. Stress values on the elements

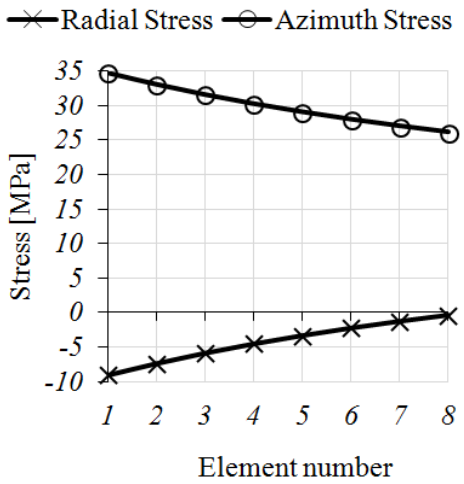


Figure 10.34. The “tube chart”

Numerical example 8. (Analyzing a pressure vessel with axysimmetric elements)
The pressure vessel is subjected to internal pressure, see in Figure 10.35. The aim is to determine the displacements and the stresses.

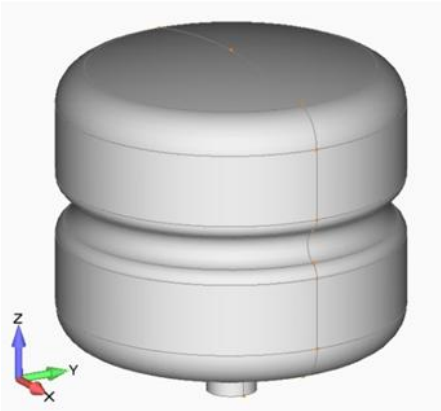


Figure 10.35. The 3D model of the pressure vessel

<u>Data:</u>
Internal pressure: $p = 20\ bar$
Thickness: $v = 10\ mm$
$E = 69000\ MPa$
$\nu = 0{,}33$

Importing the Model Geometry
File/Import/ Geometry
In Geometry File to Import dialog box:
Go to the Examples directory and open pressure_vessel.stp file
In STEP Read Options dialog box:
Check the Geometry Scale Factor: 1000
Click OK
//When importing CAD geometry from a CAD software it is important to check the sizes of the geometry. Now the thickness of the vessel is checked.//
Tools/Measure/Distance
In Locate – Define Location to Measure From dialog box
Click Methods Button, and Select On Point (or press CTRL+P key when the cursor in the X,Y or Z field)
On Point – Define Location to Measure From dialog box
Point ID: 32, or Pick the point 1 on the model, Click OK
On Point – Define Location to Measure From dialog box
Point ID: 36, or Pick the point 2 on the model, Click OK then Cancel
//The measured distance data can be read in the Message window. Distance: 10.//

Preparing the Model to Mesh
//If the geometry of the structure and the boundary conditions are axisymmetric then the structure has to be modeled using its meridian section.//
Geometry/ Solid/ Slice
In Entity Selection – Select Solid to Slice dialog box
Pick the solid part on the graphics window, then Click OK
In Plane Locate - Specify Plane for Intersection dialog box
Click Methods Button and Select Points

In Plane Points –Specify Plane for Intersection dialog box
Select 3 points on the modell which is lying on the XZ plane
Base Point ID: 50; Plane Point 1: 49; Plane Point 2: 48, Click OK then Cancel

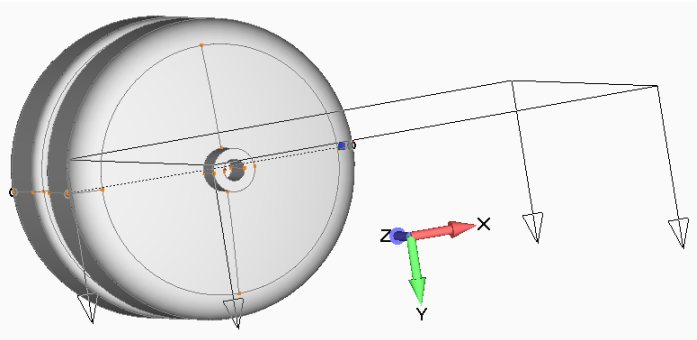


Figure 10.36. Creating thee meridian section from the 3D model

View/ Visibility (or hit CTRL+Q key)
In Visibility dialog box Select Geometry tab, Uncheck 1..Geometry, then Click Done
Geometry/ Solid/ Slice
In Entity Selection – Select Solid to Slice dialog box
Pick the solid part on the graphics window, then Click OK
In Plane Points –Specify Plane for Intersection box
Click Methods Button and Select Global Plane
In Global Plane –Specify Plane for Intersection box
Base: X=0; Y=0; Z=0 and Select YZ Plane,
Click preview (you can checking the cutting plane), Then Click OK
View/ Visibility
In Visibility dialog box Select Geometry tab, Uncheck 2..Untitled Geometry, then Click Done
Geometry/ Surface/Offset
In Entity Selection – Select Surfaces to Offset
Click Pick Button and select Front
Pick the surface 51 on the graphics window, and Click OK
In Offset Surface dialog box
Enter offset Value: 0, and Click OK, then Cancel
View/ Visibility
In Visibility dialog box Select Geometry tab, Uncheck 3..Untitled Geometry, then Click Done
View/ Rotate/ Model (or hit F8 Button)
In View Rotate dialog box Click to ZX Front Button, then Click OK

The graphic window should look like this:

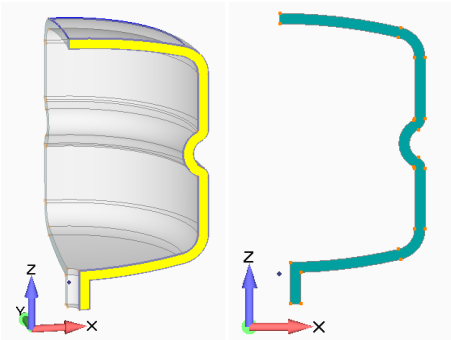


Figure 10.37. The meridian section of the pressure vessel

Defining the Material and Property
//Before meshing it is necessary to define the element property and the material. The defined property contains the type of the chosen finite element and also the assigned constitutive law.//

Defining the Material
//The material of the structure is S235 steel which is a linear, isotropic and homogeneous material. The Hooke's law is valid so it is enough to give the Young's modulus and the Poisson ratio.//
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set
Title: Aluminum
Young Modulus, E: 6,9E4
Poisson's Ratio, nu: 0,33
Click OK (material 1 created), then Cancel

Defining the Property
// After defining the material the property has to be given choosing axisymmetric volume elements.//
Model/Property
In Define Property – PLATE Element Type dialog box
Click to Elem/Property Type Button
In Element/Property Type dialog box choose Axysymmetric (Volume Elements)
Click OK
Ok to show Axisymmetric Axis in all Views? Click to Yes
In Define Property – AXYSIMMETRIC Element Type dialog box
Give a Title such as: Axisymmetric Aluminum
Material: 1..Aluminum
Click to OK, then Cancel
View/ Options (F6 key)
In View Options dialog box
Category: Tools and View Style
Options: Axisymmetric Axes
Direction: 4..Global Z, X Radial
Color/ Draw Mode: 5..RGB Solid
Click OK

Meshing the Model
Mesh/Mesh control/Default Size
In Default Mesh Size dialog box
Element Size: 2
Click OK

View/ Options
In View Options dialog box

Category: Labels, Entities and Color
Options: Curve – Mesh Size
Show As: 2..Symbols (all curves)
Click OK

Mesh/ Mesh Control/ Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Pick curve 209 and 211 on the graphics window, then Click OK
In Mesh Size Along Curve dialog box
Element Size: 0,5
Click OK, then Cancel

Mesh/Geometry/Surface
In Entity Selection – Select Surfaces to Mesh dialog box
Select the meridian section's surface in the graphics window, then click OK
In Automesh Surfaces dialog box Set
Property: 1..Axysimmetric Aluminum
Mesher: Triangles
Click OK
View/ Visibility
In Visibility dialog box Select Entity/ Label tab
Select Labels, Click to All off Button
Select Draw Entity, Click to All off Button, then Select Mesh/ Element
Click Done

The graphic window should look like this:

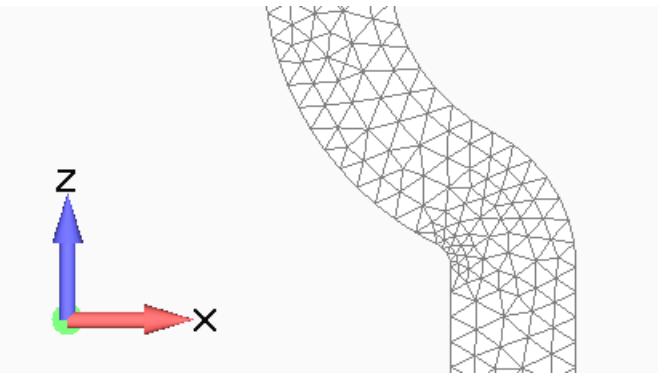


Figure 10.38. The meshed model

Applying Constraints
Model/ Constraint/ On Curve
In New Constraint Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Curve(s) to Select dialog box
Choose the curve 204 on the bottom of the part, then click OK
In Create Constraints on Geometry dialog box
Title: Support
Select Arbitrary in CSys and 0..Basic Rectangular from the drop down list
Check TZ degrees of freedom
Click Ok, then Cancel

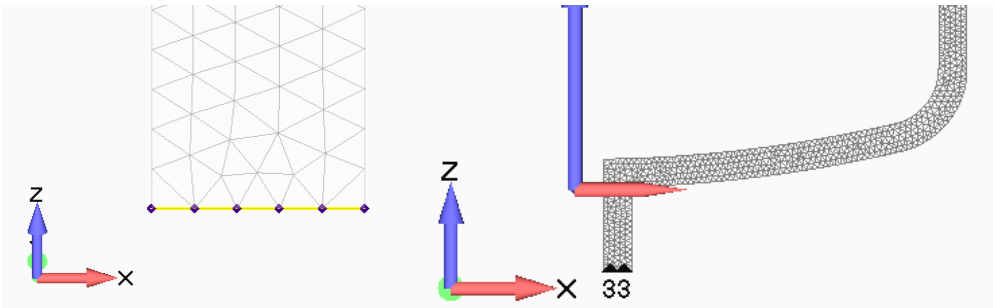


Figure 10.39. Kinematical boundary conditions definition

Applying Load

//The loading of the pressure vessel is internal pressure.//
Model/ Load/ Elemental
In New Load Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Element(s) to Select dialog box
Click to Select All Button, then click OK
In Create Loads on Elements dialog box
Title: Belső Nyomás; Choose Pressure
Direction: Normal to Element Face (The direction of this vector will always shown out of the material)
Set Pressure value to: 2 (in MPa dimension)
Click OK
In Face Selection dialog box
Method: Adjacent Faces
Click in Face field, and pick an element face on the inside of the vessel
Tolerance: 89
Select Front Face, then Click OK

In Entity Selection – Enter Element(s) to Select dialog box
Click to Select All Button, then click OK
In Create Loads on Elements dialog box
Title: Internal pressure; Choose Pressure
Set Pressure value to: 2
Click OK
In Face Selection dialog box
Method: Adjacent Faces
Click in Face field, and pick an element according to the Figure 10.40.
Tolerance: 20
Click OK, then Cancel

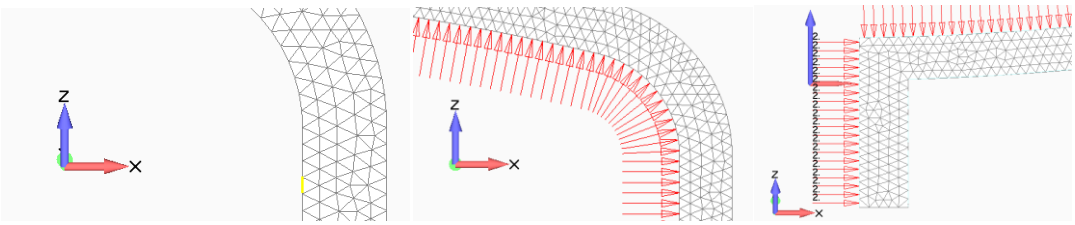


Figure 10.40. Load definition on the pressure vessel

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click Next 8 times
In Nastran Output Requests dialog box
Uncheck Applied Load, Constraint Force, Force nad Check Strain
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the Results
//The important data from the finite element analysis are now the nodal displacements and the equivalent stress distribution. Now the results are animated.//
View/ Rotate/ Modell (or hit F8 key)
In view rotate dialog box
Click ZX Front Button, then OK
View/Select (or hit F5 button)
In View Select dialog box
Deformed Style: Deform
Contour Style: Contour
Click to Deformed and Contour Data… Button
In Select PostProcessing Data dialog box
Output Sets: NX NASTRAN Case 1
Deform: 1..Total Translation
Contour: 1..Total Translation
Click OK all dialob boxes
Click Post Options and Select Actual Deformation
Select Scale Deformation
In Deformation Scale dialog box
Actual: 10, Click OK

Post Options on the toolbar
View/Select (or hit F5 button)
In View Select dialog box
Click to Deformed and Contour Data
In Select PostProcessing Data dialog box
Deform: 1..Total Translation
Contour: 6035..Axisym Von Mises Stress
Click OK all dialob boxes

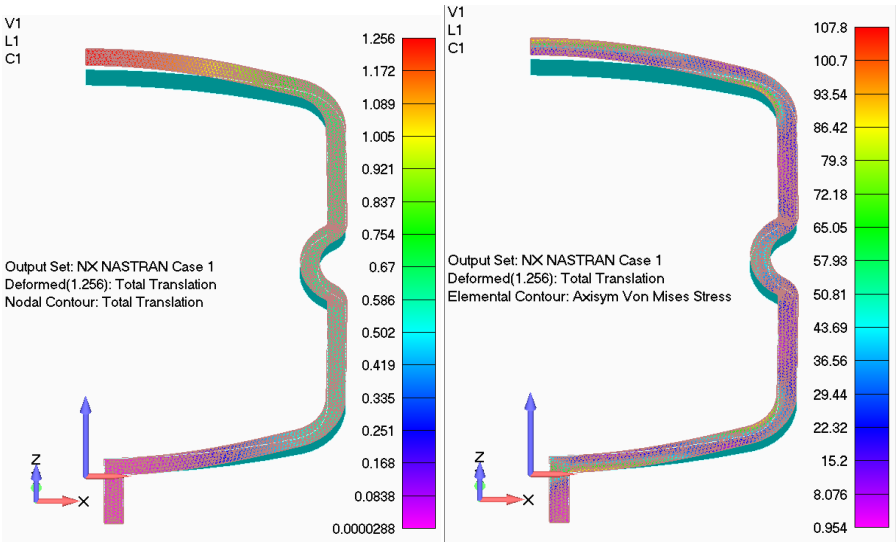


Figure 10.41. The total translation and the stress distribution of the pressure vessel

//Animation of the deformation.//
View/Select (or hit F5 button)
In View Select dialog box
Deformed Style: Animate
Contour Style: Contour
Click OK and then you had to see the deformation animation about the vessel load and unload
Some other recommended options for animation settings:
Click Post Options and Select Animation – Positive only
Click Animation Frames
In animation Frames dialog box
Frames: 20, then Click OK

//Listing the maximum stress values//
List/Output/Results to Data Table (If this option is unavailable, Turn on it Tools/Data Table)
OK to Unlock Data Table? Click Yes
In Send Results to Data Table dialog box
Select Output Sets: in Columns; Output Vectors: in Columns; Nodes/Elements: in Rows
In Coordinate System (Nodal Output Only) Select 0..Basic Rectangular
In Results to Add to Data Table dialog box
Pick Outputs Sets: 1..NX NASTRAN Case 1
Pick Output Vectors: 6035..Axisym Von Mises Stress; Click OK
In Entity Selection – Select element(s) to Report dialog box
Choose Select All Button, then Click OK
Open the Data Table and sort in decreasing order the value of 6035..Axisym Von Mises Stress

dangerous elements ID	1..NX NASTRAN Case 1, 6035..Axisym Von Mises Stress
523	119,2865 [*MPa*]
920	108,7452 [*MPa*]
925	108,2656 [*MPa*]
921	106,7549 [*MPa*]
56	106,5607 [*MPa*]

Table 10.7. The five most dangerous elements' ID and the resultant stresses

Numerical example 9. (Simply supported plate loaded by pressure)
A simply supported plate is subjected to distributed pressure. The mechanical model of the plate is shown in Figure 10.42.

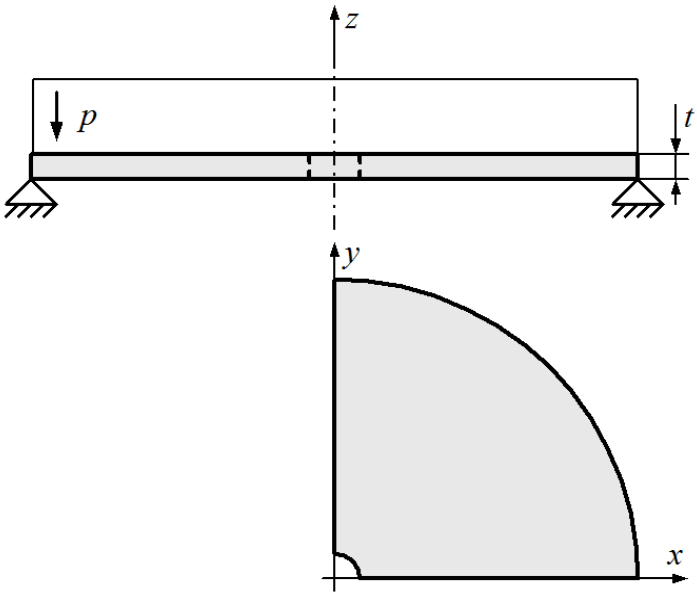


Figure 10.42. The mechanical model of the plate

<u>Data:</u>
Outer diameter: $D = 1000mm$
Inner diameter: $d = 100mm$
Plate thickness: $t = 50mm$
$E = 69000MPa$
$\nu = 0.33$
Pressure: $p = 6bar$

Defining model geometry
//Because of the symmetry in geometry and boundary conditions the key geometry is a quarter model of the plate with a hole.//
Geometry/Curve – Line/Project points
In Locate-Enter First Location for Projected Line dialog box set
X=50; Y=0; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=500; Y=0; Z=0
Click OK (curve 1 created), then click Cancel
Geometry/Curve – Arc/Center – Start – End
In Locate–Enter Location at Center of Arc dialog box set
X=0; Y=0; Z=0
Click OK
In Locate-Enter Location at Start of Arc dialog box set
X=500; Y=0; Z=0
Click OK
In Locate-Enter Location at End of Arc dialog box set
X=0; Y=500; Z=0
Click OK (curve 2 created), then click Cancel

Geometry/Curve – Line/Project points
In Locate-Enter First Location for Projected Line dialog box set
X=0; Y=500; Z=0
Click OK
In Locate-Enter Second Location for Projected Line dialog box set
X=0; Y=50; Z=0
Click OK (curve 3 created), then click Cancel
Geometry/Curve – Arc/Center – Start – End
In Locate–Enter Location at Center of Arc dialog box set
X=0; Y=0; Z=0
Click OK
In Locate-Enter Location at Start of Arc dialog box set
X=50; Y=0; Z=0
Click OK
In Locate-Enter Location at End of Arc dialog box set
X=0; Y=50; Z=0
Click OK (curve 4 created), then click Cancel
Geometry/Boundary Surface/From Curves
In Entity Selection – Select Curve(s) on Closed Boundary dialog box pick
Select All, click OK (boundary 1 created), then click Cancel.

Creating Material
Model/Material
In Define Material dialog box click
Type,
In Material Type dialog box choose Isotropic
Click OK
In Define Material-ISOTROPIC Dialog box set
Title=aluminum
Youngs Modulus, E=69000
Poisson's Ration, nu=0.33
Click OK (Material 1 created), then click Cancel.

Defining Property
Model/Property
In Define Property – PLATE Element Type dialog box set
Title: plate
Thicknesses, Tavg or T1=50
Material: 1..aluminum
Click OK (Property 1 created), then click Cancel

Meshing the Model
Mesh/Mesh Control/Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box pick
curve 1, then click OK
In Mesh Size Along Curve dialog box set
Number of Elements=25
Node Spacing: Biased
Bias Factor=5
Small Elements at Start

Click OK
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box pick curve 3, then click OK
In Mesh Size Along Curve dialog box set
Number of Elements=25
Node Spacing: Biased
Bias Factor=5
Small Elements at End
Click OK
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box pick curve 2, then click OK
In Mesh Size Along Curve dialog box set
Number of Elements=25
Node Spacing: Equal
Click OK
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box pick curve 4, then click OK
In Mesh Size Along Curve dialog box set
Number of Elements=10
Node Spacing: Equal
Click OK, then click Cancel

Mesh/Geometry/Surface
In Entity Selection – Select Surfaces to Mesh dialog box pick
Select All, then click OK
In Automesh Surfaces dialog box set
Property: 1..plate
Click OK

Applying Load
Model/Load/On Surface
In New Load Set dialog box set
Title: Loading
Click OK
In Entity Selection – Enter Surface(s) to Select dialog box pick
Select All
Click OK
In Create Loads on Surface dialog box pick
Pressure
Pressure=0.6
Click OK, then click Cancel

Constraining the Model
Model/Constraint Nodal
In New Constraint Set dialog box set
Title: Support
Click OK
In Entity Selection – Enter Node(s) to Select dialog box pick
Method – on Curve
Select curve 1, click OK

In Create Nodal Constraints/DOF dialog box pick
TY; RX
Click OK
In Entity Selection – Enter Node(s) to Select dialog box pick
Method – on Curve
Select curve 3, click OK
In Create Nodal Constraints/DOF dialog box pick
TX; RY
Click OK
In Entity Selection – Enter Node(s) to Select dialog box pick
Method – on Curve
Select curve 2, click OK
In Create Nodal Constraints/DOF dialog box pick
Pinned
Click OK
Selected Constraints Already Exists. OK to Overwrite (NO=Combine)? No
Click Cancel
Modify/Update Other/Perm Constraint
In Entity Selection – Select Node(s) to Update Permanent Constraints dialog box pick
Select All
In Update Nodal Permanent Constraints dialog box set
RZ
Click OK

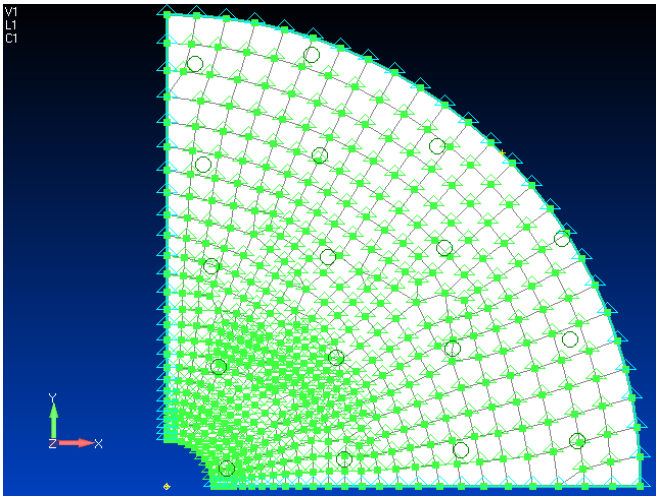


Figure 10.43. Finite element model of the plate

Analyzing the Model
//Using the NX Nastran solver linear static analysis is executed.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click Next 8 times
Click OK, then analyze (In Analysis Set Manager dialog box)

Postprocessing the results
View/Select
In View Select dialog box set
Deform Style: Deform
Contour Style: Contour
Click Deformed and Contour Data…
In Select PostProcessing Data dialog box set
Output Set: 1..NX NASTRAN Case 1
Output Vectors
Deformation: 1..Total Translation
Contour: 1..Total Translation
Click OK
Click OK
View/Select
In View Select dialog box set
Deform Style: Deform
Contour Style: Contour
Click Deformed and Contour Data…
In Select PostProcessing Data dialog box set
Output Set: 1..NX NASTRAN Case 1
Output Vectors
Deformation: 1..Total Translation
Contour: 6035.. Axisym Von Mises Stress
Click OK
Click OK

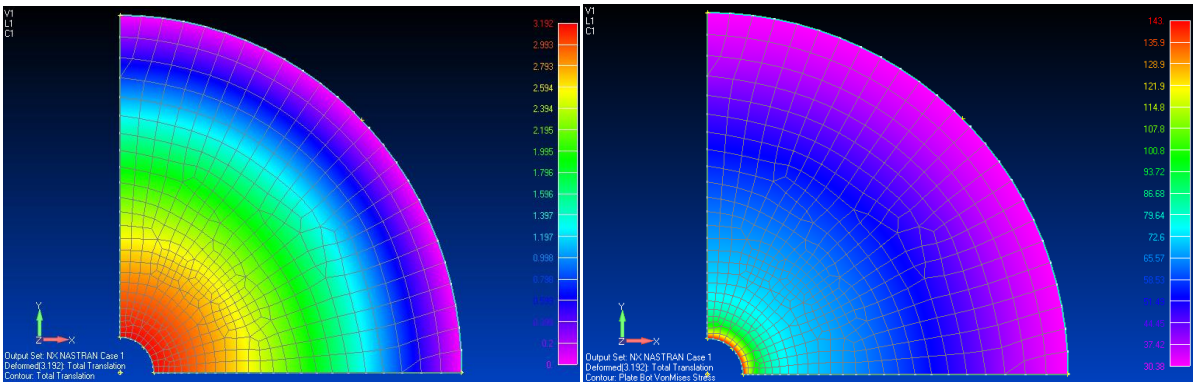


Figure 10.44. Total translation and stress distribution of the plate

Unlock Data Table in toolbar
List/Output/Results to Data Table
In Send Results to Data Table dialog box set
Output Selection: Elemental
Click OK
In Results to Add to Data Table dialog box pick
1..NX NASTRAN Case 1
Output Vectors (from Output Set 1): 7030—Plate Top Mean Stress
Output Vectors (from Output Set 1): 7031—Plate Top MaxShear Stress
Output Vectors (from Output Set 1): 7033—Plate Top VonMises Stress
Click OK

In Entity Selection-Select Element(s) to Report dialog box pick
Method – on Curve
Select curve 1
Click OK

ID	1..NX NASTRAN Case 1, 7030..Plate Top Mean Stress	1..NX NASTRAN Case 1, 7031..Plate Top MaxShear Stress	1..NX NASTRAN Case 1, 7033..Plate Top VonMises Stress
413	-53.4458 [*MPa*]	7.090441 [*MPa*]	54.83864 [*MPa*]
416	-56.42203 [*MPa*]	6.654825 [*MPa*]	57.58737 [*MPa*]
417	-50.07182 [*MPa*]	7.683464 [*MPa*]	51.81017 [*MPa*]
420	-59.01939 [*MPa*]	6.375372 [*MPa*]	60.04352 [*MPa*]
422	-61.2808 [*MPa*]	6.260712 [*MPa*]	62.23284 [*MPa*]
424	-63.26665 [*MPa*]	6.35216 [*MPa*]	64.21619 [*MPa*]
426	-64.99482 [*MPa*]	6.685522 [*MPa*]	66.0183 [*MPa*]
428	-66.44609 [*MPa*]	7.235489 [*MPa*]	67.6176 [*MPa*]
430	-67.66678 [*MPa*]	8.077028 [*MPa*]	69.09782 [*MPa*]
447	-30.81892 [*MPa*]	11.9621 [*MPa*]	37.13599 [*MPa*]
448	-37.15897 [*MPa*]	10.36846 [*MPa*]	41.27111 [*MPa*]
449	-41.98561 [*MPa*]	9.342827 [*MPa*]	44.99619 [*MPa*]
453	-25.63874 [*MPa*]	13.23143 [*MPa*]	34.38832 [*MPa*]
458	-18.83364 [*MPa*]	14.73518 [*MPa*]	31.71881 [*MPa*]
460	-46.26427 [*MPa*]	8.434076 [*MPa*]	48.51581 [*MPa*]
475	-68.69603 [*MPa*]	9.282112 [*MPa*]	70.55223 [*MPa*]
478	-69.54889 [*MPa*]	10.95035 [*MPa*]	72.08869 [*MPa*]
481	-70.25391 [*MPa*]	13.19449 [*MPa*]	73.87758 [*MPa*]
482	-70.79776 [*MPa*]	16.14915 [*MPa*]	76.12298 [*MPa*]
483	-71.22256 [*MPa*]	19.99701 [*MPa*]	79.19782 [*MPa*]
484	-71.52769 [*MPa*]	24.88964 [*MPa*]	83.51463 [*MPa*]
485	-71.62747 [*MPa*]	31.00818 [*MPa*]	89.52663 [*MPa*]
486	-72.0363 [*MPa*]	39.60439 [*MPa*]	99.47237 [*MPa*]
487	-72.01177 [*MPa*]	50.28646 [*MPa*]	113.0127 [*MPa*]
499	-71.91799 [*MPa*]	63.6239 [*MPa*]	131.591 [*MPa*]

Table 10.8. Stress values on elements along curve 1

Numerical example 10. (Analyzing a sheet shape specimen with plate elements)
In Figure 10.45. a sheet shaped specimen can be seen. A tensile test is analyzed when the two ends of the specimen is fixed. The task is to construct the finite element model and evaluate the stress-strain curve in linear sector. The total translation and the stress distribution are also evaluated and illustrated.

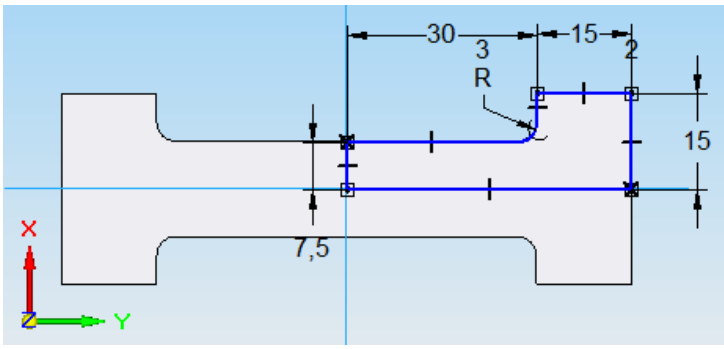


Figure 10.45. The geometry of the specimen

<u>Data:</u>
Thickness $t = 2\ mm$
Displacement steps: $\Delta v = 0{,}2\ mm$

Defining model Geometry:
//Because of the symmetry in geometry and boundary conditions the key geometry is a quarter model.//
Geometry/Point
In Locate - Enter Coordinates or Select with Cursor dialog box set
X=0; Y=0; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=7,5; Y=0; Z=0, then Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=7,5; Y=30; Z=0, Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=15; Y=30; Z=0, Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=15; Y=45; Z=0, Click OK
In Locate- Enter Coordinates or Select with Cursor dialog box set
X=0; Y=45; Z=0, Click OK then Cancel
Hit Ctrl+A key to Autoscale the graphics window
Geometry/Curve-Line/Points…
In Create Line from Points dialog box
Pick point 1 then point 2
Click OK (curve 1 created)
In Create Line from Points dialog box
Pick point 2 then point 3
Click OK (curve 2 created)
In Create Line from Points dialog box
Pick point 3 then point 4
Click OK (curve 3 created)
In Create Line from Points dialog box
Pick point 4 then point 5

Click OK (curve 4 created)
In Create Line from Points dialog box
Pick point 5 then point 6
Click OK (curve 5 created)
In Create Line from Points dialog box
Pick point 6 then point 1
Click OK (curve 6 created), then CANCEL
Modify/ Fillet
In Fillet Curves dialog box
Curve 1: 2; Curve 2: 3; Radius: 3
With Center Near: Click in X Field then click on the center of the fillet on the screen
Click OK, then Cancel
Geometry/ Boundary Surfaces/ From Curves
In Entity Selection - Select Curve(s) on closed Boundary dialog box Click to
Select All Button, then Ok

Defining the Material
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set
Title: S235 Steel
Young Modulus, E: 2,06E5
Poisson's Ratio, nu: 0,3
Click OK (material 1 created), then Cancel

Defining the Property
In Define Property – PLATE Element Type dialog box
Click Elem/Property Type
Give a Title such as: S235 PLATE
Material: 1..S235 Steel
Thicknesses, Tavg or T1: 2
Click OK (Property 1 created), then Cancel

Meshing the Model
//We have to take into consideration that high stress values might be found near the fillet so fine mesh is used at that region.//
Mesh/Mesh control/Default Size
In Default Mesh Size dialog box UnCheck Set Element Size on Next Use
Set Element Size: 2
Set Minimum Number of Elements: 1
Click OK

View/ Options
In View Options dialog box Select
Category: Labels, Entities and Color
Options: Curve – Mesh Size
Show As: 2..Symbols (all curves)

Options: Curve- Surface Directions
Parametric Directions: 2..Show Curve Arrows
Click OK

Mesh/ Mesh Control/ Size Along Curve
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Pick curve 2 on the graphics window, then Click OK
In Mesh Size Along Curve dialog box
Number of Elements: 30
Select Noda Spacing: Biased
Bias Factor: 5
Select Small Elements at End
Click OK
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Pick curve 3 on the graphics window, then Click OK
In Mesh Size Along Curve dialog box
Number of Elements: 10
Select Node Spacing: Biased
Bias Factor: 5
Select Small Elements at Start
Click OK
In Entity Selection – Select Curve(s) to Set Mesh Size dialog box
Pick curve 7 on the graphics window, then Click OK
In Mesh Size Along Curve dialog box Set
Element size: 0,2
Click OK, then Cancel

Mesh/Geometry/Surface
In Entity Selection – Select Surfaces to Mesh dialog box
Select the defined boundary surface, then Click OK
In Automesh Surfaces dialog box Set
Property: 1..S235 PLATE
Mesher: Triangles
Click OK
The graphic window should look like this:

Applying Load
//The end part of the specimen is fixed. The reality can be approximated when prescribed displacement is applied. The loading is given in 10 steps.//
Model/ Function
In Function Definition dialog box Set
Title: Load vs. Time
Type: 1..vs. Time
Data Entry: X=0; Y=0, then Click Add
Data Entry: X=10; Y=10, Click Add, then OK

Model/Load/On Curve
In New Load Set dialog box set
Title: loading
Click OK

In Entity Selection-Enter Curve(s) to Select dialog box
Pick curve 5
Click OK
In Create Loads on Curves dialog box choose Displacement and set
Load Value: Pick TY=0,01
Load Time/ Freq Dependence: Choose 1..Load vs. Time from the drop down list
Title: Enforced Displacement
Click OK, then Cancel

Applying Constraints
Model/ Constraint/ On Curve
In New Constraint Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Curve(s) to Select dialog box
Choose curve 6, then click OK
In Create Constraints on Geometry dialog box Set
Title: X Symmetry
Select Arbitrary in CSys and 0..Basic Rectangular from the drop down list
Check TX degrees of freedom
Click Ok
In Entity Selection – Enter Curve(s) to Select dialog box
Choose curve 1, then click OK
In Create Constraints on Geometry dialog box Set
Title: Y Symmetry
Select Arbitrary in CSys and 0..Basic Rectangular from the drop down list
Check TY degrees of freedom
Click Ok
In Entity Selection – Enter Curve(s) to Select dialog box
Choose curve 5, then click OK
In Create Constraints on Geometry dialog box Set
Title: Constraint at Displacement
Select Arbitrary in CSys and 0..Basic Rectangular from the drop down list
Check TY degrees of freedom
Click Ok, then Cancel

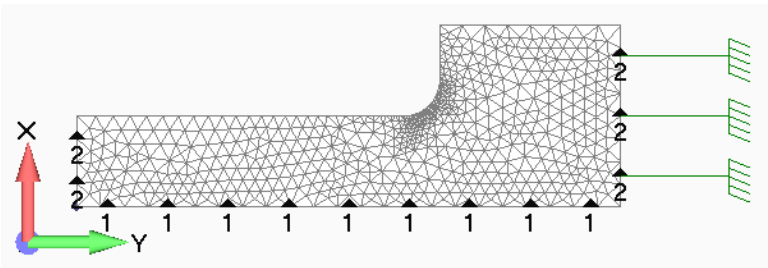


Figure 10.46. The finite element model of the specimen

Analyzing the Model
//Using the NX Nastran solver transient dynamic analysis is executed. The reason of the usage of transient dynamic analysis is that the loading of the structure is time dependent.//
Model/Analysis
In Analysis Set Manager dialog box
Click New button

In Analysis Set dialog box
Title: Direct Transient
Analysis Program: 36..NX Nastran
Analysis Type: 3..Transient Dynamic/ Time History
Click Next 6 times
In Dynamic Control Options dialog box Set
Transient Time Step Intervals:
Number of Steps: 10
Time per Step: 1
Output Interval: 1
Click Next 4 times
In Nastran Output Requests dialog box Select:
Nodal: Displacement, Applied Load, Constraint Force,
Elemental: Force, Stress, Strain
Click OK, then Analyze
When you see the following message: Cleanup of Output Set 11 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the results
View/Select
In View Select dialog box set
Deform Style: Deform
Contour Style: Contour
Click Deformed and Contour Data…
In Select PostProcessing Data dialog box set
Output Sets: 11..Case 11 Time 10
Output Vectors
Deformation: 1..Total Translation
Contour: 1.. Total Translation
Click OK all dialog boxes
View/Select
In View Select dialog box set
Deform Style: Deform
Contour Style: Contour
Click Deformed and Contour Data…
In Select PostProcessing Data dialog box set
Output Set: 11..Case 11 Time 10
Output Vectors
Deformation: 1..Total Translation
Contour: 7033..Plate Top Von Mises Stress
Click OK all dialog boxes

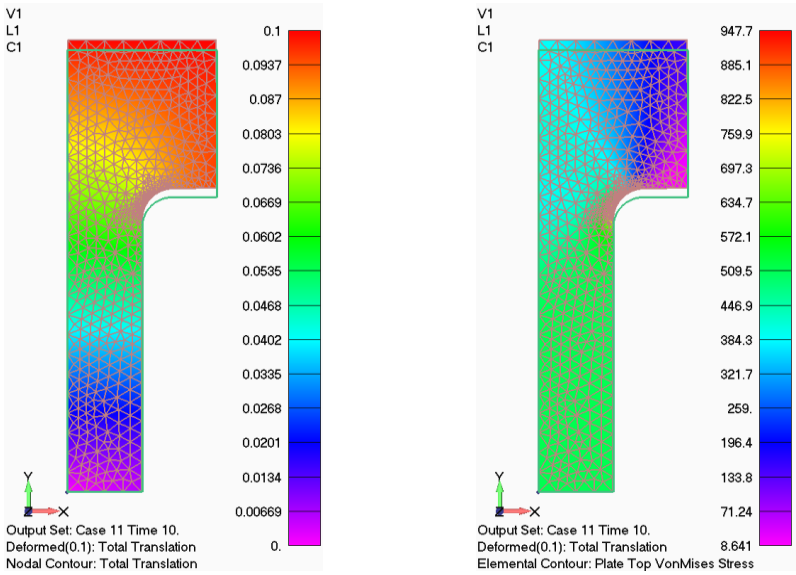


Figure 10.47. Total translation and stress distribution of the specimen

View/Select
In View Select dialog box set
Deform Style: None- Model Only
Contour Style: None- Model Only
Click OK
Tools/ Charting
In Charting dialog box Click to Data Series Manager Button
In Chart Data Series Manager dialog box
Click to New Data Series Button
In Chart Data Series dialog box Set
Title: Stress vs. Time step
Data Type: 2..XY vs. Set Value
UnCheck: Use All Output Sets
Start: 1..Case 1 Time 0.
End: 11..Case 11 Time 10.
Vector: Plate Top Von Mises Stress
Click to Location field: 849 (Select element 849)
Click Ok then Cancel
In Chart Data Series Manager dialog box Click Done

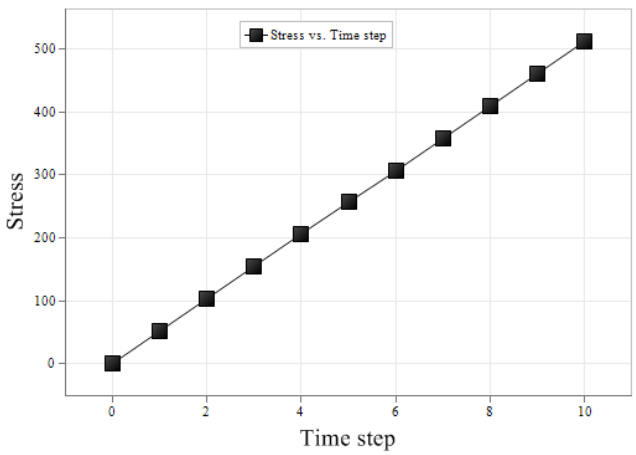


Figure 10.48. Stress vs. time

Numerical example 11. (Analysis of an Assembly)
In Figure 10.49. a simple assembly can be seen. The cover is fixed to the part with bolts and nuts. The bolts are pre-stressed. The preparation of the finite element model of the problem and the analysis is presented. The symmetry property of the parts and boundary conditions are used to simplify the model.

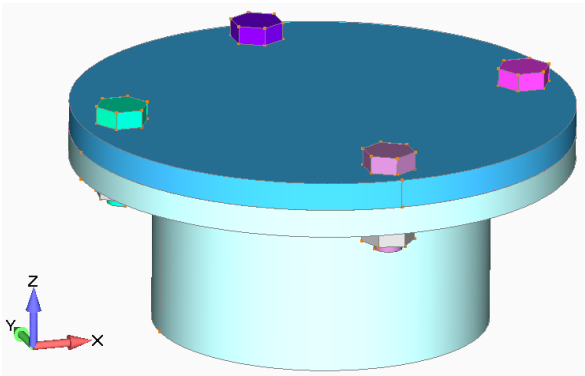


Figure10.49. The assembled parts

<u>Data:</u>
Internal pressure: $p = 100\ bar$
Thickness: $t = 10\ mm$
$E = 206000\ MPa$
$\nu = 0{,}3$
The bolt preload: $F = 1000N$

Importing the Model Geometry
File/Import/ Geometry
In Geometry File to Import dialog box:
Go to the Examples directory and open assembly.stp file
In STEP Read Options dialog box:
Check the Geometry Scale Factor: 1000
Assembly Options:
Check Increment Layer, Increment Colour
Click OK

Tools/Measure/Distance
In Locate – Define Location to Measure From dialog box
Click Methods Button, and Select One Point (or press CTRL+P key when the cursor in the X, Y or Z field)
On Point – Define Location to Measure From dialog box
Point ID: 138, Click OK
On Point – Define Location to Measure From dialog box
Point ID: 137, Click OK then Cancel
//The measured distance data can be read in the Message window. Distance: 10.//

Preparing the Model to Mesh
//Because of the symmetry in geometry and boundary conditions the key geometry is a quarter model of the assembly.//
Geometry/ Point
In Locate – Enter Coordinates or Select with Cursor dialog box

Click to Methods and Select Along Curve
In Along Curve – Enter Coordinates or Select with Cursor dialog box Set Curve ID: 247; Along: 25%
Click Ok
In Along Curve – Enter Coordinates or Select with Cursor dialog box Set Curve ID: 247; Along: 25%; From End Near: Select the Line other End
Click Ok
In Along Curve – Enter Coordinates or Select with Cursor dialog box Set Curve ID: 248; Along: 25%
Click Ok
In Along Curve – Enter Coordinates or Select with Cursor dialog box Set Curve ID: 248; Along: 25%; From End Near: Select the Line other End
Click Ok
In Along Curve – Enter Coordinates or Select with Cursor dialog box Set Curve ID: 258; Along: 25%
Click Ok
In Along Curve – Enter Coordinates or Select with Cursor dialog box Set Curve ID: 258; Along: 25%; From End Near: Select the Line other End
Click Ok, then Cancel
Geometry/ Solid/ Slice
In Entity Selection – Select Solid to Slice dialog box
Click to Select All Button, then Click OK
In Plane Locate - Specify Plane for Intersection dialog box
Click Methods Button and Select Points
In Plane Points –Specify Plane for Intersection dialog box
Base Point ID: 182; Plane Point 1: 180; Plane Point 2: 177, Click OK
On Model info panel UnCheck:
Geometry: 11..Untitled and 12..Untitled
Geometry/ Solid/ Slice
In Entity Selection – Select Solid to Slice dialog box
Add ID:9 and 10, then Click OK
In Plane Points –Specify Plane for Intersection dialog box
Base Point ID: 181; Plane Point 1: 179; Plane Point 2: 178, Click OK
On Model info panel UnCheck:Unused Geometry

The graphic window should look like this:

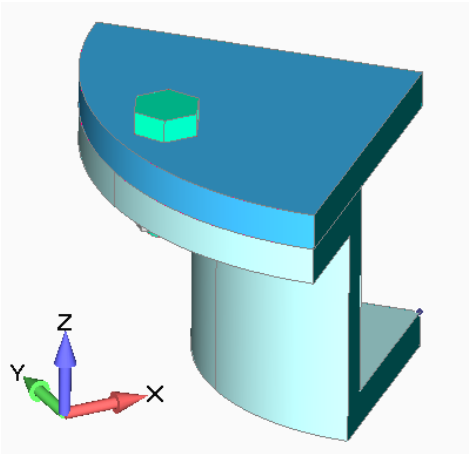


Figure 10.50. The quarter model prepared for the analysis

Defining the Material
Model/Material
In Define Material – ISOTROPIC dialog box
Click Type
In Material Type dialog box choose Isotropic
Click OK
In Define Material – ISOTROPIC dialog box set
Title: Steel
Young Modulus, E: 2,06E5
Poisson's Ratio, nu: 0,3
Click OK (material 1 created), then Cancel

Defining the Property
//The solid parts are meshed with volume elements so element property is defined for this case.//
In Define Property – PLATE Element Type dialog box
Click Elem/Property Type
In Element/Property Type dialog box Select
Volume elements: Solid, then click OK
In Define Property – SOLID Element Type dialog box Set
Title: S235 SOLID
Material: 1..Steel
Click OK (Property 1 created), then Cancel

Meshing the Model
//During the meshing two types of volume elements are used. For sheet models hexahedron elements are recommended to use so the running time can be decreased.//
In Model Info panel Click with Left Mouse button to the Layers
Select View Visible Layers Only
Check 10..vessel
Mesh/Mesh control/Size On Solid
In Entity Selection – Select Solid(s) to Set Mesh Size dialog box
Add ID: 10, then Select OK
In Automatic Mesh Sizing dialog box Set
Element Size: 2,5
Min Elements on Edge: 4
Click OK
Mesh/Geometry/ Solids
In Entity Selection – Select Solid(s) to Mesh dialog box
Add ID: 10, then Click Ok
In Automesh Solids dialog box Select
Property: 1..S235SOLID
UnCheck: Midsides Nodes
Click OK

In Model Info panel
Check Visibility and Activate 9..cover
Mesh/Mesh control/Size On Solid
In Entity Selection – Select Solid(s) to Set Mesh Size dialog box

Add ID: 9, then Click OK
In Automatic Mesh Sizing dialog box Set
Size for: Hex Meshing
Element Size: 2,5
Min Elements on Edge: 4
Click OK
//If the part changes its colour to blue “hexmesh” can be used.//

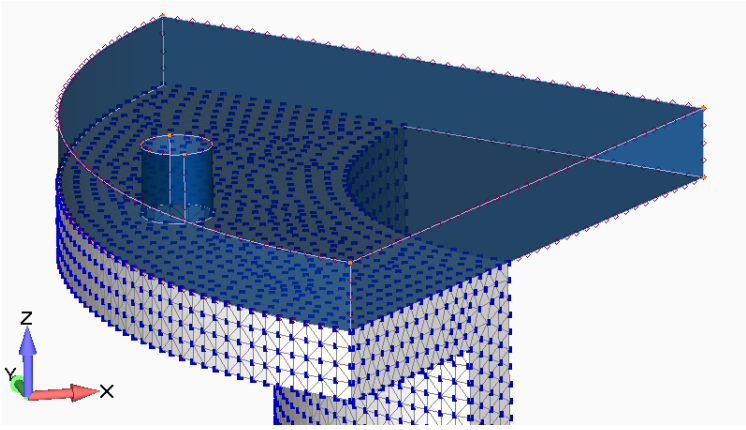


Figure 10.51. The cover is ready for the hexmesh

Mesh/Geometry/ HexMesh Solids
In Entity Selection – Select Solid(s) to Mesh dialog box
Add ID: 9, then Click Ok
In Hex Mesh Solids dialog box Select
Property: 1..S235SOLID
UnCheck: Midsides Nodes
Click OK

In Model Info panel
Check Visibility and Activate 6..bolt
Check Visibility: 2..nut
UnCheck Visibility: 9..cover and 10..vessel

Mesh/Mesh control/Size On Solid
In Entity Selection – Select Solid(s) to Set Mech Size dialog box
Add ID: 6 and 2, then Select OK
In Automatic Mesh Sizing dialog box Set
Size for: Hex Meshing
Click OK

Geometry/ Solid/ Slice
In Entity Selection – Select Solid to Slice dialog box
Add ID: 6, then Click OK
In Plane Points - Specify Plane for Intersection dialog box
Base Point ID: 90; Plane Point 1: 87; Plane Point 2: 189, Click OK

Mesh/Mesh control/Size On Solid
In Entity Selection – Select Solid(s) to Set Mech Size dialog box
Add ID: 6,2 and 15, then Click OK

In Automatic Mesh Sizing dialog box Set
Size for: Hex Meshing
Click OK

Mesh/Geometry/ HexMesh Solids
In Entity Selection – Select Solid(s) to Mesh dialog box
Add ID: 6,2 and 15, then Click Ok
In Hex Mesh Solids dialog box Select
Property: 1..S235SOLID
UnCheck: Midsides Nodes
Click OK

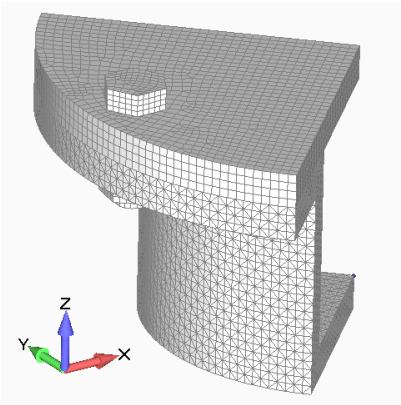


Figure 10.52. The meshed parts

Creating Contact
The bolt is meshed with two type of loads but it is one part, so glued connection has to be set. Between the vessel, the cover, the bolt and the nuts contact is necessary to be defined.//
Connect/ Automatic
In Entity Selection – Select Solid(s) to Detect Connections dialog box
Add ID: 6,15 and 2; then Click OK
In Auto Detection Options for Connections dialog box
Connection Property: Glued
Click OK
Add ID: 9,10; then Click OK
In Auto Detection Options for Connections dialog box
Connection Property: Contact
Click OK, then Cancel

Connect/ Connection Region
In Connection region dialog box Set
Title: bolt top
Add Surface: 57, 53 and 52
Click OK
Connect/ Connection Region
In Connection region dialog box Set
Title: top
Add Surface: 85, 91 and 96
Click OK

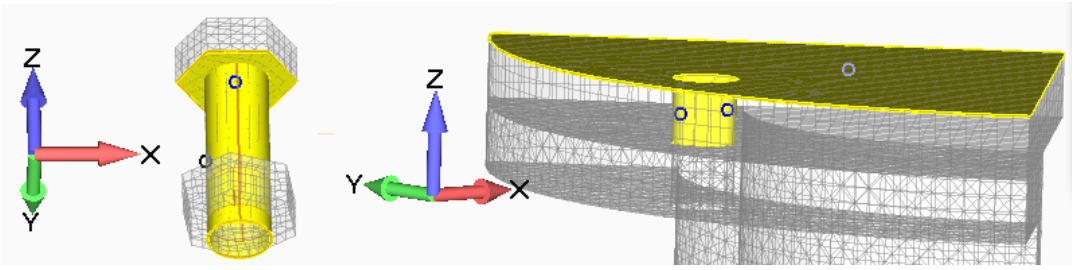


Figure 10.53. The contact regions (top)

Connect/ Connection Region
In Connection region dialog box Set
Title: bolt bottom
Add Surface: 12, 53 and 52
Click OK
Connect/ Connection Region
In Connection region dialog box Set
Title: bottom
Add Surface: 102, 106 and 108
Click OK, then Cancel

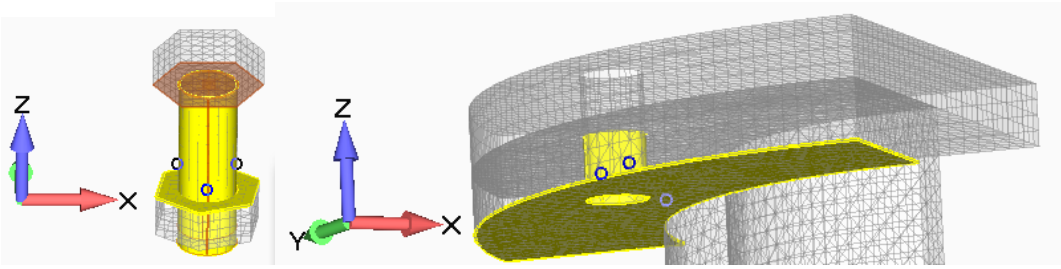


Figure 10.54. The contact region (bottom)

Connect/ Connector
In Define Contact Connector – Select Connection Regions dialog box Set
Title: top
Property: 2..Untitled
Master (Target): 7..bolt top
Slave (Source): 9..top
Click Ok (Connector 4 Created)
Title: bottom
Property: 2..Untitled
Master (Target): 8..bolt bottom
Slave (Source): 10..bottom
Click Ok, then Cancel

Applying Load
Model/ Load/ Elemental
In New Load Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Element(s) to Select dialog box
Click to Pick Button, and Select Around Point
In Along Curve – Specify Location to Select Around dialog box Set
Point ID: 215, Click OK

In Select by Distance From Point dialog box Set Distance Specification
Pick Closer Than
Min: 50, Click Ok
In Entity Selection – Enter Element(s) to Select dialog box Click Ok
in Create Loads on Elements dialog box Set
Title: Pressure on the cover
Select Pressure
Direction: Normal to Element Face
Set Pressure Value to: 10, Click OK
In Face Selection dialog box
Method: Adjacent Faces
Click in Face field, and pick an element
Tolerance: 20
Select Front Face, Click OK, then Cancel

Model/ Load/ On Surface
in Entity Selection – Enter Surfaces(s) to Select dialog box Add
ID: 100, 109, 110
In Create Loads on Surface dialog box Set
Title: Pressure inside the vessel
Select Pressure
Direction: Normal to Element Face
Set Pressure Value to: 10, Click OK

Applying Constraints
Model/ Constraint/ On Surface
In New Constraint Set dialog box
Title: (give a title), then click OK
In Entity Selection – Enter Surface(s) to Select dialog box Add
Surface ID: 115, 121, Click OK
In Create Constraints on Geometry dialog box Set
Title: Symmetry1
Advanved Type: Select Surface
Check Allow Sliding only along Surface (Symmetry)
Click Ok
In Entity Selection – Enter Surface(s) to Select dialog box Add
Surface ID: 133, 139, Click OK
In Create Constraints on Geometry dialog box Set
Title: Symmetry2
Advanved Type: Select Surface
Check Allow Sliding only along Surface (Symmetry)
Click Ok
In Entity Selection – Enter Surface(s) to Select dialog box Add
Surface ID: 113, Click OK
In Create Constraints on Geometry dialog box Set
Title: TZ
Select Arbitrary in CSys and 0..Basic Rectangular from the drop down list
Check TZ degree of freedom
Click Ok, then Cancel

Creating Bolt Preload
Hit F8 Key, Select ZX Front, then Click Ok
Group/ Create/Manage
In Group Manager dialog box Click to New Group Button
In New Group dialog box Set Title: Bolt Group, Click Ok, then Done
Group/ Node/ ID
In Entity Selection – Select Node(s) for Group dialog box Click to Pick Button
Select Box from the drop down list, choose element according to the Figure 10.55.
Click OK

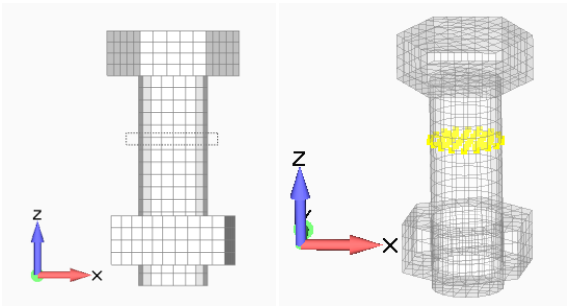


Figure 10.55. A step of the bolt preload definition

Connect/ Bolt Region
In Bolt Region dialog box Set
Title: Bolt Region
Bolt Type: Solid
Click to Multiple Button
In Entity Selection – Enter Node(s) to Select dialog box
Select Group: 1..Bolt Region from the drop down list, Click Ok
In Bolt Region dialog box Set
Bolt Axis Direction: 3..Z, Click OK
Model/ Load/ Bolt Preload
In Create Bolt Preload dialog box Set
Title: Bolt Preload Load
Preload Value: 1000
Pick Bolt Region(s)
Click OK
In Entity Selection – Select Region(s) for Preload dialog box
Click More Button, Click OK, then Cancel

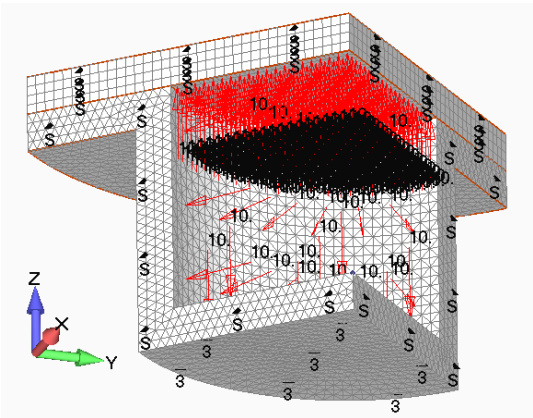


Figure 10.56. The finite element model of the assembly

Analyzing the Model
Model/Analysis
In Analysis Set Manager dialog box
Click New button
In Analysis Set dialog box
Title: Linear static analysis
Analysis Program: 36..NX Nastran
Analysis Type: 1..Static
Click OK, then analyze (In Analysis Set Manager dialog box)
When you see the following message: Cleanup of Output Set 1 is Complete, close the NX Nastran Analysis Monitor

Postprocessing the Results
//Now we represent the total translation then the stress distribution.//
View/Select (or hit F5 button)
In View Select dialog box
Deformed Style: Deform
Contour Style: Contour
Click to Deformed and Contour Data…
In Select PostProcessing Data dialog box
Output Sets: NX NASTRAN Case 1
Deform: 1..Total Translation
Contour: 1..Total Translation
Click OK all dialog boxes
Click Post Options and Select Actual Deformation
Select Scale Deformation
In Deformation Scale dialog box
Actual: 20, Click OK
View/Select (or hit F5 button)
In View Select dialog box
Click to Deformed and Contour Data…
In Select PostProcessing Data dialog box
Deform: 1..Total Translation
Contour: 60031..Solid Von Mises Stress
Click OK all dialob boxes

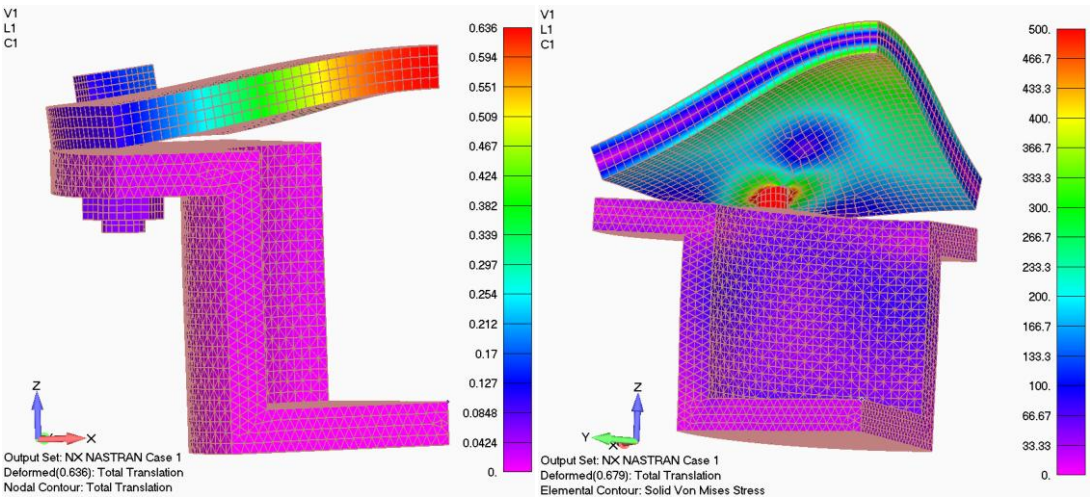


Figure 10.57. Total translation and the stress distribution of the assembly

REFERENCES

BATHE K.J.: Finite Element Procedures, Prentice-Hall International, New Jersey, USA, p. 1037, 1996.

COOK R.D.: Finite Element Modeling for Stress Analysis, John Wiley & Sons, USA, p. 330, 1995.

ÉGERT J., KEPPLER I.: A végeselem módszer mechanikai alapjai, Universitas-Győr Nonprofit Kft., Győr, Hungary, p. 210, 2007.

HUTTON D.V.: Fundamentals of Finite Element Analysis, McGraw-Hill, New York, USA, p. 505, 2004.

KOVÁCS Á., MOHAROS I., OLDAL I., SZEKRÉNYES A.: Finite Element Methode, Typotex Kiadó, Budapest, Hungary, p. 378, 2012.

KOVÁCS Á., MOHAROS I., OLDAL I., SZEKRÉNYES A.: Végeselem-módszer, Typotex Kiadó, Budapest, Hungary, p. 383, 2011.

LIU G.R., QUEK S.S.: The Finite Element Method – A Practical Course, Butterworth-Heinemann, London, England, p. 361, 2003.

LIU Y.: Introduction to the Finite Element Method (Lecture Notes), University of Cincinnati, USA, p. 188, 2003.

NIKISHKOV G.P.: Introduction to the Finite Element Method (Lecture Notes), University of Aizu, Japan, p. 45, 2004.

PÁCZELT I.: A végeselem-módszer alapjai, Miskolci Egyetem, Miskolc, Hungary, p. 94, 1993.

RAO S.S.: The Finite Element Method in Engineering, 4th edition, Elsevier Science & Technology Books, p. 658, 2004.

SADD M.H.: Elasticity – Theory, Applications, and Numerics, Butterworth-Heinemann, Burlington, USA, p. 474, 2005.

SZABÓ T.: Alkalmazott mechanika – Végeselem-módszer alkalmazásai, Miskolci Egyetem, Miskolc, Hungary, p. 81, 2003.

UGS: Femap User Guide Version 9.2, UGS Corporation, USA

ZIENKIEWICZ O.C., TAYLOR R.L.: The Finite Element Method – The Basis, 5th edition, Butterworth-Heinemann, London, England, p. 708, 2000.

ZIENKIEWICZ O.C., TAYLOR R.L.: The Finite Element Method – Solid Mechanics, 5th edition, Butterworth-Heinemann, London, England, p. 477, 2000.

SZÉCHENYI 2020

HUNGARIAN GOVERNMENT

European Union
European Social Fund

INVESTING IN YOUR FUTURE